COSMOS/M CAD Interface® User Guide
First Edition
Version 2.0
December 1997

Copyright

This software product is copyrighted and all rights are reserved by Structural Research and Analysis Corporation. Copyright© 1995 - 1996 Structural Research and Analysis Corporation. All Rights Reserved.

The distribution and sale of this product (COSMOS/M CAD Interface, Version 2.0) is intended for the use of the original purchaser only and for use only on the computer system specified. The software product may be used only under the provisions of the license agreement that accompanies the product package.

This manual (COSMOS/M CAD Interface, Version 2.0) may not be copied, photocopied, reproduced, translated or reduced to any electronic medium or machine readable form in whole or part without prior written consent from Structural Research and Analysis Corporation. Structural Research and Analysis Corporation makes no warranty that COSMOS/M CAD Interface is free from errors or defects and assumes no liability for the program. Structural Research and Analysis Corporation disclaims any express warranty or fitness for any intended use or purpose. You are legally accountable for any violation of the License Agreement or of copyright or trademark. You have no rights to alter the software or printed materials.

The COSMOS/M CAD Interface program is constantly being developed, modified and checked and any known errors should be reported to Structural Research and Analysis Corporation.

Disclaimer

The authors have taken due care in preparing this manual and the examples presented herein. In no event shall SRAC assume any liability or responsibility to any person or company for direct or indirect damage resulting from the use of the information contained herein or any discrepancies between this information and the actual operation of the software.

Licenses & Trademarks

Use by Structural Research and Analysis Corporation of ANSYS Input Commands and Command Structure herein is licensed under agreement with Swanson Analysis Systems, Inc. All rights reserved. COSMOS/M®, COSMOS/M CAD Interface, and COSMOS® are registered trademarks of Structural Research and Analysis Corporation. All other COSMOS/M module names are trademarks of Structural Research and Analysis Corporation.

ABAQUS is the registered trademark of Hibbitt, Karlsson & Sorensen, Inc. ANSYS is a registered trademark of Swanson Analysis Systems. AutoCAD is registered in the U.S. Patent and Trademark Office by Autodesk, Inc. DXF and AutoSolid are the registered trademarks of Autodesk, Inc. CADKEY is the registered trademark of CADKEY, Inc. DECStation is the registered trademark of Digital Equipment Corporation. EPSON is the registered trademark of Epson Computers. HP is the registered trademark of Hewlett-Packard. IBM is the registered trademark of International Business Machines Corporation. MicroStation and MicroStation Modeler are the registered trademarks of Bentley Systems. MS-DOS is the registered trademark of Microsoft Corporation. MSC/NASTRAN is the registered trademark of MacNeal-Schwender Corp. PATRAN is the registered trademark of PDA Engineering. PostScript is the registered trademark of Adobe Systems, Inc. SINDA/G is the registered trademark of Network Analysis Associates, Inc. Sun is the registered trademark of Sun Microsystems, Inc. SGI is the trademark of Silicon Graphics, Inc. WORDSTAR is the registered trademark of Micropro International Corporation. All other trade names mentioned are trademarks or registered trademarks of their respective owners.
Preface

About This Manual...

Structural Research and Analysis Corporation (SRAC) has developed the COSMOS/M CAD Interface program to give design engineers working with various CAD programs access to the state-of-the-art technology in Finite Element Analysis (FEA).

This manual is intended to be a complete, stand-alone document for installing and using the COSMOS/M CAD Interface program, in addition to providing an introduction to FEA and related topics. Basic knowledge of the intended CAD programs, hardware and operating system is assumed.

Related COSMOS/M Documents

In addition to capabilities directly supported by the interface, COSMOS/M CAD Interface provides a silent full access to version 2.0 of the COSMOS/M FEA System. The following documents are recommended as references but are required if access to the capabilities of the silent version is to be made.

- COSMOS/M Finite Element Analysis System User Guide
- COSMOS/M Finite Element Analysis System Command Reference
- COSMOS/M Finite Element Analysis System Basic System User Guide
- COSMOS/M CAD Interface Advanced Modules Manual
- COSMOS/M ENGINEER Optimization Manual
- Theoretical Manual for COSMOS/M Finite Element Analysis System

Manual Organization

After the installation instructions, the material in this manual is divided into seven chapters and five appendices as follows. The chapters and appendices are functionally independent and may be read in any order.

Chapter 1: Introduction: presents an overview of COSMOS/M CAD Interface and a brief introduction to some fundamental aspects of computer aided design (CAD) and mechanical computer aided engineering (MCAE).

Chapter 2: Getting Started: describes the COSMOS/M CAD Interface screen and provides an early start on the use of the program through an illustrating step-by-step example.

Chapter 3: Exploring COSMOS/M CAD Interface: illustrates the steps to be followed for model building and analysis, and explores all the menus of the COSMOS/M CAD Interface. Helpful notes and comments are also provided for proper understanding of the various utilities.

Chapter 4: Element Library: this chapter provides an introduction to element definition in COSMOS/M and a full description of all elements used in the COSMOS/M CAD Interface. In addition, a brief description for all other elements used in the full COSMOS/M FEA system is provided. Detailed descriptions of
elements used in the nonlinear structural analysis is provided in the COSMOS/M Advanced Modules Manual.

Chapter 5  **Command Reference**: provides full description of all COSMOS/M CAD Interface menus and commands in hierarchical order.

Chapter 6  **Examples**: presents more examples on the use of COSMOS/M CAD Interface for various types of analyses and features. Examples on the nonlinear and dynamic structural analyses are provided in the COSMOS/M CAD Interface Advanced Modules Manual.

Chapter 7  **About Finite Elements**: this chapter presents some fundamental concepts on finite element modeling (FEM) and finite element analysis (FEA). The chapter is intended as an introduction for users who have no or little knowledge of the finite element analysis method.

Appendix A  **Glossary and Acronyms**: briefly explains common acronyms and terminology used in the FEA technology.

Appendix B  **Units**: this appendix presents useful notes on units and provides tables of consistent units in various unit systems.

Appendix C  **Material Library**: lists the contents of COSMOS/M material library.

Appendix D  **GEOSTAR Mode Commands**: presents brief description of all GEOSTAR commands.
Introduction

This installation guide provides information for installing and operating the COSMOS/M and COSMOS/M CAD Interface software on Unix platforms only. Please refer to the "COSMOS/M 2.0 Getting Started" book for installation of the Windows version of the software. For an updated version of this procedure, please refer to the Getting Started manual.

Detailed information about the network license is covered under the heading Licensing Scheme, which contains a description of the new licensing options and introduces the terminology to be used in the rest of the document. The next section, Key Components of the License Scheme, contains detailed, end-user relevant descriptions of the different components that work together to provide the network-wide floating license services. The section, Location of the License File, addresses some of the issues related to the location of the license file, which is a key component of the whole license mechanism. Format of the License File explains the format of the license file and identifies user definable fields in this file. The final section describes features of the licensing scheme which can be customized by the user.

For standalone systems, the user can refer directly to Program Installation, which provides a step by step installation procedure. Information on X Window systems is covered in the last section, Special Notes for the X Window System Version. A section on Performance Issues is located at the end.

Throughout this installation guide, text type is used to convey the source of information and to highlight important information. The following are the standard text conventions in this guide:

<table>
<thead>
<tr>
<th>Type</th>
<th>Appearance</th>
<th>Example</th>
</tr>
</thead>
<tbody>
<tr>
<td>Computer prompts</td>
<td>LIGHT</td>
<td>%</td>
</tr>
<tr>
<td>Commands you type</td>
<td>Bold</td>
<td>% tar</td>
</tr>
</tbody>
</table>

Computer Systems

Software

One of the following workstation computers with the listed operating system version (or later) and additional software is supported. All systems require that the X-windows display be active and that the C-shell be installed on the system.

<table>
<thead>
<tr>
<th>Platform</th>
<th>OS Version</th>
<th>Additional Software</th>
</tr>
</thead>
<tbody>
<tr>
<td>Sun SPARCstation</td>
<td>SunOS Version 4.1.3</td>
<td></td>
</tr>
<tr>
<td></td>
<td>SOLARIS Version 2.2</td>
<td></td>
</tr>
<tr>
<td>HP 9000/700</td>
<td>HP-UX Version 9.01</td>
<td></td>
</tr>
<tr>
<td>SGI Computers</td>
<td>IRIX Version 4.0.5H</td>
<td></td>
</tr>
<tr>
<td>IBM RS/6000</td>
<td>AIX Version 3.2.4</td>
<td>XLFRETE 02.03.0000.0000</td>
</tr>
<tr>
<td></td>
<td></td>
<td>(Fortran Run Time Environment)</td>
</tr>
<tr>
<td>Digital ALPHA</td>
<td>OSF/1 2.0B</td>
<td></td>
</tr>
</tbody>
</table>
Hardware

The following resources are required:

- A minimum of 32MB of memory, with a minimum of 64MB if you run optimization within ENGINEER. At least 64MB of swap is required. However, you should always have at least twice the physical RAM memory in swap area.

- A hard disk with at least 50MB for program storage and 150MB available for running COSMOS/M problems (for small to moderate size problems).

- Any standard color or grayscale monitor for the above graphic workstations.

- A mouse.

- For the 3D Dynamic Viewing option, the following additional hardware is required:
  - HP 9000/700:
    Graphics system with at least 8/8 plane double buffering capability. The minimum graphics configuration is a CRX24 system.
  - SGI:
    XL, XZ, Extreme, Reality Engine Graphics.

- Plotters are optional. The supported plotters are:
  - HP 7475
  - HP 7550
  - ColorPro
  - DraftMaster
  - Any plotter which supports HPGL output

- Printers are optional. The printers supported are:
  - HP PaintJet
  - HP LaserJet
  - HP DeskJet
  - Any printer which supports PostScript or HPGL output

Licensing Scheme

COSMOS/M is now available with a new licensing mechanism, which uses Globetrotter Software’s FLEXlm license manager (version 4.1) to perform security.

While the previous COSMOS/M licensing scheme was a per-computer based or node locked scheme, the new licensing scheme allows COSMOS/M to be licensed on a concurrent usage basis among users on a heterogeneous network. The following are the features of the licensing mechanism available with the purchase of the Network Version of COSMOS/M:

1. A network wide floating license which allows a specified number of invocations of COSMOS/M from any machine in the network. Users trying to access COSMOS/M after
the limit on the maximum number of users has been reached would not be able to run COSMOS/M, but instead would see a descriptive message on their screen.

2. An independent number of users for different modules of COSMOS/M. For example, five copies of GEOSTAR and two each for the various analysis modules could be licensed at a site where there are more users involved with model building than there are users running analyses.

3. A combination of floating and node-locked licenses under which concurrent usage can be made available only from selected hosts.

4. A single licensing mechanism for both standalone systems as well as a network of hosts.

5. Redundant server host support to increase reliability and reduce down-time.

6. Graceful program termination when license daemon death is detected due to a failure in network connectivity or the shutdown of the license server. The model database is completely saved before program termination.

The scheme involves one or more license servers. A license server is a machine which contains the license information and runs the required processes to implement and perform housekeeping tasks for the license mechanism.

The license server(s) may have a different architecture from that of the hosts which would run COSMOS/M. In addition, the license server could be any machine in the network and, in particular, may be different from the file server which actually stores the COSMOS/M files. For a standalone system, the file server, license server and client running COSMOS/M are all one and the same.

The COSMOS/M kit includes a license request form which must be filled out by the user and sent back to Structural Research. This form contains instructions to allow the user to furnish the requested site-specific information. Once the license request form is received, Structural Research will provide the user with a compressed version of the license file which needs to be present on your license server and the client machines running COSMOS/M.

**Key Components of the License Scheme**

The key components of the new license scheme are the following:

1. Two license daemons ("lmgrd" and "SRAC").
2. The license file.
3. FLEXlm Libraries linked with individual COSMOS/M modules.
4. The "configcosmos" script.

The license daemons are processes which need to be running on the license server. These are typically started at boot time by adding an appropriate line to one of the "rc" scripts in the "/etc" directory. The license daemons are supplied along with the COSMOS/M executables. Only the "lmgrd" daemon needs to be invoked, since it will start up the "SRAC" daemon. If the license daemons are not running when the COSMOS/M modules are executed, the program will terminate after displaying a status alert. The daemons can be invoked by any user although multiple copies are not allowed on the same server.
The license file is the crucial link between the license daemons and the COSMOS/M modules and it contains license information such as modules available, number of concurrent users allowed and expiration date, if any.

The program `configcosmos` supplied with the COSMOS/M package creates the license file after reading the compressed license provided by Structural Research.

**Location of the License File**

Both the license daemons and the COSMOS/M modules read the license file. This file must be accessible to both the daemons and the COSMOS/M modules. The COSMOS/M modules read the license file typically from the COSMOS/M directory. However, the license daemons, which are invoked at boot time from the license server, may not have access to the COSMOS/M directory at that time. To solve this problem, a copy of the license file should be placed in a file system which is already mounted when the daemons are invoked. Structural Research recommends the "/etc" directory on the license server as an ideal location.

The location of the license file is indicated to the daemons through a command line switch in the "lmgrd" command when it is invoked. The following is a C shell syntax.

**Syntax:**

```bash
lmgrd -c <licenseFileLocation> >& logFile
```

**Example:**

```bash
lmgrd -c "/etc/license.dat" >& /usr/spool/license.log
```

For the Bourne shell, use:

```bash
lmgrd -c /etc/license.dat > /usr/local/license.log 2>&1
```

Be sure to run the process in the background (by appending an "&" symbol after the command) if you are invoking it from any place other than the "rc" scripts.

The COSMOS/M modules look for the license file in a location pointed to by the "LM_LICENSE_FILE" environment variable. This is typically set as "<<COSMOS/M mount point>/license.dat". If this variable is not set then the program runs as a limited version in which some of the modules run with a model size limit of 100 nodes.

If your site already has an application from another vendor using the FLEXlm license mechanism, then you need to be careful about choosing the location of the license file. There are two cases:

**Case 1: All products use the same license server(s).**

In this case, the solution is simple: combine license files by taking the set of SERVER lines from any one license file and add all the DAEMON, FEATURE and FEATURESET lines from all the license files. The combined file can be located at any convenient location or multiple copies can be located at fixed locations as required by the various applications.
Case 2: The products use different license server node(s).

In this case, separate license files will be required, one for each distinct set of license servers. (Where multiple products use the same set of license server nodes, the technique described above can be used to combine their license files). The resulting multiple license files can be installed at convenient locations and the LM_LICENSE_FILE environment can be set as follows:

\[
\text{setenv LM_LICENSE_FILE "lfpath1:lfpath2:.....lfpathN"}
\]

The \textit{configcosmos} script generates the license file and also prints out precise instructions for completing the installation of COSMOS/M.

The IBM RS/6000 and HP9000/700 series computers use the Korn shell to execute the boot scripts, whereas most other Unix systems use the Bourne shell. This could cause a problem in starting up the daemons at boot time. If they are not running, you can try altering the startup command to:

\[
\begin{align*}
\text{(trap ' ' 1 2 15 ; /etc/lmgrd -c /etc/license.dat > \} \\
&\text{/etc/license.log 2>&1 ) &}
\end{align*}
\]

This traps signals that cause the daemons to die. Put this line at the end of your /etc/rc.nfs file. (Thanks to Larry Kelley at Magnavox for his help.)

Program Installation

1. Identify machines in your network which will play the roles of the file server, the license server and the clients.

   The file server is the machine which provides disk space to store the program. The license server is one which will run the daemon programs to provide the licensing services. The clients are machines which will actually run COSMOS/M.

   All clients must be of the same architecture to run the same executables. However, it is still possible to have homogeneous sets of clients in a heterogeneous network all sharing a single license server.

   The file server can be any machine which can serve as an NFS server.

   The license server is currently restricted to belong to one of the following classes of machines:

   1. IBM RS/6000
   2. Sun 4
   3. Silicon Graphics
   4. HP 9000/700
   5. DEC alpha/OSF

   If you do not have any of the machines listed above, you cannot use the network license manager. You will use the older, node-locked security system. See "Installation Notes for Other Computers".
For a standalone system, the same machine plays the roles of file server, license server and client.

If all end-user data is on one machine, then there is no need for redundant servers. Globetrotter Software and Structural Research both recommend the use of a single license server.

On the other hand, if the end-user's data is split among two or more server nodes and work is still possible when one of the nodes goes down or off the network, then multiple server nodes can be employed. Only in very volatile situations or in very large networks should more than three server nodes be required. The system remains fully functional as long as a simple majority of the servers are running. In all cases, an effort should be made to select stable systems which are not frequently shut down as server nodes.

A single server setup is sufficient to support up to 6400 users. The only advantage of the multi server option is that it reduces "off-line" time for the program where it is still possible to work when one of the servers is shut down.

2. Load the COSMOS/M files from the CD onto the file server at a location which is exportable to all your clients running COSMOS/M and preferably also to the license server. If COSMOS/M directory already exists, delete this directory prior to loading the COSMOS/M files from the CD. For mounting the CD and installing COSMOS/M, follow the instructions on the sheet enclosed with the CD.

3. Obtain the machine identification number(s) of the license server(s).

This information is necessary for Structural Research to supply you with the license codes. This number is different for different architectures. COSMOS/M uses the Ethernet address for HP 9000/700 machines. The 8 digit hexadecimal "hostid" is used for SUN machines, the hexadecimal form of the "/etc/sysinfo -s" command is used on Silicon Graphics machines and part of the string printed out by the "uname -m" command is used on the IBM RS/6000 machines.

A script called "machine_id" is provided as part of the COSMOS/M package. This script runs a compiled program appropriate for your license server architecture. The "machine_id" script is present in the COSMOS/M directory. You may either copy the "machine_id" script to each of your license servers or export this directory to all the license servers.

Execute the "machine_id" script on all the license servers at your site and obtain the machine identification numbers of these machines. The program will print out a line such as:

```
The FLEXlm host ID of this machine is "0800091224c0"
```

4. Fill in the requested data on the license request form included with the COSMOS/M kit.

Fax the information back to the location indicated on the form. Structural Research will then fax you the license information specific to your site.

5. Enter the license data verbatim into a temporary file on your license server(s).

You may choose a file designated as "license.info" to type in the license data sent to you by fax. The default name and location of this file are "/tmp/license.info". The
license information faxed back to you is ASCII text and contains all the necessary information to be entered in your license file. The configcosmos script contained in the COSMOS/M directory will expand the contents of this compressed file into the actual FLEXlm license file required by COSMOS/M. Remember to enter everything provided on the license authorization sheet.

6. Run the configcosmos script.

The license server(s), file server and all COSMOS/M clients have to be configured before you can start running COSMOS/M.

The configcosmos script performs all configuration steps required to run COSMOS/M. The same script is used to configure the license server, the file server and all COSMOS/M clients. On a standalone machine, the script will automatically run through the three configuration steps. At sites involving more than one machine, you have to run this script on each of these different machines. This script is part of the COSMOS/M package. Copy this script from the file server to each of your license servers.

7. Follow the instructions printed by the configcosmos script.

The license daemons need only be run if you have purchased the network license option. If more than one license server is involved, then the license daemons must be running on at least half (simple majority) of the servers involved. These daemons should be started at boot time by the addition of appropriate lines to one of the "rc" scripts in the "/etc" directory.

The configcosmos script prints out precise instructions that must be followed to successfully complete the COSMOS/M installation. The instructions contain site specific file names and directory path names obtained from the user.

**NOTE:** If you have a CAD program installed at your site, you will be prompted, during the running of configcosmos, for the name of the CAD program you want to use for invoking it. The CAD programs currently supported by COSMOS/M are: Pro/ENGINEER, CADDS5, UG2, BSI, and others.

8. Run your preprocessor.

At this time you have created a new command to invoke COSMOS/M, typically called "runcosmos". Try invoking the GEOSTAR preprocessor or the CAD Interface software. If a window appears ready to accept commands and it will start a new database or load an existing one, the software is correctly installed and is ready to run.

If a small window shows up with an error or it runs as a demo version, there is a problem. Make sure that the "license.info" file was entered correctly. All security codes are composed of hexadecimal characters. One error commonly seen is discussed below.

**Error:**

Cannot connect to license server.
Several problems can cause this error. The license server machine may be down, the name of the license server was entered incorrectly, or the license daemons are not running. Since both the "SRAC" and "lmgrd" programs need to be running, check them with the system command "ps". Review the "license.log" file since this may contain error messages that will give you insight into the problem. If you do find an error in your typing, kill the lmgrd daemon currently running and rerun configcosmos from the beginning.

9. Run COSMOS/M CAD Interface Directly Using the GEOSTAR or COSPRO Commands.

For those users who prefer to run COSMOS/M as before, using the GEOSTAR or COSPRO commands directly, it is very similar to prior installations. After completing the license manager installation, you can modify the .cshrc files of the users who will use COSMOS/M. The .cshrc file is read upon starting any shell, and will contain these variables that COSMOS/M needs. These values are all defined at the beginning to the command generated by configcosmos, a shell script that is in the /bin directory. The default name for the command is 'runcosmos'. This should be in your .cshrc file:

```bash
setenv COSMOSM < load point of cosmosm >
setenv CAD_EXEC_NAME < name of CAD command >
setenv LM_LICENSE_FILE $COSMOSM/license.dat
set path = ( $path $COSMOSM)
```

The first environment variable, COSMOSM, has the value of the full directory pathname where COSMOS/M is loaded on the local machine. COSMOSM is used by the preprocessors to execute the various analysis solvers.

The second, CAD_EXEC_NAME, is only applicable for customers who use the COSMOS/M CAD Interface. The value of that environment variable is the command name used to execute the CAD program at your site. CAD_EXEC_NAME allows the CAD Interface to invoke the CAD program. The last two commands are absolutely literal. LM_LICENSE_FILE is the name of your local license file. If this is not defined correctly, you will not be able to use COSMOS/M in anything but DEMO mode - 100 nodes maximum. Finally, we alter the search path so that the shell (your operating environment) will find the commands GEOSTAR and COSPRO.

**Installation Notes for Other Computers**

This section is for those computers not yet covered by the new license server. Currently, this includes the NEC 4800 workstation.

**Installation**

1. Extract Files.

   If COSMOS/M directory already exists, delete this directory prior to loading the COSMOS/M files from the CD. Follow the instructions on the sheet enclosed with the COSMOS/M CD for extracting the files. You will need to be super user for extracting files.
2. Set Up Environments.

Now you need to set up the user environments. Every user who will use COSMOS/M needs to have the following two lines in their .cshrc file.

```
setenv COSMOSM /usr/cosmosm
set path = ( $path $COSMOSM )
```

This assumes that you extracted files into the directory "/usr/cosmosm". Replace that path with the directory where you installed COSMOS/M.

COSMOS/M CAD Interface users have another environment variable: CAD_EXEC_NAME. This is the name that you use to invoke your CAD program.

This, of course assumes that you are using the C-shell. Make sure that these variables are set before continuing. You can use "source ~/.cshrc" to activate the changes in your initialization files immediately. Bourne shell users (such as root) use the following instead:

```
COSMOSM=/usr/cosmosm
PATH=$PATH:$COSMOSM
export COSMOSM PATH
```

Users of Bourne shell should modify the ".profile file" in their home directory.


Run "configcosmos" or "./configcosmos" if you do not have the current working directory in your search path. Choose "3", for regular installation. You will get a machine identification number, which you will need for your authorization string request form. Fax this back to the appropriate phone number at SRAC and we will send you a code as soon as possible.


In the COSMOS/M directory, edit a file named VALIDATE.COS and type in the string that we sent you. This string only contains the digits 0-9 (zero through nine) and characters a-f (a, b, c, d, e, and f). Once you are done, run the program "check_auth" which will check your typing. If the output indicates: "Security passed for the following modules ...", your installation is complete.

5. Run GEOSTAR.

To run as a regular user, make sure that the environment variables are set correctly as noted above. Now type GEOSTAR to see if the program starts. The user must be executing in a directory where he has permission to write a file.

6. Configuring Other Users.

Every user of GEOSTAR or CAD Interface needs to have the environment variables set properly.
Printer & Plotter Troubleshooting

It is strongly recommended that the user becomes thoroughly familiar with the procedures outlined in the operating system manuals before proceeding. Printers and plotters are used interchangeably for this discussion.

Verifying that the printer works and that no problems lie within printer installation is the first step. After checking that all cables are tightly connected, the power is on, and the printer is on-line, the user should try to print something from the operating system.

COSMOS/M Printing Procedure

When you make a COSMOS/M print request, a scratch file is first written to the local directory. Then a command is invoked to print that file. Finally, the scratch file is deleted. The main part of the printing command is in the customization file 'Cosmosm', under the resource Cosmosm*printCommand. This is an X windows resource file, documented in the section 'COSMOS/M Resources for the X Version'. This command has the following options set: no burst (also known as banner) page, and data is sent in the 'raw' mode - unfiltered (so that printer control characters will work properly). The last part of the command has a switch so that the name of your printer can be appended. The default printer name is itself defined with the resource Cosmosm*defaultHardCopyDevice. See your operating system manuals for the 'lp' or 'lpr' commands for more information.

SGI, IBM, SUN and HP Equipment

All System V equipment: (HP’s, SGI, IBM, and Suns loaded with Solaris 2.x) have some sort of system administration interface. The interface has a number of different names: HP is called 'sam', the SGI has the 'System' pick in the main menu, the IBM has 'smit', and the Solaris machines have 'admintool'. All of these commands have online help and are menu driven. You should refer to your documentation for help with those commands.

Sun Computers do require an additional piece of software, called NEWSprint. This software is not bundled with the basic equipment, and is necessary to drive Deskjet and Laserjet printers. Likewise, Silicon Graphics only supports a few line-printers in its normal configuration, and requires the software package Impressario. These packages must be purchased separately from COSMOS/M.

A printcap file is provided for users with HP Paintjets on SunOS systems, which still uses the BSD printcap database.

```
paintjet:\n  :lp=/dev/ttya:\n  :sd=/var/spool/paintjet:\n  :lf=/var/spool/paintjet/errorlog:\n  :mx#0:\n  :br#19200:sh:sf:ms=-parenb,cs8,hup,-cstopb,cread,-clocal,raw,crtscts:
```

Every printer has a name, which defines its type. The name is the first part of the entry. In the example above, it is 'paintjet'.
To add a new printer, find the entry in the file that matches your printer type. Copy that entry to the beginning of entries. If the printer model is not there, you should contact your workstation vendor.

You can change the first entry to one more to your liking, since it is just the name. The lp entry is the name of the output device to the printer, in this case /dev/ttya, or serial port 'a'. Sd is the spooling directory, a directory where files are stored during printing. This directory must exist. Any errors will go to the file 'errorlog', in the same spooling directory.

**Printer Errors**

What if your output is a mess? You should first ensure that you are using the correct command: e.g., the laserjet command on a laserjet printer. If so, some printer problems on Sun Workstations have been rectified by adding the following clause. In the last line, there is a section 'ms=-parenb', which turns off parity. Ensure that this clause is in the printcap entry for your printer/plotter, and try again.

**Special Notes for the X Window System Version**

In the X Window (X11) version of COSMOS/M there are distinct separations between the core, finite element code and the code responsible for graphics operations including user input. The task of interacting with the user, through the graphics output on the display and through the mouse and keyboard inputs, are handled completely by the X server.

This clear cut division of duties frees the core program from the burden of handling the user interface and makes the program more efficient and portable. The following are some general notes for the X version.

1. The X server on your system must be running before you start executing COSMOS/M.

2. In the X version, the output is constrained to be within an X window. (This allows the user to run programs in other windows.)

3. The X version has both pull down menus as well as pop up menus. The pull down menu headings serve as a reminder of available commands while the pop up menus offer flexibility.

4. The menu items can be selected in a single motion involving the mouse. To select a menu item, the user presses the right button while the mouse cursor is either in the display area or in the top bar which contains the major menu names. The user can then select the desired menu item by keeping the right button pressed and moving to the right to pop up successive submenus.

5. The dialog, icon and display areas in GEOSTAR are in fact separate windows allowing for greater use of the mouse. The same buttons can be used to perform different functions in different windows. For example, pressing the right mouse button in the display brings up the main command menu, while in the text window it terminates the command, and in the icon window it serves the same functions as the left mouse button.
6. All commands in the "Windows" submenu (under the "Control" menu), with the exception of the "Wcreate" command, are replaced by more convenient mouse button sequences. The mouse buttons can be used to move or resize a window or to iconify or delete a window. The mouse buttons are also used to move the window icon and to de-iconify an icon or delete the iconified window. Each window also has a rectangular spot at the left end of the title bar. This spot becomes highlighted when a window is selected. When the user clicks the right or left mouse button on this spot (even when the window is not active), the window menu appears. This menu allows the user to close or delete an open window and also provides help information on the mouse movements that move or resize a window.

COSMOS/M Resources for the X Version

Version 2.0 of COSMOS/M has a resources database. The resource mechanism of X provides a convenient means of managing user preferences. A resource is an application specific parameter which affects the behavior of the application. The user changes the values of resources in the resource file to obtain different behavior at run-time from the program. It is possible to have private copies of a global resource file to allow a user to override the global setting of resources by changing the resource values in one's private copy of the resource file. Thus, different users can independently customize COSMOS/M to their liking.

The resource file for COSMOS/M is called "Cosmosm" and is shipped as part of the COSMOS/M package. The user can make private copies of this file by copying this file to the user's HOME or login directory and renaming it ".Cosmosm". Notice the period "." prefix attached to the filename of the private copy. The entries in the resource have the following syntax:

    <resource label>: value

where <resource label> is a pointer to the resource and "value" is the value of the resource. The resource label itself can be broken into two parts. The first is the application or application class name and the second part is the actual resource name. Both of these parts are mandatory and are separated either by a "*" or a ".". The user can comment out resource settings by inserting a "!" character at the beginning of the line. If different values are assigned to the same resource in the same file, then the last assignment overrides. All the resources included in the resource file have default values assigned to them by the program. In the absence of a resource file, these default values will be used. Additionally, the resource manager imposes strict spelling checks and ignores misspelled resource names. Care must be taken while editing this file and adding new lines to the file. For a detailed discussion of X resources, consult the relevant X Window System manuals. (For everyday operation of COSMOS/M, the user will have very little need to consult these manuals.) The following is a detailed description of the user customizable resources for COSMOS/M. If used correctly, they add immense versatility to the program.

**Cosmosm*colorStrategy**

This resource determines how the program uses the color resources of your X server. There are three possible values for this resource. These are: *auto*, *default* and *private*.

"auto" makes the program choose the best available color scheme on the system. The X server may provide many alternative schemes (Visuals) for using the hardware color
resources on your machine. This setting causes the program to search for the better visuals first and terminate the search as soon as a match is obtained.

The criteria for selecting a color scheme is

a. whether it uses a single index for addressing a particular color or three separate indices for addressing the RGB components of the color (decomposed indices),

b. whether it is color or grayscale, and

c. whether it is read/write or read only. The best is the DirectColor visual which uses decomposed indices for the RGB components of color and is read/write. The next best is the TrueColor visual which is just like the DirectColor visual except that it is read only. At the lower end is a Monochrome system which just has two read only colors. If more than one choice of the same class is available the program uses the visual which provides the largest colormap. The auto value is the default setting for this resource.

"default" makes the program bypass the search for the best possible visual and choose the default visual and colormap of the X server. In some situations, the user might want to use the same visual and colormap which are used by the Window Manager on their system. This setting lets the user achieve that. The window decorations put in by the Window Manager are unchanged in this case. The auto setting may sometimes change the colors appearing in the window decorations.

"private" is very similar to the auto setting, with the added feature that after choosing the best visual as in auto, the program creates a private colormap for its use. This guarantees availability of the maximum number of colors for any given visual. The auto setting, on the other hand, tries to share the colormap resources with other applications which might be running on the system (an ideal for all X based applications). The disadvantage with using the private setting is that it always causes the colors outside the COSMOS/M window to change when the user moves the cursor into the COSMOS/M window.

**Cosmosm*textFont**

This is the font used in the dialog areas of GEOSTAR and the screen outputs of the analysis modules. The value should be the name of an available font on your system.

**Cosmosm*menuFont**

This is the font used in the menus of GEOSTAR. The value should be the name of an available font on your system. The size of this font affects the overall size of the COSMOS/M top level window.

**Cosmosm*tableFont**

This is the font used in the help and list screens of GEOSTAR. The value should be the name of an available font on your system. The size of this font affects the overall size of the COSMOS/M top level window.

**Cosmosm*menuForeground**

This resource is the foreground color for the menu strings and icons appearing in GEOSTAR. The value can be the name of any color recognized by the X server. The X server has a color name database usually located in the /usr/lib/X11 directory.
Cosmosm*menuBackground

This resource is the background color for the menu and icon windows appearing in GEOSTAR. The value can be the name of any color recognized by the X server.

The menuForeground and menuBackground colors are used in the reversed video highlighting of menu strings and icons.

Cosmosm*textForeground

This is the color of the text appearing in the dialog window. It must be a valid color name as above.

Cosmosm*textBackground

This is the background color of the dialog window. It must be a valid color name as above.

Cosmosm*textMessageColor

This is the color of the message strings generated by GEOSTAR. It must be a valid color name as above.

Cosmosm*textHelpColor

This is the color of the help strings in the dialog window and the color of the GEOSTAR prompt. It must be a valid color name as above.

Cosmosm*textErrorColor

This is the color of the error messages generated by GEOSTAR. It must be a valid color name as above.

Cosmosm*dialogLines

This specifies the number of lines to be displayed in the dialog area and it determines the height of the dialog area.

Cosmosm*windowCommand

This specifies the command to bring up a terminal window on your system. This resource includes the full pathname of the terminal command which is communicated to GEOSTAR. This resource affects the EDIT and SYSTEM commands of GEOSTAR.

Cosmosm*printCommand

This resource communicates the command to be used to generate hardcopies if a hardcopy device is on-line. For instance, if you have a print spooler program such as "lp" running on your system, set the value of this resource to "lp-d". The trailing "-d" allows the program to append the name of the destination device for each print request.
**Cosmosm*defaultHardCopyDevice**

This specifies the name of the default printer device recognized by your system. Note that this is not a model name such as HP PaintJet or LaserJet but rather is a logical name of the hardcopy device to which your print command sends files for hard copy output. If a value is supplied, it is used as the default hardcopy destination by the program. Examples for this resource values are: LP0, PJET, MYPRINTER, PR136.

**Cosmosm*editor**

The user can use this resource to select an editor which is invoked by the EDIT command in GEOSTAR.

**GEOSTAR*numFuncKeyDefs**

The user can assign a GEOSTAR command string (including arguments) to a function key and make the program execute this command when the user presses that function key. The value of this resource specifies the number of function key definitions that follow in the resource file. Any positive value is valid.

**GEOSTAR*f<n>**

Where <n> is the function key number between 1 and the value of the numFuncKeyDefs resource. The value of this resource is any valid GEOSTAR command string. The user can assign the most commonly used command strings to the function keys.

**GEOSTAR*exposeStrategy**

This resource allows the user to choose the repaint mechanism used when a GEOSTAR window is exposed. In a multi-window environment of a typical desktop, the GEOSTAR window often gets temporarily obscured by other applications and is then exposed. When the window is exposed, the contents of the window are lost unless the program arranges through some means to get back the contents. There are two possible alternatives:

1. **BACKING_STORE**
   
   Under this option (which may not be available on some X servers), the X server maintains a copy of the contents for each window belonging to the application. Whenever the application's window is exposed, the X server copies the retained image back onto the window. BACKING_STORE is not supported by all servers. Further, the memory for backing store comes from the server, which usually has limited memory. Each drawing operation is duplicated internally by the server which updates the backup copy of the window each time the window contents change. Therefore, backing store can be time consuming at times. The backing store option is selected by setting the value of the resource to BACKING_STORE.

2. **PRIMITIVE_LIST**

   GEOSTAR offers this alternative mechanism to handle repainting on exposure as the default option. Under this option, the program maintains a list of all screen output primitives. This list is dynamically built as needed. When a window exposure occurs, the program repaints just the portion of the window which is exposed. This is a more efficient method of repainting. The user can also set a limit to the memory to be set aside for the
primitive list. The default value is 2.0 MB but can be overridden by setting the value of
the resource \texttt{GEOSTAR\textasciitilde prim\textunderscore list\_MBs} to any real number. The default value of 2.0 MB
is usually sufficient.

3. **NONE**

The user can disable all repainting mechanisms by setting the value of the
\texttt{GEOSTAR\textasciitilde exposeStrategy} resource to \texttt{NONE}. In this case, the user has to manually
repaint a particular GEOSTAR window after exposure using the \texttt{REPAINT} command of
GEOSTAR. If the user does not anticipate too many exposures or has limited memory on
the system, this is a good option to use.

\texttt{GEOSTAR\textasciitilde colorSetKey\textless n\textgreater}

Where \texttt{\textless n\textgreater} takes the value 1,2, etc., up to 12. This resource is used to define color sets to be
used in postprocessing. The program comes with 6 colorsets already defined. The user can
add 12 additional colorsets to be used in displaying analysis results. The value of this
resource has the following format:

\begin{verbatim}
C_1C_2C_3C_4C_5C_6#N
\end{verbatim}

where \(C_i\) is one of \{B, C, G, Y, R, M, W\} and stands for the first letter of the following basic
colors \{Blue, Cyan, Green, Yellow, Red, Magenta and White\} and \texttt{N} is a positive integer.

The number of colors used in the key can be any number between \texttt{1} and \texttt{6}. The order of
colors is immaterial but each letter can occur \textit{only once}. The "\texttt{#N}" suffix is optional and \texttt{N}
specifies the number of shades of each color to be used in the colorset. If not included in the
key, it defaults to a value decided by the number of colors available on the system. If \texttt{N} is
specified too large, it is then ignored and the above default value is used. Some examples of
colorSetKeys are:

\begin{verbatim}
GEOSTAR\textasciitilde colorSetKey1: R\#12
GEOSTAR\textasciitilde colorSetKey2: RB
GEOSTAR\textasciitilde colorSetKey3: BCY
GEOSTAR\textasciitilde colorSetKey4: WCR\#8
GEOSTAR\textasciitilde colorSetKey5: BCGWYR
\end{verbatim}

Colorset 2 consists of Red and Blue shades while colorset 4 consists of exactly 8 shades of
White (gray), Cyan and Red.

Some examples of invalid colorSetKeys are:

\begin{verbatim}
GEOSTAR\textasciitilde colorSetKey1: BRB (repetition of letters)
GEOSTAR\textasciitilde colorSetKey1: BCGYNMR (more than 6 letters in the key)
\end{verbatim}

The user can look at the available color sets by issuing the "\texttt{colors}" command from within
GEOSTAR.

\texttt{GEOSTAR\textasciitilde defaultColorSet}

This resource selects the default color set to be used while displaying analysis results. The
value should be set to the colorset number of choice. GEOSTAR provides 6 colorsets on its
own and the user specified colorsets are internally numbered from 7 on up.
GEOSTAR*vectorFontScaleFactor

This resource is used to change the size of the fonts appearing in charts accompanying postprocessing plots. The charts appear in a font size which is proportional to the value of this resource.

GEOSTAR*symbolScaleFactor

This resource is used to change the size of the fonts appearing in labels accompanying various plots displayed by the program. The label font size is proportional to the value of this resource.

Cosmosm*sizeMemBlock

This resource is used by the analysis modules to specify the analysis block size in bytes. The block size determines the number of blocks required to store the stiffness matrix used in the analysis phase. The larger the block size, the smaller the number of blocks required and the less execution time needed for analysis. If too high a value is specified, the dynamic memory allocation calls might fail and the program may terminate with an error message. Provide a value which is reasonable for your system. The default value provided is for a machine loaded with 16 MB. See the Performance Issues section for more information.

FFE*textFont

This is the font used by the Iterative Solvers. This must be a valid font name. A list of fonts can be generated by the program 'xlsfonts'.

FFE*foreground
FFE.background
FFE*borderColor
FFE*lightColor
FFE*shadowColor

All of these settings set colors for various portions of the FFE solver screen. These must be valid colors.

FFE.errorMessage.borderColor

Any errors encountered by the FFE solvers will be printed in a box surrounded by this color.

FFE.indicator.foreground

This is the color of the indicator bar that moves across the screen showing progress of the solution. Again, it must have a valid color.

FFE.memory

This is the memory given when running FFE solvers. The value should be specified in megabytes, followed by "M", e.g., 24M specifies that 24 MB will be given when running FFE modules.
Command Line Switches

In addition to the above resources, the GEOSTAR program also recognizes switches on the command line. These switches can occur in any order and can also be interspersed with the database and input filename arguments which are more commonly used while using GEOSTAR. The following is a brief description of these command line switches:

1. `-display <X server name>`
   This switch is used to select the X server to be used. The user can redirect output to a remote X server on the network with this option. The actual syntax for the X server name itself is machine dependent. Consult your system manuals for running applications on a remote X server.

2. `-geometry widthxheight`
   This switch sets the startup width and height of the GEOSTAR window. The "width" and "height" specify the desired dimensions. Without this switch, the GEOSTAR main window is sized to just contain all its sub windows (the menu bar, icon and text windows). The user can specify larger start up dimensions with this switch. The user can always increase or decrease the window size interactively, using the normal window manipulation techniques provided by the system's window manager.

In addition, there is another switch which can be used only when the database name and input file arguments are also provided on the command line. It should appear only after the input file name. This switch, when present, turns off the echoing of commands and the displaying of entities, as commands are being read from the file. It is useful when reading a large session file in GEOSTAR since it significantly cuts down on the time for reading these files. For example:

   GEOSTAR myDbase bigFile --off

will create a COSMOS/M database called myDbase and execute the commands contained in the file bigFile without any echo or display while the file is being read in.

Use with ASCII Files

The UNIX version can be used with command (ASCII) files generated by other versions of COSMOS/M. This allows the user to run problems analyzed on other machines. Database files are stored in binary formats, so they may not be compatible.

Globetrotter Software Information

Format of the License File

The license file contains one or more of four types of lines. These are server lines, daemon lines, feature lines and featureset lines. As an end-user of COSMOS/M, you need to be concerned only with the Server and Daemon lines. Only these lines contain site specific data which can be changed.

The following is the syntax for the server and daemon lines:
SERVER hostName ID portNo
DAEMON SRAC daemonPath [optionsFile]

where:

- **hostName** is the name of your license server.
- **ID** is the host identification number for the license server.
- **portNo** is the TCP port number used by the program.
- **daemonPath** is the full path name of the directory containing the SRAC daemon.
- **optionsFile** is an optional argument which points to an options file which contains daemon options used to customize license allocation.

The user can edit all the fields mentioned above except the "ID" field.

- The "hostName" field is to be changed only when the license server's name is changed.
- The "portNo" field remains typically unchanged although it can be changed if desired.
- The "daemonPath" points to the directory which contains the SRAC daemon executable.
- The "optionsFile" is the full path name of a file which contains site specific data. The format of this file is explained later in the section titled *End User Customization*.

The feature lines have the following format:

```
FEATURE name daemon version exp_date #users code "vendor-string" [hostid]
```

where:

- **name** is the name of the feature (COSMOS/M module).
- **daemon** is always SRAC.
- **version** is the current major version number of the feature.
- **exp_date** is the expiration date. If it is set to "1-jan-0" it implies that there is no expiration date.
- **#user** is the maximum number of users allowed to use this feature concurrently.
- **code** is an encrypted code string.
- **vendor-string** currently is of the following format "#numNodes#Customer Name" where numNodes is the maximum number of nodes allowed in your models.
- **hostid** is an optional field which restricts the use of the feature to a particular machine.
End User Customization

The administrator for COSMOS/M at the user site can regulate the access and usage of COSMOS/M by editing the daemon options file. The full path name of the file is then entered as the fourth field in the DAEMON line in the license file.

This file consists of lines in the following format. (Feature is an individual COSMOS/M module such as GEOSTAR, STAR, etc.):

```
RESERVE    number feature {USER | HOST | DISPLAY | GROUP} name
INCLUDE    feature   {USER | HOST | DISPLAY | GROUP} name
EXCLUDE    feature   {USER | HOST | DISPLAY | GROUP} name
GROUP      name      <list_of_users>
NOLOG      {IN | OUT | DENIED | QUEUED}
```

where:

- **RESERVE** reserves "number" of copies either to a single "USER", to any user on a "HOST", to all users in a particular "GROUP", or users utilizing a particular DISPLAY.

- **INCLUDE** allows the administrator to specify a list of users who are allowed access to a particular feature. Everyone not listed is excluded access.

- **EXCLUDE** allows the administrator to exclude the named users or groups of users on the specified host.

- **GROUP** allows the definition of a group of users for use in the other commands.

- **NOLOG** causes messages of the named type to be filtered out of the daemon's log output.

Examples:

- `RESERVE 1 GEOSTAR USER john`  Reserves 1 copy of GEOSTAR for john.
- `RESERVE 2 STAR HOST alpha`     Reserves 2 copies of STAR for users on the machine alpha.
- `GROUP thermal joe jane`        The group "thermal" is defined to include users joe and jane.
- `EXCLUDE HSTAR GROUP thermal`   The users in group thermal are excluded from running the HSTAR module.
- `NOLOG QUEUED`                  Filters "QUEUED" messages from the daemon log files.

Performance Issues

**Low Memory Machines**

If you find that there is very little available memory on your system, there are several things that you can do. Edit your Cosmosm resource file and reduce some resource values. The first
resource is GEOSTAR\textsuperscript{*}exposeStrategy. (If you are using the PRIMITIVE\_LIST option, you are saving the graphics in dynamically allocated memory.) Change the option to NONE. The second resource, sizeMemBlock, is typically set to 6,000,000 bytes. (This memory is obtained during runtime to solve the large matrices that finite element analysis requires.) If you receive the "cannot allocate memory" error, try reducing the amount requested.

**Increasing Performance**

The fastest way to speed up the analysis of problems is to increase the sizeMemBlock resource value. However, as noted in the section on X windows resources, you cannot simply increase the value. You must also ensure that your machine has more memory available. The table below has some suggested values. As a UNIX system has more memory available, more is used by various system buffers. Additionally, these numbers assume that COSMOS/M is the only major application running at one time. Check with your system administrator about how much can be allocated.

<table>
<thead>
<tr>
<th>Physical memory installed</th>
<th>Suggested sizeMemBlock parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>32 MB</td>
<td>12,000,000</td>
</tr>
<tr>
<td>48 MB</td>
<td>24,000,000</td>
</tr>
<tr>
<td>64 MB</td>
<td>40,000,000</td>
</tr>
<tr>
<td>96 MB</td>
<td>60,000,000</td>
</tr>
</tbody>
</table>

Also, if possible, you should use a local disk when running COSMOS/M. If your database is on a NFS-mounted disk, your performance will drop substantially and problems will take three or four times normal speed due to network overhead. You should also use the fastest disk possible on your machine, if this is known to you. Note: because UNIX buffers all data, using a RAM disk is not especially helpful, so the memory is better used increasing the size of the stiffness matrix that can be solved.

**Supplementary Installation Information for Running Optimization and COSMOS/M Modules Directly from Pro/ENGINEER**

You must go through these installation steps in order to:

1. Access the COSMOS GEOM menu which may be used to transfer solid geometry from Pro/ENGINEER to COSMOS/M CAD Interface.
2. Access the COSMOS OPTM menu to run optimization.
3. Run COSMOS/M modules directly from Pro/ENGINEER using the Solve command where Pro/MESH and Pro/FEM-POST are used for pre- and postprocessing respectively.

The steps noted below must be carried out subsequent to the program installation in order to have access to the COSMOS GEOM menu and also the COSMOS OPTM menu once Pro/ENGINEER has been loaded. Therefore you must perform these steps to complete the COSMOS/M CAD Interface installation.
1. Copy the two files, ‘config.pro’ and ‘prodev.dat’ from the COSMOS/M subdirectory to the 'text' subdirectory of your Pro/ENGINEER installation. To do this go to the 'text' directory and copy the two files as follows:

   cp $COSMOSM/config.pro .
   cp $COSMOSM/prodev.dat .

**NOTE:** COSMOSM above and also in other statements below refers to the COSMOS/M program installation directory. If a different name was chosen for this directory, then obviously that name should replace COSMOSM.

**config.pro file**

The 'config.pro' file contains the following two statements:

   pro_cosmos_path /bin/runcosmos
   fem_cosmos_version direct-171

Please note that if a 'config.pro' file already exists, then it should be modified by adding the above two statements. Also note that the 'runcosmos' is the default name used for launching various COSMOS/M programs. If a different name is specified during the initial installation, then it should be changed accordingly in the above statement.

**prodev.dat file**

The 'prodev.dat' file contains the following statements:

   Name = cosoptm
   STARTUP = spawn
   COMM = pipe
   EXEC_PATH = $COSMOSM/cosoptm
   TEXT_PATH = $COSMOSM
   END

Again if a 'prodev.dat' file already exits, then it should be modified by adding the above statements to it. These statements are basically required to launch the optimization module directly from Pro/ENGINEER.

2. The above two files are needed in order to launch COSMOS/M and COSMOS/M CAD Interface by issuing the 'runcosmos' command. However to run COSMOS/M CAD Interface by issuing the 'COSPRO' command or Pro/ENGINEER by issuing the 'pro' command and then perform optimization from Pro/ENGINEER, you should either define explicitly the global environment variables in the window from which you wish to launch the programs or preferably add the following statements to your '.cshrc' file so you can issue these commands from any window:

   setenv COSMOSM < installation directory of COSMOS/M>
   set path = ($COSMOSM $path)
   setenv LM_LICENSE_FILE $COSMOSM/license.dat
   setenv CAD_EXEC_NAME pro
For running the optimization module, you need to access the COSMOS OPTM menu in order to define the input related to the optimization process. With the configuration steps noted above, the COSMOS OPTM menu should appear in the PART menu of Pro/ENGINEER after creating or retrieving a part. To access the COSMOS OPTM menu, you may launch Pro/ENGINEER from the COSMOS/M CAD Interface or directly from Pro/ENGINEER. If you want to use COSMOS/M for postprocessing of your optimization results including convergence plots, then you must start from COSMOS/M CAD Interface. If you choose to use Pro/FEM-POST for postprocessing, then you have the choice of starting Pro/ENGINEER either directly or through the interface.

3. Notes on precedence.

   The 'config.pro' and 'prodev.dat' files can be put into three different places:
   1. The current working directory
   2. The user's home directory
   3. The installation directory ( 'text' subdirectory )


   The local directory has precedence over the home directory of the users, which supersedes the 'text' directory. Thus, if you have a local version in your home directory or the current working directory, you will not be able to utilize COSMOS/M features, unless they contain the above mentioned changes.
Chapter 1

Introduction
Introduction

This chapter describes the general features and capabilities of COSMOS/M CAD Interface, as well as how it fits within the Mechanical Computer Aided Engineering (MCAE) design cycle.

COSMOS/M CAD Interface

COSMOS/M CAD Interface is a seamless interface that combines modeling strengths of various CAD programs and the analysis pre- and postprocessing strengths of the COSMOS/M finite element analysis system. The analysis capabilities available in the interface represent a subset of the analysis capabilities of the full COSMOS/M finite element analysis system. The analysis capabilities of the interface are selected to maintain simplicity and cover most of design engineers needs which include linear structural analysis, stress, frequency and buckling analyses as well as nonlinear and dynamic analyses. The program is simple to use. Interaction with the program is further simplified by providing a new system of help that features flowcharts, a masking utility and a What Next? command. This system of help will effectively guide the user throughout the pre- and postprocessing steps. Design engineers may use COSMOS/M CAD Interface in a very productive way towards the selection of the optimum model by evaluating alternative conceptual designs.

Prerequisite Knowledge

Basic understanding of FEA and the type of analysis to be performed is also assumed. Chapter 7 presents useful material for design engineers who have little or no knowledge of FEA. A CAD program and COSMOS/M CAD Interface software should first be successfully installed on your computer system.

COSMOS/M Finite Element Analysis System

The COSMOS/M finite element analysis package is a complete system for modeling and analyzing structural and non-structural designs. The system is modular and is composed of a pre and postprocessor called GEOSTAR, and several analysis modules in the areas of mechanical (linear, nonlinear and dynamic), heat transfer, fluid flow, and electromagnetic analyses. Appendix E offers an overview of Structural Research and Analysis Corporation products. The COSMOS/M Finite Element Analysis System is available on a variety of platforms including PC's, Macintoshes, and engineering workstations. The software is used by many prominent companies and institutions in a wide range of applications including aerospace, automotive, consumer products, electronics, industrial equipment, medical, mechanical, transportation, and many others.

COSMOS/M CAD Interface Screen

The screen is made up of three basic windows (areas). The three windows are:

a. The display window in the center,

b. the dialog or message window at the bottom, and

c. the menu system window to the right.
The display area is used for plotting and listing. The user may create multiple windows inside the Interface.

The dialog window is used for dialog with the interface; all prompts and messages as well as typed inputs are shown in the dialog window. Clicking the left button of the mouse on the border of the window expands the window and shows error messages more clearly.

The menu system is placed in the vertical strip to the right. Move the mouse to the desired option and click the left button to pick a submenu, activate a flag, or execute a command. Once a command is executed, the user may need to use the mouse for picking, or the keyboard to answer prompts in the dialog window. The right button may be used to get on-line help for the highlighted command. Picking entities is done by the left button. The first click highlights an item and the second confirms the choice. The right button may be used to highlight other members of the same entity type if the highlighted member is not the targeted one.

Pressing the Esc key at any time aborts the command. Pressing Enter accepts the default and typing a semicolon “;” accepts all default values in the command.

*The Flowchart menu is linked with a Navigator that guides the user throughout pre- and postprocessing for all types of analyses. The What Next? command helps the user by displaying a message that explains the status of the model and what the program expects next for linear static, frequency, buckling, and thermal analyses.*

**Geometry Transfer Procedures**

The geometry generated by CAD programs can easily be transferred into COSMOS/M CAD Interface using the IGES translator.

It should be noted however, that geometry obtained in COSMOS/M CAD Interface through the IGES translator should be used as is and may not be modified inside the interface. You may add geometric entities inside the interface, but you may not modify the geometric entities that came from CAD programs through the translator.

If a change is needed after the translation is made, go back to the CAD program to make the desired change and repeat the process.

**Other Usages**

It is possible to build simple models from scratch inside the interface without needing a CAD program. Commands to create simple geometric entities are directly provided in the interface. More complicated models can be created by users experienced with GEOSTAR through the activation of the ‘GEOSTAR Command’ mode where full access to all GEOSTAR commands is provided. The access is silent meaning that no menus or on-line help is available. GEOSTAR is the user-interface for all pre- and postprocessing of COSMOS/M, Version 1.75. Appendix D gives brief help for all GEOSTAR commands.

In addition, models created using the full COSMOS/M FEA system, may be simply imported to COSMOS/M CAD Interface.
Features of COSMOS/M CAD Interface

The following is a list of salient features of COSMOS/M CAD Interface.

Navigation and Help System

COSMOS/M CAD Interface comes with an expanded help system carefully designed to provide you with information that simplifies interaction with the interface and guides you through all stages of the design cycle. The system of help in COSMOS/M CAD Interface consists of:

- An on-line help utility that displays information about menus, menu options, commands, and flags, and their usages. The help is accessed by pressing the right button of the mouse.

- A navigator designed to walk you step-by-step through the various stages of finite element analysis starting from geometry creation to postprocessing of results. The navigator is made up of a flowchart utility, a masking feature, and the **What Next?** command as described below.

Flowchart System

Flowcharts are diagrams that graphically depict the sequence of steps needed to perform a specific type of analysis. Flowcharts are available for all types of analyses and may be used to display the status of the various steps in the database. Pressing the right button of the mouse at the location of a particular step in the flowchart will result in listing the information available in the database for that step. The navigation system helps you further by automating the access to the relevant menu and option required to perform the next step. Pressing the left button of the mouse at the location of an unfinished step will automatically take you to the command in the menu system that may be used to complete that step. As an example, if you use the flowchart to check the status of the database and the navigator finds that no element group has been defined, then clicking the left mouse button in the location of an unfinished step (colored red for convenience) will automatically take you to the **Define** option in the Element Group menu so that you may define an element group and proceed with the analysis.

Masking Feature

The masking feature simplifies the interaction with the program by allowing you to only access menus and options that are relevant to the status of the database. The access to irrelevant menus and options is blocked and such menus and options are grayed out. Masking eliminates many sources of errors by preventing inconsistent information in the database. As an example, you may not mesh a curve or a region if the active element group is a tetrahedral solid element. The masking feature may be toggled on and off using the F1 Key.

What Next?

The **What Next?** command is currently available for linear static, frequency, buckling and thermal analyses. As the name of the command suggests, the **What Next?** command gives
you information on the current status of the various steps and suggests the next step needed to proceed with the analysis.

The combination of Flow Chart, masking, and the "What Next?" command results in a user-friendly environment that saves you time and optimizes your learning curve.

**Analysis Capabilities**

Analysis capabilities in COSMOS/M CAD Interface are a subset of the analysis capabilities in the full COSMOS/M FEA system. The following types of analyses are currently supported in the COSMOS/M CAD Interface:

1. Calculation of displacements and stresses for designs subjected to mechanical, gravity, centrifugal, and thermal loadings. Displacements are calculated by the STAR module. Strains and stresses are calculated by the STRESS module. Also available is the new COSMOS/FFE Static module powered by the new FFE Solver.

2. Calculation of natural frequencies and corresponding mode shapes by the DSTAR module. The FFE Solver is also available for natural frequency analysis.

3. Calculation of linear critical buckling loads and corresponding mode shapes by the DSTAR module.

4. Solving linear and nonlinear, steady state and transient heat transfer problems by the HSTAR and the COSMOS/FFE Thermal modules.

5. Solving linear dynamic problems in time and frequency domains by the ASTAR module.

6. Solving nonlinear static and dynamic structural problems by the NSTAR module.

7. Solving fluid flow problems by the FLOWPLUS module.

All modules use a unified database. Using the same database for all modules simplifies the task of running and interfacing various analysis types on the same problem. Temperatures resulting from heat transfer analysis, for example, can be directly used for thermal loading in structural analyses.

**Elements**

The following elements may be used in COSMOS/M CAD Interface:

1. Concentrated masses
2. Trusses and Springs
3. Beams
4. Triangular and quadratic plane elements
5. Triangular and quadratic shells
6. Convection and radiation links
7. Tetrahedral elements
8. Gaps
 Loads and Boundary Conditions

The following loads and boundary conditions may be specified in COSMOS/M CAD Interface:

1. Concentrated forces and moments
2. Pressure loads
3. Thermal loads
4. Gravity loads
5. Centrifugal loads
6. Prescribed displacements
7. Uniform base motion
8. Multiple base motion
9. Prescribed temperatures
10. Heat flux
11. Convection
12. Radiation
13. Loads and boundary conditions for fluid flow analysis

Loads and boundary conditions may be associated with time or frequency curves for appropriate types of analyses.

Material Properties

Various material properties may be specified individually in COSMOS/M CAD Interface, or may be selected from a supplied material library. Material properties may be associated with temperature curves to define temperature-dependent material properties.

Postprocessing

Evaluating the results is made extremely simple by the interface's powerful postprocessing features. Postprocessing features include:

1. An ASCII output file that describes the input data and results.
2. Listing commands to list the results on the screen. List screens may be directed to files.
3. Plotting commands to graphically display output results on the screen. Section plots as well as isoplane plots are provided.
4. Generating hardcopies or files that may be used to generate hardcopies for displayed graphical results.
5. Animating results from both steady state and transient analyses. Stresses for example may be animated on top of deflected shapes.

MCAE In The Design Process

Traditionally, the design and manufacturing process consists of a series of separate steps, performed sequentially, with little or no overlap. Conceptual designs were produced on the back of sketch pads and documented on a drafting board. Design changes were time
consuming and costly to implement, since the design was not analyzed until well past the
drafting and detailing phase. This process often required the design engineer to go back to the
first phase of the conceptual design process to redefine the basic geometry specifications.

However, engineering design is now becoming a more parallel and iterative process. A
design can be evaluated for functionality and manufacturing prior to being finalized. A
greater number of conceptual designs can be quickly studied, allowing for optimized designs.
With the growth of additional software tools and more powerful hardware platforms, the
computer aided processes can encompass all of the “design engineering” steps (Figure 1.1). In
addition to automating design documentation, complex engineering analyses and studies,
combined with intelligent databases, can offer methods of creating optimized and cost
effective products.
Analysis In The Design Process

Specific types of analyses are required in the formulation of most engineering design problems. The analysis may involve stress-strain calculations, thermal computations, or the use of software to describe the dynamic behavior of the system being designed. Since many design components have complex shapes and configurations, they are not easily evaluated using textbook solutions. As a result, engineering analysis computer programs using the finite element analysis techniques have been developed to perform these calculations.

The Finite Element Method Of Analysis

COSMOS/M CAD Interface employs the finite element method, which is one of the most widely used numerical techniques for performing engineering analyses.

In the finite element method, the behavior of the entire object may be predicted by determining the interrelating behavior of the sum of the behavior of its parts. This method significantly reduces the time and cost associated with analyzing complex physical behaviors.

Customer Support

Structural Research provides customer support through a telephone hotline for customers with maintenance agreements. If you have difficulty in modeling your problem or in using COSMOS/M CAD Interface, contact technical support. A BBS (Bulletin Board Service) is available to download analysis databases and files.

Training for the COSMOS/M family of products is held regularly at locations across the world. Contact Structural Research for further information. Structural Research also publishes a newsletter which contains technical tips, training schedules, and news on future releases.
(This page is intentionally left blank)
Chapter 2

Getting Started
Introduction

This chapter gives a description of the various parts of the COSMOS/M CAD Interface screen and their functions. The user is then guided through two examples which demonstrate, in detail, all the steps involved in a typical analysis. As mentioned earlier, you may either import the geometry or the mesh from a CAD program. The two methods shown in Figure 2.1 are illustrated using the examples in this chapter.

Starting COSMOS/M CAD Interface

After a successful installation, COSMOS/M CAD Interface may be started by typing `COSPRO` or `runcosmos CADINT` from the operating system level on Unix platforms or by double-clicking on the "CAD Interface" icon on Windows platforms. Calling the interface from the
COSMOS DIRECTORY should be avoided since the files in that directory should remain intact. After COSMOS/M CAD Interface is started, the interface screen is displayed with the top level startup menu from which the user may switch to a CAD program or load a COSMOS/M database. Once in a CAD program, you may either create a new design or retrieve an existing one. At this time, you have the choice to mesh the model and export the meshed model to COSMOS/M CAD Interface, or develop COSMOS/M geometry for the model by selecting the IGES, Neutral or COSMOS Geom options from the CAD program. You will be returned to COSMOS/M CAD Interface upon exiting the CAD program. Either the geometry or the mesh will be available depending on the choice made in the previous step.

In addition, the interface can be started by typing COSPRO with an argument which is the base name of a previously created COSMOS/M database. For example COSPRO test, loads the previously created COSMOS/M problem test. This method is convenient for quick loading of an existing COSMOS/M database.

You may also start COSMOS/M CAD Interface by typing runcosmos and pressing Enter, and then selecting COSMOS/M CAD Interface from the displayed menu.

COSMOS/M CAD Interface Screen

Figure 2.2 shows an example of the COSMOS/M CAD Interface screen and illustrates its various parts.
The screen is made up of three basic windows (areas). The three windows are:

a. The display window in the center,

b. the dialog or message window at the bottom, and

c. the menu system window to the right.

The display area is used for plotting and listing. The user may create multiple windows inside the interface.

The dialog window is used for dialog with the interface, all prompts and messages as well as typed input are shown in the dialog window. Clicking the left button of the mouse on the border of the window expands the window and shows error messages more clearly.

The menu system is placed in the vertical strip to the right. Move the mouse to the desired option and click the left button to pick a submenu, activate a flag, or execute a command. Once a command is executed, the user may need to use the mouse for picking, or the keyboard to answer prompts in the dialog window. The right button may be used to get on-line help for the highlighted command. Picking entities is done by the left button. The first...
click highlights an item and the second confirms the choice. The right button may be used to highlight other members of the same entity type if the highlighted member is not the targeted one.

Pressing the Esc key at any time aborts the command. Pressing Enter accepts the default and typing a semicolon ";" accepts all default values in the command.

The Flowchart menu is linked with a Navigator that guides the user throughout pre- and postprocessing for all types of analyses. The What Next? command helps the user by displaying a message that explains the status of the model and what the program expects next for linear static, frequency, buckling, and thermal analyses.

### Description of the Vibration Isolator Example

The example in this chapter is a model of a vibration isolator typically used in shop floors to isolate base vibrations of machines. The isolator is made of alloy steel.

The Pro/ENGINEER part (drawing) file ex1.prt for the isolator is provided in the COSMOS/M directory. It should be noted that material properties defined in Pro/ENGINEER will be exported to the interface, only if the mesh is generated in Pro/ENGINEER. With the transfer of geometry, the material properties will not be exported to the interface and therefore should be defined in COSMOS/M CAD Interface even if previously defined in Pro/ENGINEER.

Before starting the COSMOS/M CAD Interface, copy the ex1.prt file from the COSMOS/M directory to the directory which you intend to use for this example.

It is strongly recommended that you do not load the interface from the COSMOS/M directory. Any other directory may be used. All files related to the problem are created in the subdirectory from which COSMOS/M CAD Interface is started. Files storing information related to the problem will be created in the same subdirectory and will all have different extensions but the same name as the part name (refer to Appendix D).

Two procedures will be described for importing the model to COSMOS/M CAD Interface. In the first procedure, only geometry is imported to the interface. The Pro/INTERFACE module is required. Meshing and all other preprocessing steps are performed in COSMOS/M CAD Interface. The emphasis in these examples is on the procedure rather than the accuracy of the solution. See Chapter 6 for more examples.

In the second procedure, all preprocessing is done in Pro/ENGINEER, including the meshing which is generated in Pro/MESH. You need to have the Pro/MESH module in Pro/ENGINEER to proceed with the second example.

### Example 1: Importing the Geometry to COSMOS/M CAD Interface

1. Copy the file ex1.prt to your current work directory, and start the interface. The COSMOS/M CAD Interface screen and the startup menu will be displayed.

2. Choose the CAD Program option from the startup menu using the mouse left button. This will take you to Pro/ENGINEER.
3. From Pro/ENGINEER main menu, select **Mode**, then **Part** from Mode submenu, and **Search/Retr** from the Part submenu. Select file **ex1.prt** to retrieve the vibration isolator model. The solid geometry of the model will be displayed on the Pro/ENGINEER screen.

4. Select **COSMOS Geom** from the Part submenu and **IGES Format** from COSMOS Geom. The geometry files **ex1.igs**, and **ex1.GEO** will be created. You will get a message saying that cosmos trimmed surface file has been created.

5. Exit Pro/ENGINEER. This will take you back to the COSMOS/M CAD Interface and the model will be automatically loaded into COSMOS/M CAD Interface under the same name. Figure 2.3 shows the plot you should see on the screen.

![Figure 2.3 The Vibration Isolator Model as Displayed in COSMOS/M CAD Interface](image)

The geometry of Pro/ENGINEER has now been imported to the interface. The geometric entities of Pro/ENGINEER have been translated into Keypoints, Curves, Contours, Regions, a Polyhedron, and a Part. We will demonstrate the use of the flowcharts and the navigator linked with it in proceeding with the analysis.

### Element Attributes

1. Choose **Flow Chart** from the Main menu, **Import Pro/E Geometry** from the top chart (stack level o) and **Steady State** from the stack level 1 chart. The flow chart displayed represents the steps needed to perform Steady Thermal analysis when geometry is transferred from Pro/ENGINEER. Note that **Geometry** is shown in green indicating that geometric entities exist in the database while other steps are shown in red indicating that these steps were not completed.

2. Click the right button of the mouse on the **Define Element Group Step** to get help on how you may complete this step and press **ECS** when done with reading the information. You do not need to pull down the menus described in the help, the program will do that...
automatically for you. Using the left button of the mouse, choose **Define Element Group** and **Exit** at the lower right corner of the screen. The **Define Group** option in the Element Groups menu from Properties will be automatically highlighted for you. Choose this option. A menu of available types of elements will be displayed. We will use a tetrahedral element called TETRA4R (refer to Chapter 4 for details). The TETRA4R element is a 4-node tetrahedral element with 6 degrees of freedom per node (3 translations and 3 rotations). Choose **TETRA4R** and **Accept**.

The following message will be shown in the dialog area:

```
Element Group [1] >
```

The user should enter a number using the keyboard to label the element group. The number between the brackets is the default number. Press **Enter** to accept all default values in the command. Refer to Chapter 4 for explanation of element group options.

3. Choose **Flow Chart** again. Notice that the **Define Element Group** step is shown in green. Clicking the left button on this option will list the defined element group on the screen. Note also that **Define Real Constant** shows in green as well since the TETRA4R element does not need a real constant set.

4. The next red-colored step is **Define Material**. Use the right button of the mouse to get information on defining material sets, then click the left button on it and **Exit** the **Flow Chart**. The Material menu from Properties will be automatically selected for you with **Pick Material** highlighted. Material sets may be defined using several options (refer to Chapter 4 for details). Choose **Pick Material** to pick a material from the COSMOS/M library.

The following message will be shown in the dialog area:

```
Material Set number [1] >
```

Hit **Enter** to label this material as number 1. Next, the following message appears:

```
Material Name [A_STEEL] >
```

Hit **Enter** to accept the default material as alloy steel. Since we need to define the units, the following message now appears.

```
Unit Label [FPS] >
```

The **FPS** is the **Inch, Pound, Second** system of units and will be chosen in this case. The selected units are consistent with the dimensions of the model which were specified in **inches** when created in Pro/ENGINEER.

5. At this time, element group number 1 (TETRA4R), and material property set number 1 (A-steel) are active. Select **Utilities** from the Main menu, **Status** from Utilities, and verify that element group **EG** number 1, and material property **MP** number 1 (shown at the top) are active. Complete information on how to use the Status Table is given in Chapters 3 and 5. Note the number of various geometric entities in the model under label **MAXM**.
6. Choose **Flow Chart** from the Main menu and notice that the **Define Material** step is shown in green. Click the left mouse button to list the material properties assigned for material set number 1 from the library.

**Meshing**

1. Choose **Mesh Model** and **Exit** the **Flow Chart**, you will be automatically taken to the Meshing menu with the **Mesh Part** option highlighted. Choose **Mesh Part** from Meshing and **Linear** from Mesh Part.

   Use the left button of the mouse to pick the part. Remember that you need to click twice; the first click picks the part and the second click confirms the pick and starts the meshing process. **Linear** means that we want to generate 4-node tetrahedron elements. The **Quadratic** option generates 10-node tetrahedron elements (TETRA10). Note that the only meshing command that is available is Mesh Part because TETRA4R is the active element group.

2. After the meshing process is completed, select **Viewing** from the Main menu, and **Shade** from Viewing. The Shade submenu is an option menu where the shading algorithm and color are to be selected. Choose **Flat, Red**, and **Accept**. Figure 2.4 should appear on the screen.

![Figure 2.4 Shaded Plot of the Vibration Isolator](image)

Note that Gouraud shading produces better quality pictures but takes more time and requires more resources. Gouraud shading requires that the Z-buffer algorithm (from the Hide Opt menu) be active.

3. In this example, only one part was meshed. As a result there will be no coinciding nodes and no node merging is required. It is a good practice however to merge the coinciding
nodes. Select **Meshing** from the Main menu, **Node** from Meshing, and **Merge, All**, and **Accept** from Node. Accept all default values for the messages prompted in the dialog area.

COSMOS/M CAD Interface calculates and assigns an average element size to be used. This size may be displayed by selecting **Geometry** from the Main menu, **Polyhedron** from Geometry, and **List, All** and **Accept** from Polyhedron. Regions making up the polyhedron will be listed on the screen. The computed average element size and tolerance are also listed.

The mesh density may be changed using the Mesh Density menu which allows you to specify mesh control for polyhedra, regions and curves. In this example, we will use the element size calculated by COSMOS/M CAD Interface.

The mesh may only be generated after defining the element group and material property. The interface will only allow you to mesh the appropriate entities depending on the active element group.

4. Choose **Flow Chart** again. Note that **Mesh Model** shows in green. Choose **Mesh Quality**. A message stating that "0.000 percent elements exceeded aspect ratio 7.000000e+00" will be displayed.

**Loads and Boundary Conditions**

1. Choose **Apply LoadsBC** and **Exit** the Flow Chart. The Loads/BConds menu will be automatically selected for you with **Temperature** highlighted. Thermal as well as structural loads and boundary conditions will be applied to the model.

   The two holes at the base of the model will be fixed and the top region, will be subjected to pressure. Temperatures will also be specified. Loads and boundary conditions may be applied to geometric entities only after the mesh has been generated. In order to apply the boundary conditions in a clear way, follow the steps below:

2. Select **Status** from the Utilities menu, and turn on label plotting for RG (fourth column labeled LABL). This means that whenever regions are plotted, their labels will be plotted as well. Select **Save** to activate the change and exit the Status Table.

3. Choose **Viewing** from the Main menu and **Clear** from Viewing to clear the screen.

4. Select **Loads/BConds** from the Main menu and **Displacement** from Loads/BConds. An option menu is displayed. You may use this menu to define, list, and delete displacement boundary conditions. Since we want to fix all nodes associated with the regions defining the two holes at the base, select **Define**, to define the operation type, **Regions** to define the geometry association, **Rigid Joint** to define the constraint type, **Pick** so that you can pick the regions from the screen, and finally **Accept** to execute the command. The message "Pick/Input Region >" will be displayed in the dialog window.

   Move the mouse arrow to region 26 and click the left button. The picked region will be highlighted and its label will be displayed in the dialog area. If the picked region is not 26, click the right button of the mouse until region 26 is picked and then confirm by pressing the left button of the mouse. Repeat the same process for regions 27, 28, and 29. If you hit **Esc** when you are prompted to pick the region, then you may type the region
number and hit **Enter** instead of picking the region by the mouse. The constraints will be plotted on the screen as arrows. One arrow in a given direction is used for translation, two arrows for rotations, and three arrows for both. In this case all nodes associated with these regions are fixed in all directions.

5. Select **Loads/BConds** from the Main menu and **Pressure** from Loads/BConds. An option menu is displayed. You may use this menu to define, list, and delete pressure. Since we want to apply pressure to all element faces associated with the region at the top, select **Define**, to define the operation type, **Regions** to define the geometry association, **Pick** so that you can pick the regions from the screen, and finally **Done** to execute the command. The message "Pick/Input Region >" will be displayed in the dialog window.

Pick region 11 and confirm by pressing the left button. A message asking for the pressure value is then prompted: "Pressure Value [0] >". Type **1000** and press **Enter** to specify 1000 psi. The pressure is displayed as arrows at the centers of element faces on region 11 (Figure 2.5).

6. Select **Loads/BConds** from the Main menu and **Temperature** from Loads/BConds. An option menu is displayed. You may use this menu to define, list, and delete temperatures. Since we want to apply temperature to all nodes associated with specific regions, select **Define**, to define the operation type, **Regions** to define the geometry association, **Pick** so that you can pick the regions from the screen, and finally **Accept** to execute the command. The message "Pick/Input Region >" will be displayed in the dialog window.

Pick region 11. A message asking for the temperature value is then prompted: "Temperature Value [0] >". Enter **100** to specify 100 degrees Fahrenheit. Repeat the procedure to specify 28 degrees Fahrenheit for nodes associated with region 20 (Figure 2.6).

![Figure 2.5 Pressure and Displacement Boundary Conditions](image-url)
Performing Analyses

1. Choose **Flow Chart** and notice that **Run Thermal** is the next step to be performed. The **Set Analysis Options** step is optional. Click the right button of the mouse on it to get information about available options. Click the left button to list the active options.

2. Choose **Run Thermal** and **Exit**. The **Thermal** menu from Analysis will be automatically selected for you.

   We are now ready to submit the model for thermal as well as displacement and stress analyses. We will start with thermal analysis and then use the resulting temperatures to include their effects on displacement and stress calculations. We will proceed as follows:

   Choose **Thermal** from the Analysis menu, and **Run Thermal** from Thermal. Select **Steady State, Direct Solver, and Run** from Run Thermal.

   Control is now given to **HSTAR**, the heat transfer module of COSMOS/M. The COSMOS/M CAD Interface screen disappears, and **HSTAR** starts displaying some information as the analysis progresses. Once the steady state analysis is completed, control goes back to COSMOS/M CAD Interface.

   We may now examine results from the thermal analysis, but we will delay that until we complete the static analysis including the effect of thermal loading.

3. Next, we will perform linear static analysis to calculate displacements and stresses. The effect of temperatures calculated in the thermal analysis will be included in these calculations. Choose **Flow Chart** and notice that no more steps are shown in red and that results are available in the database for postprocessing. Choose **Backward** at the lower right corner and then choose **Linear Static** from the stack level 1 chart. Notice that the
4. Choose **Thermal Loading**, **No In-Plane Stiffness**, **No soft Spring**, **Stress**, **Direct Solver** and **Run** (refer to Chapter 3 for details).

Control is now given to **STAR**, the static analysis module of COSMOS/M. The COSMOS/M CAD Interface screen disappears, and **STAR** starts to display some information as the analysis progresses. Once the deformations are calculated, control is given to the **STRESS** module to calculate the corresponding stresses. After calculating the stresses, control is once again given back to COSMOS/M CAD Interface.

To read temperatures from steady state thermal analysis and use them for thermal loading, all we need to do is to activate the thermal loading flag as shown below. If results were to be read from transient thermal analysis, the **Read Temp** command from the Load Options menu would have to be issued to specify the desired time step prior to analysis.

We may now examine the results of the static analysis. But let us first run frequency analysis to calculate several natural frequencies for this structural model.

5. Choose **Flow Chart** and notice that results of the static analysis are now available in the database for postprocessing. Choose **Backward** and **Frequency** from the stack level 1 chart. The Frequency Chart will show you that the database is ready to execute frequency analysis. Choose **Run Frequency** and **Exit**. The **Frequency** menu from Analysis will be automatically selected for you.

6. Choose **Frequency** from the Analysis menu. Select **Subspace**, **Sturm sequence**, **No Eigenvalue Shift**, **No In-Plane Stiffness**, **No soft Spring**, **Lumped** for lumped mass, and **Run** (refer to Chapter 3 for details). The message "Number of Frequencies [1] >" will be prompted; type 4 and press **Enter** to calculate the lowest 4 frequencies and corresponding mode shapes. Accept the default values for the rest of the options.

Control is now given to **DSTAR**, the frequency and buckling module of COSMOS/M. The COSMOS/M CAD Interface screen disappears, and **DSTAR** starts to display some information as the analysis progresses. Once the frequency analysis is completed, control goes back to COSMOS/M CAD Interface.

7. Choose **Flow Chart** and notice that results of the frequency analysis are now available in the database for postprocessing.

**Postprocessing**

Results of the thermal, static, and frequency analyses are all now available in the database. The file **ex1.TEM** is an ASCII file that contains the results from the steady state thermal analysis. The file **ex1.OUT** contains results of the frequency analyses. The active analysis for postprocessing is the frequency analysis since it is the last analysis we submitted. We will process the results in the same order we ran the analyses.

**a. Thermal Analysis Results**

1. To activate postprocessing for thermal analysis, select **Postprocessing** from the Main menu, **Analysis Type** from Postprocessing, and **Thermal** from Analysis Type. We
may now proceed to process results from the steady state thermal analysis. We may list and plot the results as follows.

2. To list the results, select Lists from Postprocessing, and List Thermal from Lists. You will be prompted for the time step number. Accept the default value of "1" by pressing Enter to list the results from the steady state analysis; accept the default of the "Set Number" prompt. Results will be listed on the screen; press Enter or click the left button of the mouse to display more pages, or hit the Esc key to abandon listing.

3. To generate a contour plot for temperatures, select Display Plots from Postprocessing, and Plot Thermal from Display Plots. An option menu for Plot Thermal will be displayed. Select Temperature, Filled Contour and Accept. Figure 2.7 shows the generated plot.

![Figure 2.7 Temperature Contour Plot](image)

You may also plot temperature gradients following the same procedure.

**b. Displacement Results**

1. To activate postprocessing for static analysis, select Postprocessing from the Main menu, Analysis Type from Postprocessing, and Static from Analysis Type. We may now proceed to process results from the static analysis. We may list and plot the deformation, strain and stress results as follows.

2. Displacements may be listed by selecting Lists from Postprocessing, and List Displacements from Lists. Strains and stresses may be listed by selecting List Strain, and List Stress from the List menu, respectively.

3. Next, we will generate a colored plot for the resultant displacement. Select Plot Displacement from Display Plots. Select Displacement, Resultant, Filled Contour,
Deformed and Accept from Plot Displacement. Press Enter to accept the default load case number. Hit Enter to accept the default value for the scale factor "Scale Factor [416.603] >". A color plot showing the displacement levels on the deformed shape will be displayed.

4. Select Viewing from the Main menu, and Boundary Opts from Viewing. An option menu for the Boundary Opts is displayed; select Model Boundary, Evaluate Edge, and Accept. Type 45 and press Enter to answer the "Tolerance to Ignore Curvature [20] >" message. These options instruct the program to evaluate edges of the model and will generate an edge plot after selecting Repaint from Viewing.

5. We may reduce the figure on the screen by selecting Viewing from the Main menu, and Scale from Viewing. "Scale Factor [0] >", type 0.8 and press Enter to reduce the figure to 80% of its current size. Figure 2.8 shows the resulting plot.

6. To generate a section plot, select Postprocessing from the Main menu, Display Plots from Postprocessing, and Plot Section from Display Plots. The following messages will appear in the dialog window.

   Orientation of Section Planes 0=X, 1=Y, 2=Z, 3=three nodes [0]>

Press Enter. Note that the planes will be normal to the selected axis.

   Number of Section Planes to Be Plotted 1-12 [1] >

Type 12 and press Enter.

   Specify section planes positions 1=yes, 0=defaults [0] >

Press Enter.
Shape flag 0=UNDEF, 1=DEF [0] >

Press Enter. Figure 2.9 shows the resulting plot.

Figure 2.9  Displacement Section Plot

c. Stress Results

1. Stresses may be listed by selecting Lists from Postprocessing, and List Stress from the Lists menu.

2. Select Plot Stress from Display Plots. An option menu will be displayed. Note that the Top/Bottom Face, and Membrane/Bending stress flags are relevant to shell elements only. So in this case it does not matter which one you choose. Also, note that von Mises stresses are independent of coordinate systems.

3. From Plot Stress, select Von Mises, Nodal, Filled Contour, Deformed, and Accept. Hit Enter to accept the default load case, coordinate system and default scale value of "416.603" printed in the message area. The scale is used for plotting the deformed shape. Note that a scale factor of 1.0 results in using the same scale for plotting the model and the deformation. Figure 2.10 shows the resulting plot.
4. Select **Animate** from Display Plots and accept default values in the message window. Von Mises stresses will be animated on top of the deformed shape. Note that animation will be faster in smaller windows. You may stop animation by pressing the **Esc** key.

5. Select **Plot Iso Surfaces** from Display Plots. Type 5 and press **Enter** to answer the prompt: "Number of Iso Planes [1] >", accept other defaults. The resulting plot is shown in Figure 2.11.
6. The deformation plot may be plotted on its own using the **Deformation** command from the Display Plots menu.

### d. Frequency Analysis Results

1. To activate postprocessing for frequency analysis, select **Postprocessing** from the Main menu, **Analysis Type** from Postprocessing, and **Frequency** from Analysis Type. We may now proceed to process results from the frequency analysis.

2. We may list the five frequencies we requested by selecting **Lists** from Postprocessing, and **List Freq/Buckling** from Lists. The frequencies should now be listed on the screen. The resulting list is shown below.

<table>
<thead>
<tr>
<th>Frequency#</th>
<th>Frequency (Rad/sec)</th>
<th>Frequency (cycles/sec)</th>
<th>Period (seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.27179e+01</td>
<td>3.61567e+00</td>
<td>2.76974e-01</td>
</tr>
<tr>
<td>2</td>
<td>7.32951e+01</td>
<td>1.16653e+01</td>
<td>8.57245e-02</td>
</tr>
<tr>
<td>3</td>
<td>1.06312e+02</td>
<td>1.69201e+01</td>
<td>5.91012e-02</td>
</tr>
<tr>
<td>4</td>
<td>1.11737e+02</td>
<td>1.77835e+01</td>
<td>5.62318e-02</td>
</tr>
</tbody>
</table>

3. To plot a mode shape, select **Display Plots** from Postprocessing and **Deformation** from Display Plots. A message asking for the mode shape number will be prompted "Mode Shape Number [1] >", type 4, and hit **Enter** to plot mode shape number 4. Next the message "Scale Factor [21377.1] >" is displayed, hit **Enter** to accept the default scale. This scale factor normalizes the maximum deformation in the model to 10% of the maximum model size. Figure 2.12 shows the generated plot.
Multiple Window Environment

1. Select Viewing from the Main menu, Windows from Viewing, and Window Create from Windows. Click the mouse inside the newly created window.

2. Select Shade from Viewing. Choose Flat, Red, and Accept to activate shading.

3. Select Boundary Opts from Viewing. Choose Model Boundary, Evaluate Edge, and Done. Type 45; and press Enter to answer the "Tolerance Angle to Ignore Curvature [20] >" message.

4. Select Meshing from the Main menu, Elements from Meshing, and Plot, All and Accept from Elements. A shaded element plot will be generated in the active window.

5. Select Postprocessing from the Main menu, Display Plots from Postprocessing, and Animate from Display Plots. Type 3; and press Enter to animate mode shape number 3. Accept other defaults. Use the Esc key to stop the animation.

You may now practice to list, plot, and animate various components for various analysis types. Select Exit to exit COSMOS/M CAD Interface. If you need to open up this model in the future, select COSMOS/M Dbase from the Startup menu.

Example 2: Importing the Mesh from Pro/ENGINEER

In this example, we will show how to import and process models that were meshed in Pro/ENGINEER. The file ex2.prt is provided in the COSMOS/M directory and contains the geometry of the same vibration isolator model processed above. The ex2.mat file contains the material properties information specified in Pro/MESH and is also provided in the COSMOS/M directory. Both files should be copied to the work directory before proceeding with this example.

1. Start COSMOS/M CAD Interface.

2. Select CAD Program from the Startup menu. This will take you to Pro/ENGINEER.
3. **Mode** will automatically be selected from the Main menu, choose **Part** from Mode, and **Search/Retr** from EnterPart.

4. Select **ex2.prt**.

5. To view the model and the boundary conditions, select **FEM** from Part, **Define Model** from FEM, and **Loads/BConds** from Define Model. **Modify** will automatically be selected from Constr Case. Select **LCI** from Concave Name. Then select **Structural** from Loads/BConds, **All** and **Show** from Structural, and **Select All** from FEM Select. All structural loads and boundary conditions will now be displayed.

6. Thermal loads and boundary conditions may be similarly plotted by following the same steps but selecting **Thermal** from Loads/BConds, and **Display All** from Thermal. Select **Done/Return** to go back to the FEM menu.

7. To generate the mesh, select **Make Model** from FEM, and **Tet Mesh** from Make Model. Choose **Yes** to accept the default global min mesh control value. Pro/MESH will start generating the tetrahedral mesh.

8. To output the mesh, select **Output Model** from FEM, and **COSMOS/M, Linear**, **Structural**, and **Accept** from Output Mesh. Accept the default file name by pressing **Enter**.

9. Select **Exit** to exit Pro/ENGINEER and return to COSMOS/M CAD Interface.

10. Once you are in COSMOS/M CAD Interface, the entire finite element data of the model developed in Pro/MESH will automatically be imported into COSMOS/M CAD Interface. Clear the screen by selecting **Clear** from the Viewing menu.

11. Select **Properties** from the Main menu, **Element Groups** from Properties, and **List Groups** from Element Groups. Verify that TETRA4 is the active element group. We need to change the TETRA4 elements to TETRA4R elements. TETRA4R is more accurate since it considers rotations as well as translations.

12. Select **Properties** from the Main menu, **Element Groups** from Properties submenu, and **Define Group** from the Element Groups submenu. A submenu of element names will be displayed, pick **TETRA4R** and **Accept**. Type 1 and press **Enter** for the prompt for the element group label (do not accept the default label 2) and accept all other defaults for element group options. List the element group to confirm the change.

13. Select the **Properties** menu from the Main menu, **Material** from Properties, and **List Sets** from Material.

14. Select **Meshing** from the Main menu, **Element** from Meshing, and **List, All, and Accept** from Element. Examine the list and verify that all elements are associated with **EG** (Element Group) number 1, and **MP** (Material Property set) number 1. Note that **RC** (Real Constant set) is also listed as 1, but this set is empty and is not used. The **ECS** is listed as -1 for all elements. This attribute is explained in Chapters 3 and 5. Note also that 4 nodes are listed for each element.

15. Select **Meshing** from the Main menu, **Element** from Meshing, and **Plot, All, and Accept** from Element. The generated plot is shown in Figure 2.13.
16. You may run thermal, static, and frequency analyses as desired and continue postprocessing as shown in the first example. Practice with various postprocessing options.

17. To review the status of the model in the database, choose **Flow Chart** from the Main menu, **Import Pro/E Mesh** from stack level 0 chart, and **Buckling** from the stack level 1 chart. The flowchart for running buckling analysis with mesh imported from Pro/ENGINEER will be displayed, choose **Continue** to view the rest of the chart and notice that the database is ready to perform buckling analysis. Choose **Run Buckling** and **Exit**, the Analysis menu will be automatically selected for you. Choose **Buckling** from Analysis.

18. Choose **Subspace**, **No Eigenvalue Shift**, **No soft Spring**, **No Sturm Sequence**, and **Run**. Type 2 in response to the number of buckling modes to be calculated. Control will be given to DSTAR to start calculating the buckling load factor. When the analysis is completed, control is given back to COSMOS/M CAD Interface.

19. We may list the two calculated eigenvalues we requested by selecting **Lists** from Postprocessing, and **List Frequency** from Lists. The buckling load factors should now be listed on the screen as 19.5880 and 55.2207.

20. The buckling load for each mode is calculated by multiplying the corresponding load factor by all loads defined. In this case, the first mode will be realized at pressure value of 1000 times 19.5880 or 19,588 psi. The second mode will be realized at 1000 times 55.2207, or 55,220.7 psi.

21. Note that frequency and buckling analyses overwrite each other, so that if you run frequency analysis and then buckling analysis, then results of the frequency analysis will no longer be available in the database.
Chapter 3

Exploring COSMOS/M CAD Interface
**Introduction**

The success of applying the finite element method (FEM) to a physical problem is largely based on the accuracy of the input data. The degree of accuracy of the input data determines the degree of accuracy of the results. Good results are obtained only if the model is a good representation of the prototype.

Figure 3.1 represents a flowchart for typical steps in a finite element analysis.

![Figure 3.1 Finite Element Analysis Steps](image)

COSMOS/M CAD Interface comes with a navigation utility that helps you throughout preprocessing, analysis, and postprocessing. The utility has three features: Masking, Flowcharts, and the What Next? command.

**Masking**

The masking feature helps the user by graying out commands that are irrelevant to the status of the model. As an example the postprocessing menu may not be reached if no results exist in the database. Masking may be toggled on and off using the F1 key.

**Flowcharts**

Flowcharts represent systematic procedures for all the steps needed to perform all types of analyses. Flowcharts may be used any time to provide information regarding the status of the various steps. Colors are used as follows:

- A red color for a particular step indicates that the step must be completed in order to continue.
- A green color indicates that some information related to that step has been provided.
- A blue color indicates that the step is optional.
- A gray color is used in some charts for grouping of options or to indicate that the development work for that step has not been completed yet.
Pressing the right button of the mouse will give you help for performing that particular step. Pressing the left button on a step will take you to the next flowchart or menu where you may continue to input the desired information.

What Next?

The What Next? command is available for linear static (STAR), frequency and buckling (DSTAR), and heat transfer (HSTAR, and FFE Thermal) analyses only. Executing this command will give you information on the basic steps and will tell you what you need to do next. For more information on flowcharts, refer to Chapter 5.

COSMOS/M CAD Interface commands may be classified into four categories: preprocessing, analysis, postprocessing, and utilities.

Preprocessing Commands

Preprocessing encounters all the steps that are needed to prepare the model for analysis. Preprocessing may be completed in CAD programs. In this case all elements and their attributes are readily defined when the model is imported to the interface. The user needs only to specify analysis options, run the analysis, and examine the results.

The user may choose to generate the mesh inside COSMOS/M CAD Interface after importing geometry from a CAD program. CAD geometry is translated into COSMOS/M geometry which can be meshed inside the interface.

At the end of preprocessing, the model is ready for analysis which means that a mesh has been generated, elements are completely defined, loading and boundary conditions are specified, and desired output and analysis options are selected.

Preprocessing includes the following steps:

- Creating the geometry in a CAD program or in COSMOS/M CAD Interface.
- Selecting suitable types of elements to efficiently and accurately represent the physical behavior of the various components of the model. This step includes specifying material properties and real constants sets.
- Generating the finite element mesh, either in CAD programs or in COSMOS/M CAD Interface. In both cases the CAD geometry is used. This step includes verifying the integrity of the mesh by operations like merging coincident nodes and checking element attributes.
- Applying loading and boundary conditions.
- Checking input data.

Analysis Commands

In this step, the user chooses the desired analysis options and submits the model for analysis. COSMOS/M CAD Interface will then transfer control to the specified analysis program which
displays status screens to show the progress of the analysis. Once the analysis is completed, control goes back to COSMOS/M CAD Interface and the user may proceed to evaluate the results. An ASCII file that contains input data and the results of analysis is generated by the analysis module. The extension of this ASCII file is (.OUT) for structural modules, and (.TEM) for the heat transfer modules. In case the analysis fails, the user should refer to the end of this ASCII file for error messages.

Postprocessing Commands

Postprocessing refers to the display and evaluation of the results of analysis in both text and graphical formats. All postprocessing commands are in the postprocessing menu.

Control and Utility Commands

Control and Utility commands are used in both pre- and postprocessing stages. Commands in the Utilities, Viewing, and Selection menus may be classified as control and utility commands.

This chapter presents an insight on the use of various classes of commands and provides useful notes and details on their applications. After a brief section on coordinate systems in the interface, we shall follow the same order used in the menu system to present various classes of commands.

Coordinate Systems

To maintain simplicity, the interface uses the global Cartesian coordinate system for geometry creation, as well as loading and boundary conditions. Once a coordinate system is defined, it becomes active.

Three types of coordinate systems can be directly used in the COSMOS/M CAD Interface:

- Cartesian coordinate systems,
- Cylindrical coordinate systems, and
- Spherical coordinate systems.

Figure 3.2 illustrates the different types of coordinate systems in COSMOS/M. The Cartesian coordinate system is the most commonly used. Often, coordinates in other systems are internally mapped into the Cartesian system during analysis. The coordinates of a point in the Cartesian system are established entirely in units of length for one-, two- and three-dimensional geometries (X, Y, Z). All other coordinate systems use a combination of lengths and angles to represent the coordinates of a point. The cylindrical coordinate system uses two length measures and an angle to describe the location of a point in three-dimensional space (R, θ, Z). The polar coordinate system is a special two-dimensional representation of the cylindrical coordinate system (R, θ). In the spherical coordinate system, two angles and a length quantity are used for coordinate description (R, θ, ϕ).
Coordinate systems created in a CAD program will be readily available in COSMOS/M CAD Interface as part of the geometry transfer.

The following notations are used in COSMOS/M for coordinate systems:

<table>
<thead>
<tr>
<th>Type</th>
<th>Label</th>
<th>Notations on Screen</th>
<th>Notations in Command</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cartesian (X, Y, Z)</td>
<td>0</td>
<td>X, Y, Z</td>
<td>X, Y, Z</td>
</tr>
<tr>
<td>Cylindrical (R, θ, φ)</td>
<td>1</td>
<td>r, t, z</td>
<td>X, Y, Z</td>
</tr>
<tr>
<td>Spherical (R, θ, φ)</td>
<td>2</td>
<td>r, t, p</td>
<td>X, Y, Z</td>
</tr>
</tbody>
</table>

User-created coordinate systems (referred to as local coordinate systems) can be defined using the Coord System command in the Geometry menu or one of the following three commands in the GEOSTAR Command mode:

- **CSYS:** to define a local coordinate system based on 3 keypoints.
- **CSANGL:** to define a local coordinate system based on origin location and three rotations about the global Cartesian axes.
- **CSMATRIX:** to define a local coordinate system based on a specified transformation matrix.

It should be noted that the user-defined coordinate systems may be labeled 3 to 500 and may be Cartesian, cylindrical or spherical. Labels 0, 1, and 2 are reserved, as previously shown, to the built-in COSMOS/M global coordinate systems. The user can activate an existing coordinate system using the Activate command available in the Properties Menu.
Building the Geometry of the Model

Finite element modeling is applicable to almost all practical geometry models. The types of geometries encountered can be broadly classified as:

- Discrete geometry which includes points used for representing models such as concentrated masses and connecting springs.
• Lattice geometry which refers to a gridwork of line elements. Examples include beams, frames, columns and trusses in two- and three-dimensional space.

• Continuous geometry which is used to represent two- and three-dimensional continuous objects. These objects can be planar or solid. Planar objects can be flat (two-dimensional) or curved (three-dimensional). Solid continuous objects require three-dimensional representation.

At the end of the preprocessing stage, the finite element model is fully described to the corresponding analysis module in terms of nodes and elements. Loading, boundary conditions, and analysis options, are all specified in terms of nodes and elements. Geometry is used only as a tool to simplify pre- and postprocessing. It is possible therefore to define a finite element model without any geometry, but this task becomes increasingly difficult as the complexity of the model increases.

Geometric entities serve the following purposes:

a. Facilitate mesh generation:
   Finite element meshes that might take months to generate by directly defining nodes and elements, may take only a few minutes to generate in CAD programs or COSMOS/M CAD Interface.

b. Simplify loading and boundary conditions specifications:
   As far as analyses modules are concerned, all loading and boundary conditions must be applied to nodes and/or elements. COSMOS/M CAD Interface allows the user to specify loading boundary conditions to geometric entities. Whenever such conditions are applied to geometric entities, COSMOS/M CAD Interface calculates and applies the correct values to the associated nodes or elements. It should be mentioned that applying loads and boundary conditions to geometric entities should be subsequent to meshing. Material properties defined in CAD programs are translated to the interface only if the mesh is generated in CAD programs.

c. Convenient for use with selection lists:
   Selection lists act as filters. The existence of geometric entities in the model adds to the power of selection lists and makes them very convenient for pre- and postprocessing. Selection by reference allows the user to select one entity type relative to another. Selection commands are useful for both pre- and postprocessing. For detailed description of selection lists, refer to the help for Selection menu in Chapter 5.

**Geometric Entities in COSMOS/M CAD Interface**

The following table lists the geometric entities and their limits in COSMOS/M CAD Interface:
Table 3.1 Geometric Entities in COSMOS/M CAD Interface

<table>
<thead>
<tr>
<th>Entity Name</th>
<th>Symbol</th>
<th>Limit</th>
</tr>
</thead>
<tbody>
<tr>
<td>Keypoints</td>
<td>PT</td>
<td>12,000</td>
</tr>
<tr>
<td>Curves</td>
<td>CR</td>
<td>12,000</td>
</tr>
<tr>
<td>Surfaces</td>
<td>SF</td>
<td>8000</td>
</tr>
<tr>
<td>Contours</td>
<td>CT</td>
<td>2000</td>
</tr>
<tr>
<td>Regions</td>
<td>RG</td>
<td>2000</td>
</tr>
<tr>
<td>Polyhedra</td>
<td>PH</td>
<td>100</td>
</tr>
<tr>
<td>Parts</td>
<td>PA</td>
<td>100</td>
</tr>
</tbody>
</table>

Notes:

a. The hierarchy from high to low is:
   
   PA > PH > RG > CT > CR > PT
   
   or
   
   PA > PH > SF > CR > PT

b. Surfaces are not explicitly used in COSMOS/M CAD Interface and do not appear in the menus. Surfaces are basically used for defining non-planar regions. Surfaces are transparent to the user in the interface, but are fully accessed in the GEOSTAR Command mode.

c. CAD geometry is automatically translated into these entities. For example if a part is created in a CAD program then the part and corresponding polyhedra, regions, contours, curves, and keypoints are automatically created upon transfer of geometry to COSMOS/M CAD Interface.

d. Only the simplest procedures of geometry creation are provided in the interface. Each entity type has a submenu that branches from the Geometry menu and contains commands to define, list, and delete members of that entity.

The following is a detailed description of COSMOS/M CAD Interface geometric entities.

**Keypoints (PT)**

Keypoints are the most primitive geometric entity in COSMOS/M CAD Interface and the lowest in hierarchy. They are part of all other geometric entities. Keypoints are automatically created upon geometry transfer from CAD programs, or may be created in the interface.

**Curves (CR)**

Curves are 1-D parametric entities. Curves are automatically generated from the geometry transfer to COSMOS/M CAD Interface. When geometry is transferred from a CAD program, a selection list will be active for curves. Curves are represented by cubic equations and can be straight lines, conical, or Bezier. Only straight lines can be directly generated in the interface.
Curves may be meshed to generate 1-D elements like TRUSS3D, BEAM3D, GAP, and RLINK. If a curve is meshed with BEAM3D elements, a keypoint is needed to define element orientation. Every curve has a direction which can be displayed by using the Status command in the Utilities Menu, activating the MARK flag for curves and plotting. The direction of a curve determines the directions of elements generated by its meshing.

**Surfaces (SF)**

Surfaces are 2-D parametric entities which may be flat or curved, quadratic or triangular. Every surface has four curves associated with it. If the surface is triangular, then the fourth curve is degenerate (i.e. of zero length). Surfaces are transparent to the user in COSMOS/M CAD Interface and are only used to define curved regions. Surfaces are automatically generated during the geometry transfer from CAD programs.

Surfaces may be meshed to generate 2-D elements like TRIANG or SHELL3 elements.

**Contours (CT)**

A contour is a closed loop of curves. Contours are created through geometry transfer from CAD programs, or may be created inside the interface. Up to 250 curves may be present in a contour. Contours may be planar or non-planar. All the curves, and hence the keypoints making a planar contour must lie in one plane. Contours may be defined, listed, and deleted using the corresponding option in the contours submenu. If a contour is created inside the interface, the user is asked to specify an average element size that will be used during subsequent meshing.

The direction of a contour is determined by the first curve selected for its definition.

**Regions (RG)**

A region is the area enclosed by a single contour, or the area enclosed between a group of contours. If multiple contours are used, then the user specifies an outer contour and up to 19 inner contours. Inner contours must completely lie inside the outer one and should not intersect each other. Regions are created through geometry transfer from CAD programs, or may be created inside the interface. Regions may be planar or non-planar. All the contours, and hence curves and keypoints making a planar region must lie in one plane. Planar regions may be meshed to generate triangular plane or shell elements. Non-planar polyhedra may be meshed to generate triangular shell elements.

Regions may be defined, listed, and deleted using the corresponding option in the Regions submenu. The average element size for a region may be redefined using the Mesh Density submenu from the Meshing menu.

**Polyhedra (PH)**

A polyhedron is a continuous air-tight closed boundary defined by a group of regions and/or surfaces that constitute the entire surface area of a 3-D enclosure. Polyhedra may be meshed...
to generate triangular shell elements. Polyhedra are automatically generated upon geometry transfer from CAD programs, or can be created inside the interface.

Polyhedra may be defined, listed, and deleted using the corresponding option in the polyhedra submenu. The meshing density may be defined for a polyhedron using the Mesh Density Submenu from the Meshing Menu. Examples of polyhedra will be shown in the meshing section.

**Error in Creating a Polyhedron**

If the Polyhedron is not created when it is defined, repeat the command specifying a higher tolerance until the polyhedron is formed. Do not specify a tolerance greater than $\frac{1}{10^{th}}$ of the element size. Start with tolerances of $\frac{1}{100^{th}}$ or smaller than the element size.

If the polyhedron is not closed, the curves at the free edges are plotted on the screen. This normally occurs when two regions are not properly joined and there will be double curves overlapping at the joint. Using the Curve Merge command, pick the two curves to be merged and use the default increment. The tolerance to be used for merging should be carefully chosen since too large a value will destroy the geometry. Repeat the curve merging for all occurrences of free edges (all curves plotted while defining the polyhedron).

If a message such as INVALID CONTOUR appears at the time of meshing it indicates that some of the curves may have become degenerate or improper curves are merged due to the use of large tolerances during Curve Merge. If this occurs examine the contour curves and keypoints. If the end points of the curves of the contour do not share a single keypoint use the Point Merge command and merge the keypoints two at a time. If any of the contours appear to have missing curves or to be disjointed you must exit, delete the database and read in the input file <partname>.GEO over again.

**Parts (PA)**

A part is a multi-sided 3-D solid defined by the volume enclosed by a single polyhedron, or between a group of polyhedra.

If a group of polyhedra is used to define a part, then the user specifies an outer polyhedron and a group of inner ones. Inner polyhedra must completely lie inside the outer one and should not intersect each other. Parts are automatically created upon geometry transfer from CAD programs, or may be created inside the interface. Parts may be defined, listed, and deleted using the corresponding option in the part submenu. Parts can be meshed to generate 4- or 10-node tetrahedral elements. Meshing specifications are taken from the curves, regions, and polyhedra forming the part. Examples of parts will be shown in the meshing section.

**Elements and Their Attributes**

The fundamental concept of FEA is to divide the domain of the problem into a finite number of sub-domains called elements. Elements are hence the fundamental building blocks of finite element models; their shape approximates the geometry of the model; and their mathematical formulation simulates its physical behavior. For accurate simulation, the selected elements must be able to represent the physical behavior of the model. The user must develop a good
understanding of the problem and must know the various types of elements and their capabilities in COSMOS/M in order to come up with a good FEA model.

The objectives of the analysis is an important factor in the modeling process. In some cases geometry must be accurately modeled, in some other cases the model might look very much different when compared to the prototype, yet it may accurately represent its behavior. For example 1-D elements may be sufficient to model the flow of heat through a plate if the temperature distribution in the plate is not important to the user, otherwise 2-D elements must be used to see the distribution in the plane of the plate. If temperature distribution through the thickness of the plate is also desired, the plate must then be modeled with 3-D elements.

The elements supported by the COSMOS/M CAD Interface represent a subset of the elements supported by the full COSMOS/M, but these elements are carefully selected to maintain simplicity and power. Elements in the COSMOS/M CAD Interface use the state of the art technology to provide users with the most accurate possible results in various disciplines. The selected elements should be sufficient to model many complex physical problems.

**Elements in COSMOS/M CAD Interface may be categorized as follows:**

a. One-node elements (0-D) to model concentrated masses and gap elements.

b. One-dimensional (1-D) line elements for modeling trusses, beams, columns, rods, bars, stiffeners, 2-node gaps and springs. These elements are defined by two nodes. A third node is used to define the orientation of the cross section for 3-D beam elements.

c. Two-dimensional (2-D) area elements for modeling plates and shells. These elements may be triangular or quadratic. Triangular elements consist of 3 corner nodes and optional 3 mid-side nodes. Quadrilateral elements are built from 4 corner nodes (two of them may coincide), and optional 4 mid-side nodes.

d. Three-dimensional (3-D) volume elements for modeling continuums. These elements look like tetrahedral and may have 4 nodes and hence straight edges (TETRA4 or TETRA4R), or may have 10 nodes and quadratic edges (TETRA10).

For regular elements, the same polynomial order is used for both the geometry and the shape functions (linear or quadratic). P-elements are also available in COSMOS/M CAD Interface. P-elements take more time to solve but are more accurate. The P-elements in COSMOS/M CAD Interface are available for the 6-node TRIANG, 8-node PLANE2D, and the 10-node TETRA10 elements. P-elements can only be used for linear static problems (STAR and STRESS).

As mentioned earlier, the elements supported by the interface represent only a small subset of the COSMOS/M element library. Table 3.2 lists elements supported in COSMOS/M CAD Interface and Table 3.3 lists all elements used in the full COSMOS/M FEA system and the types of analysis they support. Note that one degree of freedom per node is used for all elements in thermal analysis.

Elements available in the full COSMOS/M system, but not directly in the interface may be defined through the GEOSTAR Command mode.
# Elements Supported by COSMOS/M CAD Interface

<table>
<thead>
<tr>
<th>Element Name</th>
<th>Element Type</th>
<th>Element Description</th>
<th>No. of Nodes</th>
<th>DOF/Node</th>
<th>Supported Analyses</th>
</tr>
</thead>
<tbody>
<tr>
<td>MASS</td>
<td>Point</td>
<td>Concentrated Mass</td>
<td>1</td>
<td>3</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TRUSS3D</td>
<td>3D Truss</td>
<td>3D Truss</td>
<td>2</td>
<td>3</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>BEAM3D</td>
<td>3D Beam</td>
<td>3D Beam</td>
<td>3</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SPRING</td>
<td>Axial Spring</td>
<td>Gap Element</td>
<td>2</td>
<td>3</td>
<td>S, A, N</td>
</tr>
<tr>
<td>GAP</td>
<td>Uniaxial</td>
<td>Convection Link</td>
<td>2</td>
<td>1</td>
<td>T</td>
</tr>
<tr>
<td>CLINK</td>
<td>Radiation Link</td>
<td>Radiation Link</td>
<td>2</td>
<td>1</td>
<td>T</td>
</tr>
<tr>
<td>RLINK</td>
<td>Axisymmetric Shell</td>
<td>Straight Pipe</td>
<td>2</td>
<td>6</td>
<td>S, A</td>
</tr>
<tr>
<td>SHELLAX</td>
<td>Area</td>
<td>Plane Triangular</td>
<td>3 to 6</td>
<td>2</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TRIANG</td>
<td>Plane Quadrilateral</td>
<td>Plane Quadrilateral</td>
<td>4 to 8</td>
<td>2</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>PLANE2D</td>
<td>3-node Thin Shell</td>
<td>3-node Thin Shell</td>
<td>3</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL3T</td>
<td>4-node Thin Shell</td>
<td>4-node Thin Shell</td>
<td>4</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL4T</td>
<td>4-node Thick Shell</td>
<td>4-node Thick Shell</td>
<td>4</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL6</td>
<td>6-node Shell</td>
<td>6-node Shell</td>
<td>6</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL9</td>
<td>Quadratic Shell</td>
<td>Quadratic Shell</td>
<td>8 or 9</td>
<td>6</td>
<td>S, T</td>
</tr>
<tr>
<td>SHELL3L</td>
<td>Composite 3-node Shell</td>
<td>Composite 3-node Shell</td>
<td>3</td>
<td>6</td>
<td>S, A, N</td>
</tr>
<tr>
<td>SHELL4L</td>
<td>Composite 4-node Shell</td>
<td>Composite 4-node Shell</td>
<td>4</td>
<td>6</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL9L</td>
<td>Composite 8/9-node Shell</td>
<td>Composite 8/9-node Shell</td>
<td>8 or 9</td>
<td>6</td>
<td>S, A</td>
</tr>
<tr>
<td>FLOW2D</td>
<td>Fluid Flow 2D</td>
<td>Fluid Flow 2D</td>
<td>4</td>
<td>3</td>
<td>F</td>
</tr>
<tr>
<td>TETRA4</td>
<td>Volume</td>
<td>4-node Tetrahedral without rotations</td>
<td>4</td>
<td>3</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TETRA4R</td>
<td>Volume</td>
<td>4-node Tetrahedral with rotations</td>
<td>4</td>
<td>6</td>
<td>S, T</td>
</tr>
<tr>
<td>TETRA10</td>
<td>Volume</td>
<td>10-node Tetrahedral</td>
<td>10</td>
<td>3</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>FLOW3D</td>
<td>Volume</td>
<td>Fluid Flow 3D</td>
<td>8</td>
<td>4</td>
<td>F</td>
</tr>
</tbody>
</table>

S : Linear static, frequency, buckling and stress analyses  
N : Nonlinear structural analysis (static and dynamic)  
A : Linear dynamic analysis  
T : Thermal analysis  
F : Fluid Flow analysis
## Table 3.3 Elements Used in the Full COSMOS/M FEA System

| Name                        | Description                                                                 | S  | T  | A  | S  | T  | A  | N  | S  | T  | A  | O  | P  | T  | S  | A  | F  | L  | O  | W  | S  | T  | A  | E  | S  | T  | A  |
|-----------------------------|-----------------------------------------------------------------------------|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|----|
| TRUSS2D & TRUSS3D           | 2D/3D Truss                                                                  |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| BEAM2D & BEAM3D             | 2D/3D Beam                                                                   |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| PIPE                        | Straight Pipe                                                                |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| ELBOW                       | Curved Pipe                                                                  |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| BOUND                       | Boundary                                                                     |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| MASS                        | General Mass                                                                 |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| PLANE2D                     | 4-8 node Plane stress, Plane strain, Axisymmetric                           |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| TRIANG                      | 3-6 node Plane stress, Plane strain, Axisymmetric                           |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL3                      | Triangular Thin Shell                                                        |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL4                      | Quadrilateral Thin Shell                                                     |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL3T                     | Triangular Thick Shell                                                       |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL4T                     | Quadrilateral Thick Shell                                                    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL3L                     | Composite Triangular Shell                                                  |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL4L                     | Composite Quadrilateral Shell                                               |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELLAX                     | Axisymmetric Shell                                                           |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL6                      | 6 node Shell                                                                 |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL9                      | 8-9 node Isoparametric Shell                                                |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SHELL9L                     | 8-9 node Isoparametric Composite Shell                                       |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SOLID                       | 8-20 node Isoparametric Hexahedral Solid                                     |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| TETRA4 & TETRA10            | 4-10 node Tetrahedral Solid                                                 |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| TETRA4R                     | 4 node Tetrahedral Solid with rotation                                       |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SOLIDL                      | 8 node Composite Solid Brick                                                |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SOLIDP2                     | 8-20 node Isoparametric Piezoelectric Hexahedral Solid                      |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| SPRING                      | Spring Element                                                              |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| GENSTIF                     | General Stiffness                                                           |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| RBAR                        | Rigid Bar                                                                   |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| GAP                         | Gap/Surface Contact with Friction                                           |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| CLINK                       | Convection Link                                                             |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| RLINK                       | Radiation Link                                                              |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |
| IMPIPE                      | Immersed pipe                                                               |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |    |

Table 3.3 Elements Used in the Full COSMOS/M FEA System (Continued)

<table>
<thead>
<tr>
<th>Element Library Matrix</th>
<th>S TAR</th>
<th>D ST AR</th>
<th>A ST AR</th>
<th>N ST AR</th>
<th>O P T ST AR</th>
<th>H ST AR</th>
<th>F L O W ST AR</th>
<th>E ST AR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Name</td>
<td>Description</td>
<td>Name</td>
<td>Description</td>
<td>Name</td>
<td>Description</td>
<td>Name</td>
<td>Description</td>
<td>Name</td>
</tr>
<tr>
<td>BUOY</td>
<td>Buoy</td>
<td>FLOW2D</td>
<td>4 node 2D Fluid</td>
<td>FLOW3D</td>
<td>8 node 3D Fluid</td>
<td>MAG2D</td>
<td>4 node 2D Magnetic</td>
<td>MAG3D</td>
</tr>
<tr>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>* : Supported</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td># : Not supported for buckling analysis</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>STAR : Linear static analysis module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>DSTAR : Frequency and buckling analyses module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ASTAR : Post dynamic analysis module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>NSTAR : Nonlinear structural analysis module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>OPTSTAR : Design Optimization module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>HSTAR : Heat transfer module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLOWSTAR : Fluid flow module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>FLOWPLUS : Turbulent fluid flow module (same elements as FLOWSTAR)</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>ESTAR : Electromagnetic module</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>

Element Attributes

Generally, elements are associated with attributes that are needed to completely define them. Whenever an element is generated, it assumes the active attributes. Element attributes may be defined through commands in the Properties Menu. Elements in COSMOS/M CAD Interface are completely defined by the following four attributes:

- Element Group (EG)
- Real Constant Set (RC)
- Material Property Set (MP)
- Element Coordinate System (ECS)

It is important therefore to verify that the correct sets are active before defining elements manually or through the mesh generation if meshing is performed in the interface. The last defined element group, real constant set, and material set will be active. Active attributes may be viewed by the Status command in the Utilities Menu. If you need to activate an existing set, you may redefine the set, or use the Activate command from the Properties menu.
Chapter 3   Exploring COSMOS/M CAD Interface

As an example, suppose that region 5 is to be meshed such that its elements are associated with element group 2, real constant set 3, material property set 4, and ECS 1. The following commands will do the job:

Use the Activate command in the Properties menu as follows.

Choose **Activate** from Properties, **Element Group** from Activate and enter **2**
Choose **Activate** from Properties, **Real Constant** from Activate and enter **3**
Choose **Activate** from Properties, **Material Set** from Activate and enter **4**
Choose **Activate** from Properties, **E Coord Sys** from Activate and enter **1**

After activating the proper attributes, mesh region 5 by using the Mesh Region command from the Meshing Menu.

The values of these attributes for elements are listed by listing elements using the Listing option in the Element submenu.

The following is a brief description of element attributes.

---

**Element Group (EG)**

This attribute must be defined for every element before the analysis can be performed. Through this attribute, the interface distinguishes between truss and beam elements for example because both are associated with a curve. The element group is defined by the Define option in the Element Group submenu in the Properties Menu. You may change an element group by simply redefining it. Element groups are automatically defined if meshing is performed by CAD programs.

---

**Real Constant Sets (RC)**

Real constant sets generally define geometric properties for an element group. In some cases they may also define non-geometric properties (refer to Chapter 4). For example the area of a truss element, and the thickness of a shell elements are defined by real constant sets. Some elements like TETRA4R and TETRA10 do not require a real constant set. Real constants are automatically defined if meshing is done by CAD programs.

---

**Material Property Sets (MP)**

A material property set defines a material by giving it a label and defining several material property names and their corresponding values. For example, the modulus of elasticity and coefficient of thermal expansion may be given certain values and the set is given a number. Materials may also be selected from a library. Material properties are automatically transferred if meshing is done in CAD programs.
Element Coordinate System (ECS)

The basic use of the ECS is for the definition of orthotropic material direction, and the request of stresses in a given coordinate system. Use the Activate command in the Properties menu to activate the desired coordinate system to be used as the element coordinate system.

Meshing in COSMOS/M CAD Interface

Meshing is the creation of nodes and elements. CAD geometry may be meshed in CAD programs or in COSMOS/M CAD Interface. If CAD geometry is imported to COSMOS/M CAD Interface, the user needs to redefine loads, boundary conditions, material properties, and real constant sets. In COSMOS/M CAD Interface, the user may mesh curves, regions, polyhedra, or parts. Elements may also be created manually by defining their nodal connectivity. Masking is used to eliminate problems coming from meshing geometric entities with wrong types of elements. As an example, you may mesh a part only if a tetrahedral element group is active.

Table 3.4 contains meshing commands in COSMOS/M CAD Interface.

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Curve</td>
<td>Meshes curves to generate 1-D elements like TRUSS3D, BEAM3D, SPRING, and CLINK</td>
</tr>
<tr>
<td>Mesh Region</td>
<td>Meshes regions to generate 2-D elements like TRIANG and SHELL3</td>
</tr>
<tr>
<td>Mesh Polyhedra</td>
<td>Meshes polyhedra to generate 2-D elements like SHELL3</td>
</tr>
<tr>
<td>Mesh Part (Linear)</td>
<td>Meshes parts to generate 3-D TETRA4R elements</td>
</tr>
<tr>
<td>Mesh Part (Quadratic)</td>
<td>Meshes parts to generate 3-D TETRA10 elements</td>
</tr>
<tr>
<td>Define Element</td>
<td>Manually creates all types of elements from existing nodes</td>
</tr>
</tbody>
</table>

The systematic procedure for meshing in the interface is as follows:

a. Define the element group.
b. Define material properties.
c. Define the real constant set.
d. Mesh.
e. Repeat steps 'a' to 'd' as needed before meshing other entities.

Mesh Refinement

Mesh refinement in COSMOS/M CAD Interface is provided by the Refine command in the Element submenu. The commands refine the selected elements which must be triangular or tetrahedral. Refinement helps to improve the accuracy of the results in the selected areas.
Activating the P-Method

For 6-node TRIANG, 8-node PLANE2D, and 10-node TETRA10 elements, the user may activate the P-method to improve the accuracy of the results. The order of the polynomial is increased in the analysis option as will be explained later.

3-D Automatic Meshing

3-D automatic meshing refers to the meshing of polyhedra and parts. The meshing may be uniform or non-uniform. Uniform meshing uses a constant average element size throughout the polyhedron or part. Non-uniform meshing uses varying average element sizes. Non-uniform meshing is usually needed for large models where using a uniform element size results in too many elements. The average element size specifications must be reasonable so that enough room is left to the program for transition.

The 3-D automatic mesh generation feature uses the advancing front algorithm in conjunction with a hierarchical technique. In this mesh generation scheme, the nodes are initially placed throughout the model systematically and then the elements are formed by connecting these nodes. The program checks for the best possible way of connecting the nodes. The elements are checked for aspect ratios and nodal locations are modified if necessary to improve the quality of the mesh.

Procedures for 3-D Automatic Meshing

The 3-D automatic meshing capability has been developed to work in a systematic and convenient manner using Polyhedra and Parts. With these geometric entities, the procedure for 3-D automatic meshing of solids and skin-type structures is very simple and straightforward: you just need to build your geometry, and use the appropriate command to generate the mesh.

If you consider a simple 3-D model such as a cube (with no openings) as an example, all six faces of the cube illustrated in Figure 3.3 must be defined as regions. A polyhedron may be defined as the surface area which is the combination of the six faces in this case. The whole cube as a solid may be defined as a part.

Figure 3.3   Boundary Faces of a Cube
The front and back faces, the sides as well as the top and bottom faces of the cube (shaded areas in the figure) must be defined as regions to form a POLYHEDRON which defines a 3-D hollow volume entity. To define a 3-D solid volume, a PART entity must be defined using one or more polyhedra. You can then use the commands in the Meshing menu to mesh the polyhedron or the part. If geometry is created in a CAD program and transferred to the interface, the user need not create any polyhedra or parts since they will be automatically translated and may be readily meshed. If the user needs to define polyhedra or parts in COSMOS/M CAD Interface, then the "Define Polyhedra", and "Define Part" should be used.

When you issue the Define Polyhedra command, you will also be prompted to input the average element size for use in automatic meshing (this input overrides the average element size defined earlier during contour definition). When the part is meshed, the advancing front technique proceeds with the initial placement of the nodes throughout the solid as shown in Figure 3.4.

![Figure 3.4 Initial Placement of Nodes](image)

The number of nodes generated depends on the order of tetrahedra to be generated. If you use **Linear Mesh Part**, there will be four nodes generated for each tetrahedron. After the nodes are placed, the mesh generation process continues by connecting the generated nodes to form tetrahedra. After a tetrahedron is formed, the mesh generation scheme checks for aspect ratios and internal angles of the element to ensure good quality of the elements. If an element with bad quality is encountered, the program performs several trials to form an element with good geometry by relocating the nodes. Figure 3.5 illustrates the process of element generation from initially placed nodes.
After the mesh generation is completed, there will be a summary on the number of elements and nodes generated and the total volume of the model. You can use the usual commands to list, identify and view the elements. You can also use the hidden line removal or shading (Viewing menu) features to obtain a clear picture of the finite element mesh. Figure 3.6 shows the generated finite element mesh with and without hidden lines.

A diagram for the steps required in three dimensional automatic meshing is shown in Figure 3.7.
The example of a solid cube meshing discussed above, albeit simple, serves to illustrate how the 3-D automatic meshing of solids and surfaces is performed in COSMOS/M CAD Interface. However, the 3-D mesh generation capability is applicable to models with more complex geometries. In Figures 3.8 to 3.13, several examples of 3-D solids and their finite
element meshes automatically generated are shown. You can confirm from these figures that the 3-D automatic mesh generation feature of COSMOS/M CAD Interface is quite powerful and can be applied to many real world problems.

Figure 3.8  Geometry and Finite Element Mesh of a Spring Coil

Figure 3.9  Solid Model, Geometry and Finite Element Mesh of a Connector  (example file EX3D1.GEO)

Figure 3.10  Solid Model, Geometry and Finite Element Mesh of a Connector
Figure 3.11  Solid Model, Geometry and Finite Element Mesh of a Plate-Mounted Drive Tightener

Figure 3.12  Solid Model and Geometry of a Knuckle Joint
Entities and Commands for 3-D Automatic Meshing

The geometric entities and commands listed in Table 3.5 are used for the automatic 3-D meshing. The commands described in the table can be accessed from the Geometry and Meshing menus of COSMOS/M CAD Interface.

Table 3.5 Entities and Commands for 3-D Automatic Meshing

<table>
<thead>
<tr>
<th>Entity Name</th>
<th>Menu</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>POLYHEDRON</td>
<td>Polyhedra Define</td>
<td>Define a polyhedron</td>
</tr>
<tr>
<td></td>
<td>List</td>
<td>List polyhedra</td>
</tr>
<tr>
<td></td>
<td>Delete</td>
<td>Delete polyhedra</td>
</tr>
<tr>
<td></td>
<td>Plot</td>
<td>Plot polyhedra</td>
</tr>
<tr>
<td>PART</td>
<td>Part Define</td>
<td>Define a part</td>
</tr>
<tr>
<td></td>
<td>List</td>
<td>List a part</td>
</tr>
<tr>
<td></td>
<td>Delete</td>
<td>Delete parts</td>
</tr>
<tr>
<td></td>
<td>Plot</td>
<td>Plot part</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Command Name</th>
<th>Function</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh Part</td>
<td>Automatic meshing of fully enclosed 3-D solid volumes</td>
</tr>
<tr>
<td>Linear</td>
<td>(PARTS) using 4- or 10-node tetra elements</td>
</tr>
<tr>
<td>Quadratic</td>
<td></td>
</tr>
<tr>
<td>Mesh Polyhedra</td>
<td>Automatic meshing POLYHEDRA using 3-node shell elements</td>
</tr>
</tbody>
</table>
POLYHEDRA correspond to closed boundary surfaces that are constructed from a set of regions and are generally considered as simply-connected 3-D surface geometries. PARTS on the other hand, correspond to 3-D solid geometries that are formed from one or more POLYHEDRA. PART entities can therefore be considered as multiply-connected 3-D geometries such as solid models with cavities. Therefore, PART represents a geometric entity which is one order higher than POLYHEDRON.

The relationship between polyhedra and part entities is analogous to the relationship between contours and regions, i.e., just as regions are constructed from a set of contours, parts are built from a set of polyhedra. With the construction of polyhedra and parts completed, you will be able to perform automatic meshing of the polyhedra using 3-node shell elements or 3-D automatic solid meshing of the component with 4- or 10-node tetra elements. In either case, you will have control of the overall mesh density by specifying the average element size for the associated polyhedra, regions, contours and curves.

**Entity Size Limits**

Table 3.6 presents size limits on the geometric entities used in 3-D automatic meshing.

<table>
<thead>
<tr>
<th>Entity</th>
<th>Size Limits</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>64k Version</td>
</tr>
<tr>
<td>REGIONS in a POLYHEDRON</td>
<td>2000</td>
</tr>
<tr>
<td>POLYHEDRA in a PART</td>
<td>60</td>
</tr>
<tr>
<td>POLYHEDRA</td>
<td>100</td>
</tr>
<tr>
<td>PARTS</td>
<td>100</td>
</tr>
</tbody>
</table>

128k version is available only for Windows platforms  
256k version is available only for Unix platforms

**Hints for Using 3-D Automatic Meshing**

The tips and hints presented below are quite general and they do not refer to any specific example. They are not in any sense complete, and you must always use your own judgment for meshing your models. It should be noted that if the geometry is transferred from CAD programs, then the polyhedra will be defined by flat regions or regions on underlying surfaces.

- The 3-D solid or hollow volume must be completely enclosed with regions. Cut-outs, holes and openings are permitted in the POLYHEDRON or PART entities. You must make sure that the internal faces of the openings are properly defined as regions.

- Care must be taken to see that there are no internal or common boundaries or partitions when defining polyhedra and parts. However, if you are required to have internal boundaries or partitions as shown in Figure 3.14 (for models with different material properties, etc.), then you should treat the volumes on either side of the partition separately for polyhedron and part definitions. However, the different volumes can be subsequently meshed together.
COSMOS/M CAD Interface provides a convenient way to place parts of your model in a selection list. This is done by using the commands of the Selection menu. For example, you can first place regions describing one of the cubes in the figure above (including the common boundary regions) in the selection list, and proceed to defining the polyhedron and part. Next, place the regions of the other cube (including the common boundary surface or region) in the selection list and proceed to defining the second polyhedron and part. Use the meshing commands to automatically generate the mesh of both cubes.

- The average element size is specified when you define a polyhedron. This input overrides the average element size specified during contour definition. You can therefore specify any value for average element size or mesh density during contouring.

- When specifying the average element size, you must exercise caution. It should not be too large when compared to the smallest dimension of the model. If it is more than two to three times the smallest curve dimension, the program may warn you of this condition and the meshing process may fail. The element size on the other hand should not be smaller than $1/10^{th}$ of the tolerance value.

- You don't have to apply the same element size value for all parts in order to maintain node compatibility. The common regions and curves however must have identical meshing specifications for compatibility to be realized.

- Commands in the Mesh Density submenu can be used to control the mesh density by specifying element sizes for curves, regions and polyhedra.

- If you are using TETRA10 elements in your mesh, the mid-points of these elements may sometimes appear to be misplaced. This is due to the fact that element plots are drawn by connecting the extreme nodes of the elements and they may not include the mid-side nodes. However, the mid-side nodes are accurately placed at proper locations during mesh generation and are used in that manner by the analysis modules.

- 3-D automatic meshing is in general numerically intensive and time consuming. Often, it is a good practice to mesh polyhedra with the desired average element to get a rough estimate on the number of elements and the meshing time. You can use this information to approximately estimate the number of elements for solid meshing and its execution time.
Properties Menu

The Properties menu is devoted to the management of element attributes which are:

a. element groups,
b. material property sets,
c. real constant sets, and
d. element coordinate systems.

Whenever an element is generated, it will assume the active attributes.

Element Groups

Element groups supported directly in COSMOS/M CAD Interface may be selected from the menu. Elements supported in the full COSMOS/M system that may not be chosen directly from the menu, may be defined using the EGROUP command in the GEOSTAR Command mode.

Material Properties

There are 3 ways for defining material properties:

a. Selecting a material from the COSMOS/M Material Library,
b. defining commonly used material properties from the menu set for this purpose,
c. use of the Additional Prop option to type in the property name(s) and value(s), and
d. selecting a material and associated temperature curve from the InfoDex Material Library.

Any combination of the above methods may be used to define a material property set.

COSMOS/M Material Library

The material library feature of COSMOS/M CAD Interface allows you to select standard material properties from a built-in library. The built-in material properties are available in FPS, MKS, and SI system of units. The property names used in the material libraries and their description is provided in the table below.
Table 3.7 Property Names in the Material Libraries

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ALPX</td>
<td>$\alpha_x$</td>
<td>Coefficient of thermal expansion in the x-direction</td>
</tr>
<tr>
<td>BETA</td>
<td>$\beta$</td>
<td>Coefficient of volume expansion</td>
</tr>
<tr>
<td>C ($C_p$)</td>
<td>$C_p$</td>
<td>Specific Heat</td>
</tr>
<tr>
<td>DENS</td>
<td>$\rho$</td>
<td>Mass density</td>
</tr>
<tr>
<td>EX</td>
<td>$E_x$</td>
<td>Elastic modulus in the x-direction</td>
</tr>
<tr>
<td>GAMMA</td>
<td>$\gamma$</td>
<td>Ratio of specific heats</td>
</tr>
<tr>
<td>GXY</td>
<td>$G_{xy}$</td>
<td>Shear modulus in the x-y plane</td>
</tr>
<tr>
<td>KX</td>
<td>$K_x$</td>
<td>Thermal conductivity in the x-direction</td>
</tr>
<tr>
<td>NUXY</td>
<td>$\nu_{xy}$</td>
<td>Poisson's ratio for x-strain due to y-stress</td>
</tr>
<tr>
<td>VISC</td>
<td>$\eta$</td>
<td>Dynamic viscosity</td>
</tr>
</tbody>
</table>

By default, the built-in material library assumes the material properties to be *isotropic*. If it is necessary to use orthotropic material properties in your model, you need to therefore provide the remaining property components under the same material set number. The ECS flag must be set to the appropriate coordinate system for elements with orthotropic properties. Use the Activate command for this purpose before defining the properties.

The command Pick Material from the Properties menu can be used for selecting the pre-defined material properties from the built-in material library. The following sections describe the procedure for using the material library in more detail. It should be noted however that not all material properties listed in the library are utilized in COSMOS/M CAD Interface.

When you use the Pick Material command, the material properties will be read from the text file, PICKMAT.LIB, installed in the COSMOS/M directory. Enter the label of the material set you want to define and the material name, and type of units.
### Table 3.8 Available Material Types in the Built-in Library

<table>
<thead>
<tr>
<th>COSMOS/M Name</th>
<th>Material Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>ACRYLIC</td>
<td>Medium-High Impact Acrylic</td>
</tr>
<tr>
<td>AIR</td>
<td>Air (300 K)</td>
</tr>
<tr>
<td>ALUMINUM</td>
<td>Aluminum Alloy</td>
</tr>
<tr>
<td>AL_1345</td>
<td>Aluminum 1345 Alloy</td>
</tr>
<tr>
<td>AL_1350</td>
<td>Aluminum 1350 Alloy</td>
</tr>
<tr>
<td>AL_2014</td>
<td>Aluminum 2014 Alloy</td>
</tr>
<tr>
<td>AL_2018</td>
<td>Aluminum 2018 Alloy</td>
</tr>
<tr>
<td>AL_2024</td>
<td>Aluminum 2024 Alloy</td>
</tr>
<tr>
<td>AL_3003</td>
<td>Aluminum 3003 Alloy</td>
</tr>
<tr>
<td>AL_6061</td>
<td>Aluminum 6061 Alloy</td>
</tr>
<tr>
<td>AL_7079</td>
<td>Aluminum 7079 Alloy</td>
</tr>
<tr>
<td>AL_BRONZE</td>
<td>Aluminum Bronze</td>
</tr>
<tr>
<td>A_STEEL</td>
<td>Alloy Steel</td>
</tr>
<tr>
<td>BRASS</td>
<td>Silicon Brass &amp; Bronze</td>
</tr>
<tr>
<td>CA_STEEL</td>
<td>Cast Alloy Steel, below 8% alloy content</td>
</tr>
<tr>
<td>COBALT</td>
<td>Cobalt</td>
</tr>
<tr>
<td>CS_STEEL</td>
<td>Cast Stainless Steel, CF-8M or CF-20, water quenched</td>
</tr>
<tr>
<td>D_NICKEL</td>
<td>Nickel Alloy, Duranickel 301</td>
</tr>
<tr>
<td>GC_IRON</td>
<td>Gray Cast Iron, ASTM Class 40</td>
</tr>
<tr>
<td>GLASS</td>
<td>Glass</td>
</tr>
<tr>
<td>GOLD</td>
<td>Pure Gold</td>
</tr>
<tr>
<td>IRON</td>
<td>Iron</td>
</tr>
<tr>
<td>LEAD</td>
<td>Pure Lead</td>
</tr>
<tr>
<td>MAGNES</td>
<td>Magnesium Alloy, wrought or cast</td>
</tr>
<tr>
<td>MC_IRON</td>
<td>Malleable Cast Iron, ASTM A220</td>
</tr>
<tr>
<td>MN_BRONZE</td>
<td>Manganese Bronze</td>
</tr>
<tr>
<td>MOLYBDENUM</td>
<td>Molybdenum</td>
</tr>
<tr>
<td>MONEL</td>
<td>MONEL 400 alloy of Nickel</td>
</tr>
<tr>
<td>NYLON</td>
<td>Nylon 6/10</td>
</tr>
<tr>
<td>PC_STEEL</td>
<td>Plain Carbon Steel</td>
</tr>
<tr>
<td>PORCELAIN</td>
<td>Ceramic Porcelain</td>
</tr>
<tr>
<td>RUBBER</td>
<td>Rubber</td>
</tr>
<tr>
<td>SILVER</td>
<td>Pure Silver</td>
</tr>
<tr>
<td>STEEL</td>
<td>Steel</td>
</tr>
<tr>
<td>ST_1020</td>
<td>Steel, AISI C1020 (Hot Worked)</td>
</tr>
<tr>
<td>ST_304</td>
<td>Steel, AISI 304 (Sheet)</td>
</tr>
<tr>
<td>ST_ST</td>
<td>Stainless Steel</td>
</tr>
<tr>
<td>TITANIUM</td>
<td>Titanium</td>
</tr>
<tr>
<td>TUNGSTEN</td>
<td>Tungsten</td>
</tr>
<tr>
<td>T_BRONZE</td>
<td>Tin Bronze 1B</td>
</tr>
<tr>
<td>VANADIUM</td>
<td>Vanadium</td>
</tr>
<tr>
<td>WATER</td>
<td>Water (300 K)</td>
</tr>
<tr>
<td>WS_STEEL</td>
<td>Wrought Stainless Steel</td>
</tr>
<tr>
<td>W_COPPER</td>
<td>Wrought Copper</td>
</tr>
<tr>
<td>ZIRCONIUM</td>
<td>Zirconium</td>
</tr>
</tbody>
</table>

Note: Properties are measured at 70° F.
Using the built-in material library is extremely simple. For example, if you like to select Wrought Copper using the FPS system of units for material set 1, the Pick Material command is executed as follows:

```
Material property set [1]>
Material Name [A_STEEL]> W_COPPER
Unit Label [FPS]>
```

The properties of Wrought Copper stored in the library are shown below:

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.016E+08 psi</td>
<td>0.11E+07 Kgf/cm/cm</td>
<td>0.11E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
<td>0.33</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.58E+07 psi</td>
<td>0.41E+06 Kgf/cm/cm</td>
<td>0.40E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.11E-04 /Fahrenheit</td>
<td>0.20E-04 /Centigrade</td>
<td>0.20E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.79E-03 lbf*s/s/in**4</td>
<td>0.86E-05 Kgf*s/s/cm**4</td>
<td>0.84E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.31E-02 BTU*in/lbf/s/s/F</td>
<td>0.56 Cal/cm/s/C</td>
<td>0.23E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>35. BTU*in/lbf/s/s/F</td>
<td>0.88E+05 Cal*c/kgf/s/s/C</td>
<td>0.38E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

The selected properties can be listed using the listing option in the Material submenu. Refer to Appendix C for a complete list of the contents of the material library.

User materials and additional material properties may be added to the COSMOS/M Material Library by editing the file "PICKMAT.LIB" in the COSMOS/M directory.

Temperature-dependent material properties are defined by creating and activating a temperature curve and then defining the property. The actual value of the property will be calculated by multiplying the property value by the curve value at the element temperature. Refer to Chapter 5 for details.

**InfoDex Material Library**

An extensive material library compiled by InfoDex is optionally available for use with COSMOS/M CAD Interface. The **Run Material** command temporarily creates a new window and runs a material browser which may be used to define material property sets. The browser provides a large library of materials and associated temperature curves. After selecting the desired materials and exiting the browser, the selected materials and associated temperature curves will be automatically loaded into the database. Control will then be transferred back to COSMOS/M CAD Interface. The Additional Prop command may be used to add or modify any material properties in the set. The browser is an add-on utility that must be acquired before executing this command. Contact SRAC for more information.

**Material Property Curves**

A material property curve defines a material set by defining the stress-strain relation for a material. Material property curves are used in nonlinear static and dynamic structural analyses (NSTAR) only. You do not need to define modulus of elasticity or any other material property if a material set is defined by a material property curve. Generated elements will be associated with the active material set whether defined by selection from the library, direct input, or by stress-strain curve.
Real Constant Sets

Real constant sets should be defined after an element group has been defined. For 3-D beam elements, a section may be selected from a given list where the program will calculate many of the required input based on the section type and dimensions.

Element Coordinate System

Coordinate systems created in CAD programs are directly exported to COSMOS/M CAD Interface as part of the IGS geometry transfer. Coordinate systems may also be created from the GEOSTAR Command mode.

To define orthotropic material properties: a) use the Activate command to activate an element coordinate system which may be any predefined coordinate system, and b) define the desired properties in various directions of the active element coordinate system.

As an example, if coordinate system number 3 is activated as the element coordinate system and EX and EY are defined, then EX and EY are the elasticity moduli in the X- and Y-directions of system 3 respectively.

To request stresses in a given coordinate system, set the flag for stress direction to local in the element group definition, then stresses will be calculated in the directions of the element coordinate system. If you set the stress direction flag to global for all element groups, then you may change the coordinate system for stress calculations in the postprocessing stage.

Applying Loads and Boundary Conditions

All commands related to specifying loads and boundary conditions are inside the Loads/BConds menu. The analysis modules require that loads and boundary conditions be applied to nodes or elements. The interface however lets the user specify loads and boundary conditions to meshed geometric entities. The geometric entities must be meshed before applying the loads and boundary conditions, otherwise no conditions will be applied. Loads and boundary conditions specified for a node or element are deleted if that node or element is deleted. Loads and boundary conditions submenus are arranged such that the user chooses the operation type (define, list, delete), the entity type (node, element, curve, region), and whether to pick the entity or use the selection list.

The following table illustrates the type and use of various loads and boundary conditions.
Table 3.9 Loads and Boundary Conditions in COSMOS/M CAD Interface

<table>
<thead>
<tr>
<th>Load/BConds</th>
<th>Applied to</th>
<th>Used in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacements</td>
<td>nodes</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Force / Moment</td>
<td>nodes</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Base Motion</td>
<td>nodes</td>
<td>Post-dynamic analyses</td>
</tr>
<tr>
<td>Temperature</td>
<td>nodes</td>
<td>Structural &amp; thermal analyses</td>
</tr>
<tr>
<td>Velocity</td>
<td>nodes</td>
<td>Fluid Flow analysis</td>
</tr>
<tr>
<td>TK Energy</td>
<td>nodes</td>
<td>Fluid Flow analysis</td>
</tr>
<tr>
<td>Dissipation Rate</td>
<td>nodes</td>
<td>Fluid Flow analysis</td>
</tr>
<tr>
<td>Node Pressure</td>
<td>nodes</td>
<td>Fluid Flow analysis</td>
</tr>
<tr>
<td>Density</td>
<td>nodes</td>
<td>Fluid Flow analysis</td>
</tr>
<tr>
<td>Pressure</td>
<td>element faces</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Convection</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>Heat Flux</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>Radiation</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>Boundary Element</td>
<td>element faces</td>
<td>Fluid Flow analysis</td>
</tr>
</tbody>
</table>

Repeating a load or boundary condition command and specifying a new value overwrites the old one. For example if a pressure of 10.0 was specified in the x-direction for elements associated with a region and this value needed to be changed, then the user may repeat the command and specify the new value and the same direction. If another value is specified in some other direction then both loads will be acting.

Prescribed displacements may be used as loading, i.e. you may solve a static problem by specifying prescribed displacements at some nodes without any other types of loading.

Sufficient boundary conditions should be applied to maintain stability. If the constraints are not sufficient, the model will not be stable and special provisions like soft springs or shifts need to be employed (refer to the analysis section).

A submenu is provided for load options, in which the user may specify information related to special loadings like gravity, centrifugal, and thermal. The temperatures resulting from a steady state or transient thermal analysis may be directly read and applied as thermal loading for linear static problems using the Read Temp command in the Load Options submenu.

Loads and boundary conditions are ignored if they are not activated or are not supported. For example if temperatures are specified and a static analysis is performed without activating thermal loading, then temperatures will be ignored. Also, forces are ignored by thermal analysis, and radiation is ignored by static, frequency, and buckling modules.

Loads and boundary conditions are associated with time, frequency curves for NSTAR and ASTAR. Refer to the help for the Additional Prop command in Chapter 5 for details.
Viewing in COSMOS/M CAD Interface

Producing the right view on the screen facilitates and accelerates pre- and postprocessing. The plotting commands may be used to plot the desired entities on the screen. The settings in the Status Table are also helpful since it controls plotting colors, labels, marks, and other flags. The user may need to experiment with various options to produce the correct view.

The extensive Viewing menu in the interface permits the user to get the desired views during pre- and postprocessing. The user may create up to four windows. One window is active at any given time. There are two types of viewing commands: action commands that change the picture on the screen; and setting commands that do not change the picture on the screen but change the setting for subsequent plots.

Tables 3.10 gives brief descriptions of action viewing commands.

**Table 3.10  Action Commands in the Viewing Menu**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Pscale</td>
<td>Automatically scales the plotted entities to the active window.</td>
</tr>
<tr>
<td>Scale</td>
<td>Automatically scales the whole module to the active window.</td>
</tr>
<tr>
<td>View</td>
<td>Changes the point of view and replots the plotted picture.</td>
</tr>
<tr>
<td>Zoomin</td>
<td>Magnifies a chosen part of the picture on the screen using the mouse. Up to 10 zoomin's are saved and may be recovered by Zoomout.</td>
</tr>
<tr>
<td>Zoomout</td>
<td>Recovers Zoomin's pictures in a reversed order, (i.e. last saved first recovered).</td>
</tr>
<tr>
<td>Dynamic View</td>
<td>Starts dynamic viewing of the model.</td>
</tr>
<tr>
<td>Windows</td>
<td>Opens new windows or deletes existing ones.</td>
</tr>
<tr>
<td>Repaint</td>
<td>Repaints the window, (i.e. replots the plotted entities with the active settings).</td>
</tr>
</tbody>
</table>

Table 3.11 gives brief description of the setting viewing commands.

**Table 3.11  View Setting Commands in the Viewing Menu**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Status</td>
<td>Displays the active color set and plotting status.</td>
</tr>
<tr>
<td>Fill Color</td>
<td>Specifies the filling color for subsequent element plots.</td>
</tr>
<tr>
<td>Boundary Opts</td>
<td>Sets the flags for boundary plotting and evaluation for subsequent element plots. For fine meshes, better plots are usually obtained if model boundary is used in postprocessing plots. Evaluating the boundary could be slow but usually generates nicer postprocessing plots.</td>
</tr>
<tr>
<td>Hide Opts</td>
<td>Sets the options for hidden line removal for subsequent element plots. The Z-buffer produces better quality but is slower than depth-sort, specially for hardcopies. Used only if Hidden is active.</td>
</tr>
<tr>
<td>Hidden</td>
<td>Activates or deactivates edge hiding for element plots only (not for geometry plots).</td>
</tr>
<tr>
<td>Shaded</td>
<td>Activates or deactivates element shading for subsequent element plots.</td>
</tr>
<tr>
<td>Config Dview</td>
<td>Sets options for subsequent dynamic viewing.</td>
</tr>
</tbody>
</table>
Analysis Options

For every type of analysis, there is a submenu that specifies the desired options. The Analysis Menu tree is repeated here for convenience.

*Figure 3.15  Analysis Menu Tree*
Figure 3.15  Analysis Menu Tree  (Continued)
Figure 3.15  Analysis Menu Tree  (Continued)

Verify the options that you desire before submitting the model for analysis. The following sections list and give notes on the use of various options.
Static Analysis

There are five options for running the Linear Static module (STAR). Three options are in the Static submenu, and the fourth is the P-Method option. The P-Method option is only used for: 6-node TRIANG, 8-node PLANE2D, and 10-node TETRA10 elements.

The P-Method

The P-Method can be activated or deactivated from the P-Method submenu. The P-Method takes more time (for the same number of elements) but should give more accurate results. With the P-Method coarser elements may be used, since higher polynomial orders are used for displacement shape functions. It should be noted that the P-Method works for the Static analysis only and should be turned off before running frequency or buckling analyses.

Thermal, Gravity, and Centrifugal Loadings

Thermal, gravity, and centrifugal loadings are considered as special loadings in COSMOS/M CAD Interface. Mechanical loads like forces, moments, pressure, and prescribed displacements are always considered when defined. Thermal, gravity, and centrifugal forces need to be activated before running the analysis to be considered in addition to the mechanical loads. Otherwise they are just ignored during analysis.

For thermal loading, the coefficient of thermal expansion must be specified in the property set. Temperatures at some nodes must also be specified or may be read from a subsequent thermal run using the Read Temp command from the Load Options submenu. A reference temperature may be specified, or else 0 temperature is considered as reference.

For gravity loading, density or mass elements must be defined in addition to accelerations. The program calculates the masses and multiplies it by the accelerations to get the gravity forces.

For centrifugal loading, density and/or mass elements must be defined in addition to acceleration and angular velocities. The program calculates the mass of every element and hence the equivalent centrifugal forces.

In-Plane Stiffness

For many slender structures with in-plane loading, the membrane forces alter the bending stiffness. Compressive in-plane forces decrease the bending stiffness, and tensile in-plane forces increase bending stiffness. This effect can be demonstrated by the fact that a plate, or a beam would deform less in the presence of tensile forces. A pre-stressed concrete beam is initially in tension and its deformation under lateral loads is much less than a regular similar concrete beam. On the other hand, compression forces reduce the stiffness and hence increase deflections. The fact that structures become more stiff in the presence of tensile in-plane loads is known as stress stiffening. Similarly, the fact that structures become less stiff in the presence of compressive in-plane loads is known as stress softening.

If the in-plane effects are considered, the stiffness of the structure will be a function of loads as well as deflections which themselves are function of loads. An accurate solution requires the use of nonlinear techniques since deflections are needed to calculate the contributions of the in-plane effect to the terms of the stiffness matrix. A reasonable accuracy however can be obtained using the differential stiffness approach. In this approach, the problem is
approximated by calculating a geometric stiffness matrix and adding it to the regular stiffness matrix. The approach is approximate because the deflections used to calculate the geometric stiffness matrix do not consider the effect of in-plane loading.

When the in-plane loading flag is activated, the sequence of steps for the solution will be as follows:

a. Calculate deflections ignoring the effect of in-plane loading on the bending stiffness.

b. Use the calculated deflections to calculate the geometric stiffness matrix.

c. Add the geometric stiffness matrix to the original matrix and solve for deflections one more time.

Steps b and c are not used if the in-plane loading flag is off.

**The Soft Spring Flag**

The whole structure as well as all its parts should be stable to be properly analyzed. The user should specify sufficient displacement constraints to stabilize the structure, otherwise either the whole structure or some of its parts may translate or rotate without resistance in one or more directions and the solution will fail.

If the soft spring flag is active, a soft spring is used to attach every node to the ground. The effect of adding these soft springs is to stabilize the model. A zero diagonal term in the stiffness matrix means that the corresponding degree of freedom has no resistance in that direction. As a result, the deformation will be infinite even for a very small load in that direction and the numerical solution algorithm fails. Adding a soft spring gives small resistances to all degrees of freedom and hence prevents infinite deformations.

If the structure is not stable, a message to that effect stating that a given equation has a zero or negative diagonal term, will be printed in the output file (.OUT). The first thing that the user should check is that coincident nodes that should have been merged were actually merged. This is particularly needed when meshing is done inside COSMOS/M CAD Interface. Second, the user should list and plot the displacement boundary conditions to verify that sufficient displacement boundary conditions were specified. Third, the user should check all element attributes, like element groups, material and real constant sets. As a last resort, the user should turn on the soft spring flag and submit the analysis. Animation of the deformation will show some excessive motions which should help the user fix the boundary condition problem. After fixing the problem, the soft spring flag should be turned off and the analysis should be performed one more time. It is suggested that solutions with the soft spring should not be considered final but should rather be used to correct the model as indicated above.

Coincident nodes are not automatically merged in COSMOS/M CAD Interface because it may be desired to allow coincident nodes along the interface of two entities to move relative to each other.

**The Stress Option**

When this flag is turned on, stresses are automatically calculated upon successful displacement calculations; the stress flag is active by default. If the flag is turned off, the program calculates displacements only and does not calculate stresses. Stresses however can be calculated using the Stress submenu.
**Solver Type**

Choose whether you want to use the direct or the FFE Iterative Solver.

**Multiple Load Cases**

The multiple load case option is an important feature in STAR, the linear static analysis module of COSMOS/M, that is very popular among users. You may define loading conditions for up to 50 load cases and the program will calculate the displacements and stresses in a single run. All results will be available simultaneously for all load cases.

When using multiple load cases, please note the following:

1. Loads are always associated with the active load case. Load case 1 is the default active load case, other load cases are created using the Activate command from the Properties menu.
2. Special loading effects like gravity, centrifugal, and thermal are only considered if activated.
3. Gravitational and centrifugal loadings are associated with the active load case at the time of their definition.
4. Thermal loading is only available in load case number 1.
5. If gap elements exist in the model, the program will only solve the active load case even though other load cases exist in the database. Due to the nonlinearity of gap problems, superposition of load cases may not hold.
6. Multiple load cases are only available in STAR. If the user runs a nonlinear, buckling, or frequency analysis with in-plane loading effects, then only the active load case is considered for such analyses even though other load cases exist in the database.

**Frequency Analysis**

The user has the option to choose the Direct (traditional) or the FFE Solver to perform frequency analysis.

For the FFE Solver, the user can choose whether to use the first or the second order FFE Solver. Either the number of frequencies or the low and upper bounds of frequencies can be specified for FFE frequency analysis.

For the Direct solver, there are six options for running the Linear Frequency analysis (DSTAR) module. These options are the solution method, Sturm Sequence check, Eigenvalue shift, In-plane loading effects, Soft spring, and a flag to use lumped or consistent mass matrix.

**The Solution Method**

Two solution methods are available, the Subspace method and the Lanczos method. The Subspace method is generally more accurate but is slower and requires more resources. The Lanczos on the other hand is faster, less accurate and requires much less resources. For very stiff structures, Lanczos is preferred over Subspace. The Subspace method should be used in cases of smaller problems where a large number of natural frequencies is desired.
Sturm Sequence Check

When the Sturm Sequence flag is activated, the program will check for missing modes, and gives a message in the output file. If the flag is turned off, no check will be done for missing modes.

Eigenvalue Shift

If the structure is not stable, rigid body modes will exist. A free-free beam for example will have six rigid body modes (3 translations and 3 rotations). The frequency for all rigid body modes is zero. If the system is not stable, and the user is interested in the rigid body modes, then the eigenvalue shift flag should be turned on. In that case the program will calculate and use a convenient shift value to avoid singularities and calculate the rigid body modes.

In-Plane Stiffness

In-plane forces affect the natural frequencies of slender bodies. Varying the tension in a guitar string for example varies its tone; the higher the tension, the higher the frequency of the tone. Calculations of natural frequencies and mode shapes are done by DSTAR which calls STAR for calculating and assembling the stiffness matrix. If the In-plane effect is considered, STAR will calculate the geometric stiffness matrix as described in the previous section.

Lumped or Consistent Mass

This flag instructs DSTAR on how to formulate the mass matrix. The lumped mass formulation is simpler and results in a diagonal mass matrix. The consistent mass matrix formulation uses the displacement shape functions to calculate a generally full mass matrix. If the user intends to use the resulting frequencies and mode shapes for the calculation of response under dynamic loads using ASTAR, then the lumped mass matrix formulation should be used.

Buckling Analysis

There are four options for running the Linearized Buckling analysis. The calculations are done by DSTAR which calculates buckling load factors. To calculate the buckling loads for a given mode, the user should multiply the resulting eigenvalues by all loads. A negative sign in the eigenvalue indicates that all loads must reverse their directions for buckling in that mode to be realized. Except for the solution method, the rest of the options are similar to the case of frequency calculations. The four options are the solution method, eigenvalue shift, soft spring and sturm sequence. The solution method is described below, the other options are similar to the frequency analysis.

The Solution Method

Two solution methods are available, the Inverse Power method, and the Subspace method. The Inverse power method calculates one buckling load factor only. The Subspace iterations method on the other hand may be used to calculate multiple buckling modes.
**Stress Analysis**

The Stress menu has one option which turns the calculations of principal stresses on or off. Other stress components are automatically calculated. Principal stresses are only calculated if requested. Element strains as well as nodal and element stresses are calculated. The von Mises stress gives an overall idea about the stress status and may be used to check against yield.

The von Mises stress component is calculated from the stress components as shown below.

\[
VON = \left( \frac{1}{2} \right) \left[ (SX - SY)^2 + (SX - SZ)^2 + (SY - SZ)^2 \right] + 3(TXY^2 + TXZ^2 + TYZ^2) \]

where:

- \( VON \) = von Mises stress component
- \( SX \) = normal stress in the x-direction
- \( SY \) = normal stress in the y-direction
- \( SZ \) = normal stress in the z-direction
- \( TXY \) = shear stress in the x-y plane
- \( TXZ \) = shear stress in the x-z plane
- \( TYZ \) = shear stress in the y-z plane

\( VON \) may also be expressed in terms of principal stresses \( P1, P2, P3 \) as given below:

\[
VON = \left( \frac{1}{2} \right) \left[ (P1 - P2)^2 + (P1 - P3)^2 + (P2 - P3)^2 \right]^{1/2}
\]

**Thermal Analysis**

Three options need to be specified before running thermal analysis. The first option specifies the analysis type to be steady state or transient. The second option specifies the solver and order of elements to be used, and the third activates or deactivates the restart option.

**The Analysis Type**

The user should highlight the steady state option or the transient option. If the transient option is considered, the user should use the Times command in the Load Options submenu in the Loads/BConds menu to specify the starting time, ending time, and time increment for the solution. Results will be available at all specified time steps.

**Solver Type**

Two modules are available for solving thermal problems, namely, HSTAR which uses a direct equation solver based on Gauss Elimination, and FFE Thermal which uses the COSMOS/M FFE Solver, a superior special iterative solver developed by Structural Research and may be, in many cases, more than 100 times faster in comparison with classical direct solvers. The
direct solver however may be faster for transient problems with large number of steps and temperature-independent material properties.

First and second order elements are provided in use with the FFE Thermal. The element group must be of first order when running FFE Thermal but if the 2nd Ord FFE Solver option is used, then parabolic elements are internally used. The 1st Ord FFE Solver will result in using linear elements. As an example, you may not currently use element group TETRA10 with FFE Thermal. You may however use TETRA4 and choose the 2nd Ord FFE Solver.

**The Restart Option**

In many practical situations, the user needs to continue a transient analysis from a previous run. If the restart flag is turned on, the program will build on the results from the last time step and continue. If the restart flag is turned off, previous results will be overwritten. An error message is issued if the restart flag is activated and no previous results exist in the database. The Restart option is placed in the top Analysis menu since it is common to other types of analyses like NSTAR.

**Post Dynamic Analysis**

The Post Dynamic Analysis Menu provides the available options to set up, and run Post Dynamic problems following frequency and mode shape extraction. It is suggested to use the flowcharts throughout the input process.

The following type of post dynamic analyses are available:

a. Response Spectra Analysis
b. Modal Time-History Analysis
c. Response Spectra Generation
d. Random Vibration
e. Harmonic Analysis
f. Static Stress Analysis

Options in the top menu are grouped as follows:

- **P_dyn Analysis Type:** Submenu to specify the analysis type and related input.
- **P_dyn Analysis Curves:** Submenu related to management of post dynamic analysis curves.
- **P_dyn Output Opts:** Specify options for generating the output and plot files.
- **P_dyn Damping:** Submenu related to management of damping.
- **P_dyn Gap:** Submenu related to management of gap elements.
- **Base Excitation:** Submenu related to management of base excitations.
Run Dynamic: Runs dynamic analysis.
P_dyn Prepare: Prepares plotting files.

The natural frequencies and corresponding mode shapes must have been calculated by DSTAR using the lumped-mass option before any post-dynamic analysis may be performed. The type of analysis to be performed must be defined before any other input.

If you need to run more than one type of post dynamic analysis on the same model, it is suggested to copy the database you used for DSTAR to another database name and use a different copy whenever a different type of post-dynamic analysis is to be performed.

**Nonlinear Analysis**

Options for running nonlinear structural analysis are grouped as shown below.

**NL Analysis Options**

The NL Analysis Options command specifies details for the nonlinear analysis to be performed by the NSTAR module through the Run Nonlinear command. The details include: the analysis type (static or dynamic), stiffness matrix reformation interval, equilibrium iterations interval, the maximum allowable number of iterations within a solution step, convergence tolerance used for equilibrium iterations, inclusion of special loading (gravity loading, thermal, centrifugal), displacement-dependent loading option, flag for end moment calculation for shell elements under pressure loading, local constraint penalty stiffness value to be used in the calculation of the penalty matrix, constraint tolerance for local prescribed displacements, creep/plasticity strain increment tolerance, and geometry update flag.

**NL Control**

The NL Control menu is used to set the control technique (Force, Displacement, or Arc-Length) and the iterative method (Modified Newton-Raphson (MNR), Newton-Raphson (NR), or Broyden-Fletcher-Goldfarb-Shanno (BFGS)) to be used in the analysis.

**NL R-Damping**

The NL R-Damping command is used to incorporate Rayleigh (proportional) damping dynamic analysis. The actual damping matrix is calculated by multiplying the mass and stiffness matrices by the specified coefficients and adding the resulting matrices.

**NL Base**

The NL Base command is used to incorporate effects of the uniform base motion acceleration in dynamic analysis (NSTAR). Actual values of acceleration are specified by a multiplier and the associated time curve.

**NL Integration**

The NL Integration command is used to choose the direct time implicit integration schemes incorporated in the dynamic analysis. The user can select either the Newmark-Beta or Wilson-Theta method.
**NL Auto Step**

The NL Auto Step command is used to activate the adaptive automatic stepping option to be used in nonlinear structural analysis (NSTAR). Both static and dynamic options and all control techniques are supported by this command.

**Restart Flag**

If the Restart flag is activated, solution will continue from the last successful solution step. Otherwise existing information will be overwritten.

**Fluid Flow Analysis**

The Run Fluid command performs analysis for fluid flow problems.

**Postprocessing Menu**

Postprocessing refers to the evaluation of results after a successful run. An ASCII output file which contains input and output results is created by the analysis modules. The extension of this file is (.TEM) for thermal analysis, and (.OUT) for all other types of analyses. Options in the Print Options submenu control the information to be recorded in the output file. The output file is not used by the postprocessing commands in COSMOS/M CAD Interface. Instead, binary files are used to read and list or plot the results.

The user may run more than one type of analysis for the same model and database. Postprocessing always refers to the last analysis performed unless the Analysis Type command is used to change it to the desired analysis type. For example the user may run Static analysis to calculate the effect of mechanical loads and Frequency analysis to calculate frequencies and mode shapes. The Analysis Type command can be used to switch to the desired analysis type for postprocessing. The Animate command will animate mode shapes if the frequency analysis is active for postprocessing, or deflection shape if the static analysis is active for postprocessing.

Postprocessing commands may be divided into two types: listing and plotting. The listing commands are straightforward and will list the requested information in the list screens in ASCII format. The listed results may be directed to a file by activating the Log Listing command from Utilities. Plotting should be done in two steps. The first step is to activate the desired quantity and the second is to display it. For example if a contour plot for the resultant displacement is desired, the user should go to the Display Plots submenu and specify the desired options.

Special commands are available for Section plots, Iso Planes and dynamic viewing.

**Animation**

The Animate command in the Plot menu, results in the animation of the current postprocessing plot in the active window. If no plot exists, the deformed shape will be animated for STAR, ASTAR, and NSTAR, and mode shape in the case of DSTAR (buckling...
or frequency). For other types of analyses, one of the following plots must exist in the active window for the Animate command to work.

1. A filled, line, vector contour plot for any quantity.

2. Section plots for 3-D models using the Section Planes command.

3. An iso-surface plot using the Iso Planes command.

The animation represents snap shots of plotted data versus solution steps. If results are only available for one step (linear static analysis, and steady state analyses), results are linearly interpolated for several intermediate frames.

Animation should be performed in the following sequence for deformed shapes:

a. If no postprocessing plot is in the window, you may directly give the Animate command for deformed/mode shape animation.

b. You may plot some other quantity on the deformed shape (a stress component for example).

c. You may use the Animate command to animate the deformed shape. If a quantity is plotted on top of the deformed shape, then the quantity itself as well as the deformed shape are linearly interpolated.

For animation of other data, use the following sequence:

a. Activate the component of your choice for the active analysis type using the appropriate command.

b. Plot the activated data using the corresponding command.

c. You may use the Section Planes to generate a section plot for 3D models.

d. Finally, use the Animate command. All plots present on the active window will be animated simultaneously.

In cases where results are available for several solution steps, the user will be prompted for a pattern of the steps to be used as animation frames. A maximum of 20 frames can be used for animation. Refer to the help for the Animate command in Chapter 5 for more details.

Selection Menu

A selection list is a filter that can be conveniently used in many operations. When no selection list is defined for a particular entity type, then all members of that entity are equally accessed by the interface for all operations, but when a selection list. Selection lists are particularly useful in the pre- and postprocessing of large models and are available for Points, Curves, Contours, Regions, Nodes, and Elements. Several procedures are provided for creating new selection lists or adding to existing ones. As an example, suppose that you are working on a model that has many regions, and you want to fix all nodes associated with these regions in the x-direction. This task may be done in two ways; you may apply the required constraint region by region, or you may create a selection list which contains these regions.
and then apply the desired constraint to all selected regions at once. The selected option in the Displacement submenu and similar menus is provided for this purpose. If geometry is created in a CAD program, then a selection list for curves is automatically created. Entities may be selected using various types of windows in addition to selection by reference. Commands to compliment and initialize selection lists are also provided.

It should be mentioned that stresses at a node are calculated by averaging the results from the elements meeting at that node. For example if node 5 is common to elements 1, 2, and 3, then the program calculates 3 sets of stress components at node 5. One set from each element. The corresponding values in these sets should be close to each other, but are generally different. The value that the program uses for plotting is the average of the 3 sets.

In general, the program uses nodal results from all selected elements meeting at a node to calculate the nodal stresses at that node. If, a selection list is active, then only elements in the selection list will be considered in the averaging process. Selection lists can therefore be used to select the elements that will be used in averaging the stress results.

The Utilities Menu

The Utilities menu contains several useful and convenient commands.

The System Command

The System command opens a shell window where access is provided for all system commands. The user may return to COSMOS/M CAD Interface by typing EXIT.

The Save Command

The database is automatically updated and saved whenever an action command is issued. The session file is an ASCII file with extension (.SES) that contains a chronological record of the commands issued commands. The commands are actually GEOSTAR commands. The SES file is important because it can be used to reconstruct the whole database using the Pro Model command. The SES file must be renamed or copied to a file with extension GEO.

The Status Command

The Status Table is very important because it displays useful information and may be used to control the status of many flags. The Status Table is presented in Figure 3.16.
Figure 3.16 Status Table

The top part gives the active attributes and parameters. The parameters that are currently used are listed below:

- **EG**: Element Group
- **MP**: Material Property set
- **RC**: Real Constant set
- **CS**: Coordinate System
- **LC**: Load Case Number
- **TC**: Time Curve
- **TP**: Temperature Curve
- **MC**: Magnetic Curve

The active sets cannot be changed from the status table.

The first column labeled "PRIM" lists the primitive type or entity type. The second column instructs the program to plot the entity if higher entities are plotted. For example, if the "PLOT" flag is turned on for contours, then contours will be automatically plotted whenever regions, polyhedra, or parts are plotted. The flag is not applicable for PARTS. **VL** refers to Volumes which is an entity used in GEOSTAR but not in COSMOS/M CAD Interface. Surfaces (SF) are only used in conjunction with non-flat regions.

The third column labeled "PCLR" refers to primitive color which gives the user an opportunity to choose the color for every entity type by clicking the mouse in the color location until the desired color shows up.

The fourth column labeled "LABL" refers to entities numbers. If the flag is turned on for nodes, as an example, then whenever nodes are plotted, their labels (numbers) will also be plotted.

The fifth column labeled "LCLR" specifies colors for label plotting.
The sixth column labeled "MAXM" lists the highest number of every entity in the database.

The "KEEP" flag in the seventh column controls the deletion of lower entities when associated higher entities are deleted. For example if you want to delete elements and keep nodes then before deleting the elements you should turn on the "KEEP" flag for nodes. Otherwise nodes will be deleted as well.

The "MARK" flag controls the plotting of the direction of entities. If the flag is turned on for curves, then an arrow will be plotted on each curve to show its direction. For surfaces, an asterisk is plotted on the first parametric direction near the corner from which the second parametric curve originates.

The last column labeled "DMESH" controls default meshing of entities, which means that if it is turned on for regions, as an example, then when the region is reoriented, the elements associated with it are reoriented as well.

Region orientation is particularly important for stress calculations for shell elements. If two elements next to each other do not have the same orientation, then the definition of top and bottom fibers will not be consistent and the wrong stresses are averaged. Plotting shell elements with active shading is very helpful to verify proper orientations.

### The Output Model Command

The Output Model command generates an ASCII file with extension (.GFM) that describes the model. It is a good practice to generate this file and keep it for your models since it can be used to reconstruct the database through the Read Geofile command. Another ASCII file that can also be used to rebuild a model, is the (.SES) file, but should first be renamed to (.GEO) before calling the Read Geofile command.

### Hard Copies

The Hard Copy submenu may be used to generate hardcopies for the figure on the screen. An option is provided to store the picture in a file that may be printed in a later stage. The supported devices are HP Paint Jet, HP Laser Jet, HP Desk Jet, the HP7475 Plotter, and any device with a PostScript card.

### Measure

The Measure submenu may be used to perform geometry and mesh measurement including distance, length, area, and angle.

### Flow Chart and What Next?

Refer to Chapters 1, 2, and 5.
Loading Existing Databases

Existing COSMOS/M option databases may be loaded using the "COSMOS/M Dbase" in the COSMOS/M CAD Interface Top menu. GEO files may be read using the Read Geofile option. COSMOS/M CAD Interface closes the files of the current model and loads the selected database. The options can be used as desired to switch from one model to another.

Size Limitations

The following table lists size limitations for various entities and parameters.

Table 3.12 Entities and Parameters Size Limitations

<table>
<thead>
<tr>
<th>Entity/Parameter</th>
<th>Size Limits</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>64k Version</td>
</tr>
<tr>
<td>Element Groups</td>
<td>5,000</td>
</tr>
<tr>
<td>Material Property sets</td>
<td>90</td>
</tr>
<tr>
<td>Real Constant sets</td>
<td>5,000</td>
</tr>
<tr>
<td>Nodes</td>
<td>64,000</td>
</tr>
<tr>
<td>Elements</td>
<td>64,000</td>
</tr>
<tr>
<td>Degrees of freedom:</td>
<td></td>
</tr>
<tr>
<td>STAR and DSTAR</td>
<td>200,000</td>
</tr>
<tr>
<td>HSTAR (1 DOF/node)</td>
<td>64,000</td>
</tr>
<tr>
<td>FFE</td>
<td>unlimited</td>
</tr>
<tr>
<td>Number of eigenpairs:</td>
<td></td>
</tr>
<tr>
<td>Subspace method</td>
<td>100</td>
</tr>
<tr>
<td>Lanczos method</td>
<td>150</td>
</tr>
<tr>
<td>FFE</td>
<td>200</td>
</tr>
<tr>
<td>Coordinate systems</td>
<td>5,000</td>
</tr>
<tr>
<td>Points</td>
<td>24,000</td>
</tr>
<tr>
<td>Curves</td>
<td>24,000</td>
</tr>
<tr>
<td>Contours</td>
<td>2,000</td>
</tr>
<tr>
<td>Regions</td>
<td>2,000</td>
</tr>
<tr>
<td>Polyhedra</td>
<td>100</td>
</tr>
<tr>
<td>Parts</td>
<td>100</td>
</tr>
<tr>
<td>Curves in a contour</td>
<td>500</td>
</tr>
<tr>
<td>Contours in a region</td>
<td>120</td>
</tr>
<tr>
<td>Regions in a polyhedra</td>
<td>2,000</td>
</tr>
<tr>
<td>Polyhedra in a part</td>
<td>60</td>
</tr>
<tr>
<td>Postprocessing isoplanes</td>
<td>12</td>
</tr>
<tr>
<td>Postprocessing sections</td>
<td>12</td>
</tr>
<tr>
<td>Number of interface windows</td>
<td>5</td>
</tr>
<tr>
<td>Number of colors</td>
<td>256</td>
</tr>
</tbody>
</table>

128k version is available only for Windows platforms
256k version is available only for Unix platforms
Verification Problems

A large number of verification problems is included in the COSMOS/M directory for various types of analyses. The problems are selected to check various analysis and modeling capabilities for different types of analyses. The "GEO" files for all verification problems are included in the COSMOS/M directory. The files have prefixes related to the analysis type as follows:

<table>
<thead>
<tr>
<th>Prefix Alphabets</th>
<th>Analysis Type</th>
</tr>
</thead>
<tbody>
<tr>
<td>B</td>
<td>Linear buckling analysis</td>
</tr>
<tr>
<td>D</td>
<td>Linear dynamic response analysis</td>
</tr>
<tr>
<td>F</td>
<td>Frequency (modal) analysis</td>
</tr>
<tr>
<td>ND</td>
<td>Nonlinear Dynamic analysis</td>
</tr>
<tr>
<td>NS</td>
<td>Nonlinear Static analysis</td>
</tr>
<tr>
<td>S</td>
<td>Linear static analysis</td>
</tr>
<tr>
<td>TL</td>
<td>Thermal Linear analysis</td>
</tr>
<tr>
<td>TN</td>
<td>Thermal Nonlinear analysis</td>
</tr>
</tbody>
</table>

All verification problems are placed in a subdirectory called PROBS that branches from the COSMOS directory. The following is a brief description of the various verification problems.

**Steady State Thermal Analysis**
- TL01- STEADY STATE HEAT CONDUCTION IN A SQUARE PLATE
- TL02- STEADY STATE HEAT CONDUCTION IN AN ORTHOTROPIC PLATE
- TL03- TRANSIENT HEAT CONDUCTION IN A LONG CYLINDER
- TL04- THERMAL STRESSES IN A HOLLOW CYLINDER
- TL05- HEAT CONDUCTION DUE TO A SERIES OF HEATING CABLES
- TL06A- PRESSURE DISTRIBUTION IN AN AQUIFER FLOW
- TL06B- UNCONFINED AQUIFER (AXISYMMETRIC ELEMENTS)
- TL07A- POTENTIAL FLOW AROUND A CYLINDER (STREAM FUNCTION)
- TL08- TRANSIENT HEAT CONDUCTION IN A SLAB OF CONSTANT THICKNESS
- TL09- HEAT TRANSFER FROM A COOLING FIN
- TL10- THERMO ELECTRIC COUPLING

**Transient Thermal Analysis**
- TN01- HEAT CONDUCTION WITH TEMPERATURE DEPENDENT CONDUCTIVITY
- TN02- TRANSIENT HEAT CONDUCTION (NONLINEAR ANALYSIS)
- TN03- RADIATION USING RLINK
- TN04- TEMPERATURE ANALYSIS WITH PHASE CHANGE

**Linear Static Analysis**
- S1- PIN JOINTED TRUSS
- S2- LONG THICK-WALLED CYLINDER
- S3A, S3B- SIMPLY SUPPORTED RECTANGULAR PLATE
- S5- THERMAL STRESS ANALYSIS OF A 2-D STRUCTURE
- S6- DEFLECTION OF A CANTILEVER BEAM
- S7- BEAM STRESSES AND DEFORMATIONS
- S8- TIP DISPLACEMENTS OF A CIRCULAR BEAM
- S12- DEFLECTION OF A HINGED SUPPORT
S13- STATICALLY INDETERMINATE REACTION FORCE ANALYSIS
S14A, S14B- SPACE TRUSS WITH VERTICAL LOAD
S17- RECTANGULAR PLATE UNDER TRIANGULAR THERMAL LOADING
S19- HOLLOW THICK-WALLED CYLINDER SUBJECT TO TEMPERATURE
S20- CYLINDRICAL SHELL ROOF
S22- THERMALLY LOADED SUPPORT STRUCTURE
S23- THERMAL STRESS ANALYSIS OF A FRAME
S25- TORSION OF A SQUARE BOX BEAM
S26- BEAM WITH ELASTIC SUPPORTS AND A HINGE
S27- FRAME ANALYSIS WITH COMBINED LOADS
S28- CANTILEVER UNSYMMETRIC BEAM
S30- EFFECT OF TRANSVERSE SHEAR ON MAXIMUM DEFLECTION
S33A, S33M- SUBSTRUCTURE OF AN AIRPLANE WING
S34- TIE ROD WITH LATERAL LOADING
S36A, S36M- SUBSTRUCTURE OF A SIMPLY SUPPORTED PLATE
S38- ROTATING SOLID DISK
S39- UNBALANCED ROTATING FLYWHEEL
S41- REACTIONS AND DEFLECTIONS OF A CANTILEVER BEAM
S44A, S44B- BENDING OF A CIRCULAR PLATE WITH A CENTER HOLE
S45- ECCENTRIC FRAME
S48- ROTATION OF A TANK OF FLUID (PLANE2D FLUID)
S50A, S50B, S50C, S50D, S50E, S50F, S50H- DEFLECTION OF A CURVED BEAM
S55- LATERALLY LOADED TAPERED BEAM
S57- TEST OF A PINCHED CYLINDER WITH DIAPHRAGM (SHELL9)
S58, S58B- DEFLECTION OF A TWISTED BEAM WITH TIP FORCE
S59A, S59B, S59C- SANDWICH SQUARE PLATE UNDER UNIFORM LOADING (SHELL9L)
S61- SINGLE-EDGE CRACKED BEND SPECIMEN
S62- PLATE WITH CENTRAL CRACK
S64A, S64B- CYCLIC SYMMETRY
S65- FLUID-STRUCTURE SYMMETRY, ROTATION OF A TANK OF FLUID
S66- FLUID-STRUCTURE SYMMETRY, ACCELERATION OF A TANK OF FLUID
S67- MACNEAL-HARDER TEST
S68- P-METHOD SOLUTION OF A SQUARE PLATE WITH HOLES
S69- P-METHOD ANALYSIS OF AN ELLIPTIC MEMBRANE UNDER PRESSURE LOADING
S70- THERMAL ANALYSIS WITH TEMPERATURE DEPENDENT MATERIAL
S74- CONSTANT STRESS PATCH TEST (TETRA4R)
S78- P-ADAPTIVE ANALYSIS OF A SQUARE PLATE WITH A CIRCULAR HOLE

Natural Frequency (Modal) Analysis

F1- NATURAL FREQUENCIES OF A TWO-MASS SPRING SYSTEM
F2- FREQUENCIES OF A CANTILEVER BEAM
F3- FREQUENCY OF A SIMPLY SUPPORTED BEAM
F4- NATURAL FREQUENCIES OF A CANTILEVER BEAM
F5- FREQUENCY OF A CANTILEVER BEAM WITH LUMPED MASS
F6- DYNAMIC ANALYSIS OF A 3-D STRUCTURE
F7- DYNAMIC ANALYSIS OF A SIMPLY SUPPORTED PLATE
F8- CLAMPED CIRCULAR PLATE
F9- FREQUENCIES OF A CYLINDRICAL SHELL
F10- SYMMETRIC MODES AND NATURAL FREQUENCIES OF A RING
F11- EIGENVALUES OF A TRIANGULAR WING
F12- VIBRATION OF AN UNSUPPORTED BEAM
F14- NATURAL FREQUENCY OF FLUID
F16A, F16B- VIBRATION OF A CLAMPED WEDGE
F17- LATERAL VIBRATION OF AN AXIALLY LOADED BAR
F19- LOWEST FREQUENCIES OF CLAMPED CYLINDRICAL SHELL
F20A, F20B, F20C, F20G- DYNAMIC ANALYSIS OF CANTILEVER BEAM
Chapter 3   Exploring COSMOS/M CAD Interface

F21- FREQUENCY ANALYSIS OF CIRCULAR CANAL OF FLUID WITH VARIABLE DEPTH
F23- NATURAL FREQUENCY OF FLUID IN A MANOMETER

Buckling Analysis
B1- INSTABILITY OF COLUMNS
B2- INSTABILITY OF COLUMNS
B3- INSTABILITY OF COLUMNS
B6- BUCKLING ANALYSIS OF A SMALL FRAME
B7- INSTABILITY OF FRAMES
B9- SIMPLY SUPPORTED STIFFENED PLATE

Post Dynamic Analysis
D1- MODAL TIME HISTORY OF A CANTILEVER BEAM
D2- TRANSIENT RESPONSE OF AN S.S. BEAM TO A CONSTANT FORCE WITH A FINITE RISE TIME
D3- RESPONSE SPECTRUM GENERATION FOR A GROUND ACCELERATION
D4- SEISMIC RESPONSE OF A THREE-STORY FRAME
D5- SEISMIC RESPONSE OF A FIVE-STORY FRAME
D6- FRAME SUBJECTED TO RANDOM GROUND MOTION RANDOM VIBRATION
D7- THREE-STORY FRAME SUBJECTED TO RANDOM GROUND MOTION (RANDOM VIBRATION)
D8- FREQUENCY RESPONSE OF A TWO DEGREE-OF-FREEDOM SYSTEM (HARMONIC ANALYSIS)
D9- TRANSIENT RESPONSE OF A DROPPED CONTAINER (INCLUDING A GAP ELEMENT)
D10- FREQUENCY ANALYSIS OF AN SDOF SYSTEM (COULOMB DAMPING)
D11- FREQUENCY ANALYSIS OF AN SDOF SYSTEM (COULOMB DAMPING)
D12- FREQUENCY RESPONSE OF A TWO DEGREE-OF-FREEDOM SYSTEM
D17- MULTI-BASE MOTION

Nonlinear Structural Analysis
Nonlinear Static Analysis
NS1- ELASTOPLASTIC COMPRESSION OF A COMPOSITE PIPE ASSEMBLY
NS2- NONLINEAR ANALYSIS OF A CABLE ASSEMBLY
NS3- STATIC COLLAPSE OF A TRUSS STRUCTURE
NS4- TRUSS WITH TEMPERATURE DEPENDENT MATERIAL PROPERTIES
NS5- ELASTIC-PLASTIC STATIC ANALYSIS OF A METAL SHEET
NS6- ELASTOPLASTIC ANALYSIS OF A THICK WALLED TUBE
NS7- LARGE DEFLECTION ANALYSIS OF AN INFINITELY LONG PLATE
NS8- STATIC LARGE DISPLACEMENT ANALYSIS OF A CANTILEVER BEAM
NS9- STATIC LARGE DISPLACEMENT ANALYSIS OF A SPHERICAL SHELL
NS10- LARGE DISPLACEMENT ANALYSIS OF A FIXED BEAM WITH CONCENTRATED LOAD
NS11- SIMPLY SUPPORTED RECTANGULAR PLATE, LARGE DEFLECTION ANALYSIS
NS12- LARGE DEFLECTION ANALYSIS OF A CANTILEVER
NS19- LARGE DISPLACEMENT ANALYSIS OF A S.S. CIRCULAR PLATE
NS20- LARGE DEFLECTION ANALYSIS OF A FIXED-FIXED SHALLOW ARCH
NS21- ELASTOPLASTIC SMALL DISPLACEMENT ANALYSIS OF A CANTILEVER BEAM WITH TIP MOMENT
NS22- CLAMPED SQUARE PLATE WITH PRESSURE LOADING
NS23- LARGE DISPLACEMENT STATIC ANALYSIS OF A CLAMPED SANDWICH PLATE
NS24- PLATE SUBJECTED TO TRIANGULAR TEMPERATURE LOADING
NS25- ANALYSIS OF A HOLLOW THICK-WALLED CYLINDER SUBJECTED TO TEMPERATURE AND PRESSURE LOADING
NS26- THERMAL STRESS ANALYSIS OF A PLATE - TEMP. DEPENDENT MATERIAL PROPERTIES
NS27- UNIAXIAL CREEP STRAIN IN A BAR (CYCLIC LOADING)
NS28- CREEP ANALYSIS OF A CYLINDER SUBJECT TO CYCLIC INTERNAL PRESSURE
Running a Verification Problem

To run a selected verification problem, follow the steps below:

1. Copy the corresponding file to your current subdirectory. Remember that all files have the extension GEO.
2. Type COSPRO and press Enter.
3. Select Read Geofile from the startup menu. Type the file name without extension and press Enter.
4. COSMOS/M CAD Interface will construct the model, and run the analysis.
5. You may examine the output file (extension TEM for thermal analysis, and OUT for all other analysis types).
6. You may proceed to list and display the corresponding results.

**GEOSTAR Command Mode**

The GEOSTAR Command mode switches control from the interface to GEOSTAR which is the pre- and postprocessor for the COSMOS/M finite element system. Full but silent access is provided to all GEOSTAR capabilities; full in the sense that all GEOSTAR commands of version 1.75 can be called, and silent because these commands have to be typed in and may not be called from menus. No help is available inside the interface for GEOSTAR commands.

In the following pages, some important GEOSTAR commands are presented. The following table lists the selected commands and their usages:

<table>
<thead>
<tr>
<th>Command</th>
<th>Usages</th>
</tr>
</thead>
<tbody>
<tr>
<td>EGROUP</td>
<td>Gives access to element groups not directly supported in the Define Element Group menu.</td>
</tr>
<tr>
<td>CSYS and CSANGLE</td>
<td>Defines and activates a coordinate system that may be used for defining forces, moments, and displacement boundary conditions.</td>
</tr>
<tr>
<td>EPROPCHANGE</td>
<td>Changes the element group, material property set, or real constant set associated with elements.</td>
</tr>
</tbody>
</table>

**The EGROUP Command**

The EGROUP command defines an element group and specifies the related options. A maximum of 20 element groups can be defined in a model. The defined element group becomes active and stays so until another group is created, or the Activate command is used to activate another group. The prompted options depend on the selected element group (refer to Chapter 5 for more information). Note that this command can be used to activate any element group in COSMOS/M as shown in Table 3.13.

**Thin Versus Thick Shells**

Thin shell elements (SHELL3 and SHELL4) are based on the Discrete Kirchoff Theory (D.K.T.). The results always converge to the thin plate theory.

Thick shell elements (SHELL3T and SHELL4T) are based on Mindlin's thick-plate theory. They perform well for shells that are more than moderately thick. The model includes the effects of shear deformation and rotary inertia and makes the following assumptions:
a. Deflection of the mid-surface of the plate is small in comparison to the plate thickness.
b. Transverse normal stresses are negligible.
c. Normals to the plate mid-surface in the initial configuration remain straight but not necessarily normal to the mid-surface after deformation.

The following parameters could be used as a guideline for choosing between thick and thin shells:

1. \((L/t) = a\)
   
   where: 
   
   \(L\) is the global plate structure length 
   \(t\) is the plate thickness

2. \((A/t)^{(1/2)} = b\)
   
   where: 
   
   \(A\) is the area of the element.

Use thin shells if \(a > 20\) or \(b > 3\), otherwise use thick shells.

Creating Coordinate Systems

The only coordinate system that can be used directly in the interface is system 0, the global Cartesian coordinate system. Two commands for the creation of coordinate systems are presented here. There are three types of coordinate systems that may be created: Cartesian, Cylindrical, and Spherical. Systems 0, 1, and 2 are readily defined in GEOSTAR and refer to the global Cartesian, Cylindrical, and Spherical systems respectively. Up to 498 additional local coordinate systems may be defined by the user and each system may be defined as Cartesian, Cylindrical, or Spherical.

Coordinate systems may be used to define forces, moments, and displacement constraints in other than global directions. Such loading and boundary conditions use the active coordinate system for determining directions. For example if a node is constrained in the x-direction, while system 1 is active, then it means that the radial displacement has been constrained for that node.

The steps you need to follow to constrain the displacement in a given direction which is not one of the Cartesian directions is as follows:

a. If the desired direction is contained in systems 1 or 2, use the ACTSET command to activate the relevant system. Systems 1 and 2 are predefined in the program.

b. If a new system need to be defined, then use the CSYS or CSANGLE command to create it.

c. Make sure that the desired coordinate system is active using the Status Table in the Utilities menu.

d. Use the Define option in the Displacement submenu from the Loads/BConds menu to apply the constraints.

e. List or plot displacement constraints to verify your input.
f. If you need to apply loads, or constraints in other directions, repeat the above steps.

Note that the X-, Y-, and Z-directions are used in a generalized sense. For example in a cylindrical system X is radial, and Y is tangential.

The CSYS Command

The CSYS command defines a local coordinate system based on 3 specified keypoints. This system becomes the currently active coordinate system.

**CSYS**

**Input Description:**
- Coordinate system label. Must be between 3 and 500.  
  (default is highest defined + 1)
- Type of local coordinate system.  
  = 0 Cartesian coordinate system.  
  = 1 Cylindrical coordinate system.  
  = 2 Spherical coordinate system.  
  (default is 0)
- Keypoint at the origin of the coordinate system.
- Keypoint on the X-axis of the coordinate system.
- Keypoint on the X-Y plane of the coordinate system.

The CSANGLE Command

The CSANGL command defines a local coordinate system based on specified origin location and three rotations about the global Cartesian axes. This system becomes the currently active coordinate system.

**CSANGLE**

**Input Description:**
- Coordinate system label. Must be between 3 and 500.  
  (default is highest defined + 1)
- Type of local coordinate system.  
  = 0 Cartesian coordinate system.  
  = 1 Cylindrical coordinate system.  
  = 2 Spherical coordinate system.  
  (default is 0)
- X-coordinate of the origin of the new coordinate system.  
  (default is 0)
- Y-coordinate of the origin of the new coordinate system.
  (default is 0)

- Z-coordinate of the origin of the new coordinate system.
  (default is 0)

- Angle of rotation about the global X-direction. (Use right hand rule)
  (default is 0)

- Angle of rotation about the global Y-direction. (Use right hand rule)
  (default is 0)

- Angle of rotation about the global Z-direction. (Use right hand rule)
  (default is 0)

- Order of applying rotations.
  = 0 Rotations are applied in the order X, Y, then Z.
  = 1 Rotations are applied in the order Z, Y, then X.
  (default is 0)

The **EPROPCHANGE Command**

In the systematic meshing procedure in COSMOS/M CAD Interface, the user activates the proper element attributes before meshing the corresponding entities. The EPROPCHANGE may be used to correct mistakes in associating attributes with groups of elements. The element attributes that can be changed are EG (element group), RC (real constant), MP (material property), or ECS (element coordinate system). It is convenient to create an element selection list that contains the elements whose attributes need to be changed before issuing the EPROPCHANGE command.

**EPROPCHANGE**

**Input Description:**

- Beginning element in the pattern.
  (default is 1)

- Ending element in the pattern.
  (default is highest defined)

- Increment between the elements.
  (default is 1)

- Set name. Admissible set names are:
  = EG Element group.
  = RC Real constant set.
  = MP Material property set.
  = ECS Element coordinate system.
  (default is EG)

- Set label to be assigned.
  (default is previously defined + 1)
- Color of elements. (Refer to Status table)
  (default is 3)

**ECS Attribute Usages**

The default element coordinate system (labeled -1) refers to the local coordinate system defined by the nodal connectivity of the element. Chapter 5 provides detailed information about the default ECS for every element type. The user may choose to change the ECS to some other coordinate system for the following reasons:

a. Define material angles and properties for orthotropic materials.

b. Calculate the stresses in a user-selected coordinate system.

**Calculation of Stresses in a Particular ECS**

By default, all stresses are calculated in the global Cartesian coordinate system because the flag in the element group options that controls the coordinate system for stress calculations is set to 0 for global. If the user wants to use some other coordinate system for stress calculations, then that flag should be changed to 1 for local. Once the flag is set to local, the ECS determines the coordinate system that will be used for stress calculations for each element in that group.

As an example if the user wants to use the predefined cylindrical system (system 1) for stress calculations, then the ECS should be changed to 1 using the Activate command before meshing or the EPROPCHANGE command after meshing. The flag in the element group (EGROUP command) should be set to 1 for local. After running the analysis, the resulting stresses will be in coordinate system 1. In the plotting and listing of results, the X-direction will refer to the radial direction, and the Y-direction will refer to the tangential.

The user may create and use any coordinate system for stress calculations.

**Database Files**

This section lists all files that are used and/or generated by COSMOS/M CAD Interface modules. Most of the files are transparent to the user where no interaction is needed with them. A brief description is given for the contents of each file. All files for a given problem will be created with the same name as the problem name (pn) and extensions as shown.

**Preprocessing Files**

**GEOSTAR MODULE**

The most-dealt-with files by the user are:

1. pn.SES Stores history of all action commands issued during a session.
2. pn.GFM Is a geometric or neutral format file similar to the session file.
3. pn.OUT Details the analysis input, results and error messages.
The geometric entities information is placed in the following files:

1. pn.GPT for keypoints.
2. pn.GCR for curves.
3. pn.GSF for surfaces.
4. pn.GVL for volumes.
5. pn.GCT for contours.
6. pn.GRG for regions.
7. pn.CSY for coordinate systems.
8. pn.GPH for polyhedra.
9. pn.GPA for parts.

The finite element information is stored in the following files:

1. pn.NOD for nodes.
2. pn.ELE for elements.
3. pn.LOD for loads.
4. pn.EPR for pressure loads.
5. pn.EGR for element groups.
6. pn.MAT for material properties.
7. pn.RLC for real constants.
8. pn.MAS for communication with other modules.
9. pn.RUF for debugging information.
10. pn.GEN for general information.
11. pn.LIS for list and help commands (default name). See the Log Listing command in the Properties menu.
12. pn.I?? for saving image files (default name).
    (? = 0, 1, 2, ..., 9)
13. pn.H?? for saving HALO files (default name).
    (? = 0, 1, 2, ..., 9)
14. pn.P?? for saving images in PostScript format (default name).
    (? = 0, 1, 2, ..., 9)
15. pn.M?? for saving images in meta format (default name).
    (? = 0, 1, 2, ..., 9)
16. pn.D?? for saving images in DXF/2D format (default name).
    (? = 0, 1, 2, ..., 9)

**STAR MODULE (Linear Static analysis)**

**Input files:**

1. pn.MAS Master control
2. pn.NOD Nodes
3. pn.CSY Coordinate systems
4. pn.ELE Elements
5. pn.EGR Element groups
6. pn.RLC Real constant sets
7. pn.MAT Material property sets
8. pn.ILC Element special loadings
9. pn.SUB Substructures
10. pn.ITC Temperature curves
11. pn.ICT Constraints
12. pn.LOC Prescribed displacements and local constraints
13. pn.DIS Displacements for stress calculations
14. pn.IDA ID array
15. pn.LDS  Load vectors
16. pn.ELM  Grouped element information
17. pn.MSF  Information for frequency analysis

Output files:
18. pn.OUT  ASCII Output file
19. pn.LCD  Displacements for plotting
20. pn.STF  Working file for storing the stiffness matrix
21. pn.DIA  Diagonal terms of stiffness matrix

STRESS MODULE (Stress analysis)

Input files:
1. pn.MAS  Master control
2. pn.NOD  Nodes
3. pn.CSY  Coordinate systems
4. pn.ELE  Elements
5. pn.EGR  Element groups
6. pn.RLC  Real constant sets
7. pn.MAT  Material property sets
8. pn.SUB  Substructures
9. pn.ITA  Time curves
10. pn.DYN  Post
11. pn.LOC  Prescribed constraints
12. pn.DIS  Displacements
13. pn.IDA  ID array
14. pn.ELM  Element information
15. pn.FRC  Fixed end forces for beam loading
16. pn.LCD  Displacements for stress calculation

Output files:
17. pn.OUT  ASCII output file
18. pn.AIN  AISC code check
19. pn.STE  Element and nodal stresses for linear static analysis
20. pn.STP  Element and nodal stresses for post dynamics

RENUMBER MODULE

Input files:
1. pn.MAS  Master control
2. pn.NOD  Nodes
3. pn.ELE  Elements
4. pn.ICT  Constraints

Output files:
5. pn.RNM  Binary output for renumbered nodes
6. pn.REN  Binary file for renumbered nodes

DSTAR MODULE (Frequency and Buckling analyses)

Input files:
1. pn.MAS  Master control
2. pn.NOD  Nodes
3. pn.IDA  ID array
4. pn.MAS   Frequency analysis

**Output files:**
5. pn.OUT   ASCII output
6. pn.EIG   Eigenfunctions (values and vectors)
7. pn.LCM   Eigenfunctions for postprocessing

**HSTAR MODULE (Thermal analysis)**

**Input files:**
1. pn.MAS   Master control
2. pn.NOD   Nodes
3. pn.CSY   Coordinate systems
4. pn.ELE   Elements
5. pn.EGR   Element groups
6. pn.RLC   Real constant sets
7. pn.MAT   Material property sets
8. pn.ITC   Time/Temp curves
9. pn.TR1   Surface relation for view factor in GEOSTAR
10. pn.TR2  Node information for view factor in GEOSTAR
11. pn.INI  Initial conditions
12. pn.EPR  Heat generation rates
13. pn.DSP  Node temperature
14. pn.HRT  Restart
15. pn.WRK  Temporary unformatted work unit
16. pn.BFG  Unformatted unit required for BFGS vectors
17. pn.ICT  Constraint information
18. pn.JB1  Joint-bond information
19. pn.JB2  Joint-bond information
20. pn.LOD  Load cases
21. pn.RSD  Convergence information
22. pn.VWF  Radiation view factor

**Output files:**
23. pn.TEM  ASCII Output
24. pn.HTO  Heat transfer results for graphical display

**NSTAR MODULE (Nonlinear Static and Dynamic analysis)**

**Input files:**
1. pn.MAS   Master control
2. pn.NOD   Nodes
3. pn.CSY   Coordinate systems
4. pn.ELE   Elements
5. pn.EGR   Element group sets
6. pn.RLC   Real constant sets
7. pn.MAT   Material property sets
8. pn.PSR   Element pressure
9. pn.INI   Initial conditions (displacement, velocity and acceleration)
10. pn.ILC  Centrifugal and gravity loading
11. pn.ICT  Constraints
12. pn.LDS  Mechanical load vectors written in single precision corresponding to various time curves
13. pn.IDA  ID array
14. pn.LDT  List of flags from PRE1 prescribed displacements and reaction forces
15. pn.NTP  Nodal temperatures for different time curves
16. pn.ELM  Grouped element
17. pn.HTO  File written in HSTAR with nodal temperatures at different time steps
18. pn.NP1  Variables for nonlinear material models
19. pn.NP2  Variables for nonlinear material models
20. pn.STF  Stiffness matrix is written in blocks
21. pn.CBM  Used for Lagrangian beam
22. pn.CBS  Used for Lagrangian beam
23. pn.BML  Beam stress and strains at different integration points
24. pn.STR  Stress and strain information for PLANÈ2D (QM6)
25. pn.BFG  Updating vectors for stiffness matrix in BFGS method
26. pn.GAP  Gap information
27. pn.WMT  Water motion table
28. pn.SDM  Rayleigh damping stiffness at time=0
29. pn.RES  Results verification (read by NSTAR)
30. pn.ITC  Time/temperature/stress-strain curve
31. pn.MSH  Contact related information
32. pn.PLN  Restart related information
33. pn.DSP  Local boundary conditions
34. pn.JB2  Joint-bond information
35. pn.MSF  Save mass/stiffness for other module(s) (use the same dummy unit number)
36. pn.DIA  Save mass/stiffness for other module(s) (use the same dummy unit number)

Output files:
37. pn.OUT  ASCII output file, detailed information, disp, stress and strain output
38. pn.LCN  Nodal response (displacement, velocity and acceleration)
39. pn.STN  Element/Nodal stress/force/strain
40. pn.PL3  Nodal displacements for plot files at user specified time steps
41. pn.PL4  Nodal stress, strain information at user specified time steps

ASTAR MODULE (Post Dynamic analysis)

Input files:
1. pn.MAS  Master control
2. pn.NOD  Nodes
3. pn.CSY  Coordinate systems
4. pn.LOD  Prescribed nodal forces
5. pn.INI  Initial conditions
6. pn.DYN  Master control
7. pn.CVC  Load curves
8. pn.FPS  Nodal forces
9. pn.PIN  Initial conditions
10. pn.GDS  Gaps and dampers
11. pn.IDA  ID array
12. pn.MSF  Frequency analysis
13. pn.EIG  Eigenfunctions from frequency analysis
14. pn.RLD  Dummy file (Response Spectrum Analysis)
15. pn.RES Mode displacements (Response Spectrum Analysis)
16. pn.BAS Base motion information (Time-History Analysis)
17. pn.MDP Modal response (Time-History Analysis)
18. pn.VBS Base PSD information (Random Vibration Analysis)
19. pn.VSD PSD of modal response (Random Vibration Analysis)
20. pn.VMS RMS of modal response (Random Vibration Analysis)
21. pn.HBS Base motion information (Harmonic Analysis)
22. pn.HRS Modal response (Harmonic Analysis)
23. PHI.DAT Dummy file
24. SPEC.DUM Dummy file

**Output files:**
25. pn.OUT ASCII output
26. pn.PLT Graphs
27. pn.LAB Labels for the PLT file
28. pn.PSV Pseudo velocity spectra
29. pn.PSA Pseudo acceleration spectra
30. pn.RLD Relative displacement spectra
31. pn.RLV Relative velocity spectra
32. pn.ABA Absolute acceleration spectra
33. pn.CR1 1st generated time history
34. pn.CR2 2nd generated time history
35. pn.CR3 Corrected 1st generated time history
36. pn.CR4 Corrected 2nd generated time history

**OPTSTAR MODULE (Design Optimization)**

**Input files:**
1. pn.MAS Master control
2. pn.DVA Design variables
3. pn.DCO Behavior constraints
4. pn.DOB Objective function

**Output files:**
5. pn.OPT ASCII output

**Database Utility**

The COSMOS/M database utility program consists of an object library, link information file, several sample problems and a READ.ME file. The utility enables the user to communicate with the COSMOS/M database files. Read and write routines are provided and the user may call them by his or her own programs to read from or write to COSMOS/M database files.

For more information about this section, please contact Structural Research.
(This page is intentionally left blank)
Chapter 4

Element Library
Introduction

This chapter contains a full description of the elements used in the COSMOS/M CAD Interface. These elements represent a subset of the more extensive library available in the full COSMOS/M analysis system. The elements in COSMOS/M CAD Interface are selected with two goals in mind: maintain simplicity and cover most of the needs of design engineers.

Table 4.1 shows a list of the elements available in the interface. The element names are the same names used in the Define Group submenu of the Properties Menu. For details on the elements used in the nonlinear structural analysis (NSTAR), refer to the COSMOS/M CAD Interface Advanced Modules Manual.

<table>
<thead>
<tr>
<th>Element Name</th>
<th>Element Description</th>
<th>No. of Nodes</th>
<th>DOF/Node</th>
<th>Geometry Association</th>
<th>Needed Attributes</th>
<th>Supported Analyses</th>
</tr>
</thead>
<tbody>
<tr>
<td>TETRA4</td>
<td>Linear Tetrahedral w/o rotations</td>
<td>4</td>
<td>3</td>
<td>Volume (3D)</td>
<td>EG, MP</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TETRA4R</td>
<td>Linear Tetrahedral with rotations</td>
<td>4</td>
<td>6</td>
<td>Volume (3D)</td>
<td>EG, MP</td>
<td>S, T, A</td>
</tr>
<tr>
<td>TETRA10</td>
<td>Quadratic Tetrahedral</td>
<td>10</td>
<td>3</td>
<td>Volume (3D)</td>
<td>EG, MP</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL3</td>
<td>3-Node Thin Shell</td>
<td>3</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL3T</td>
<td>3-Node Thick Shell</td>
<td>3</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL4</td>
<td>4-Node Thin Shell</td>
<td>4</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL4T</td>
<td>4-Node Thick Shell</td>
<td>4</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL6</td>
<td>6-Node Thin Shell</td>
<td>6</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL9</td>
<td>Quadratic Shell</td>
<td>8 or 9</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A</td>
</tr>
<tr>
<td>TRIANG</td>
<td>Plane or Axi. Triangular</td>
<td>3 to 6</td>
<td>2</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>PLANE2D</td>
<td>Plane or Axi. Quadrilateral</td>
<td>4 to 8</td>
<td>2</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TRUSS2D</td>
<td>2D Truss</td>
<td>2</td>
<td>2</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>TRUSS3D</td>
<td>3D Truss</td>
<td>2</td>
<td>3</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>BEAM2D</td>
<td>2D Beam</td>
<td>2</td>
<td>3</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>BEAM3D</td>
<td>3D Beam</td>
<td>3</td>
<td>6</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>CLINK</td>
<td>Convection Link</td>
<td>2</td>
<td>1</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>T</td>
</tr>
<tr>
<td>RLINK</td>
<td>Radiation Link</td>
<td>2</td>
<td>1</td>
<td>Curve (1D)</td>
<td>EG, RC</td>
<td>T</td>
</tr>
<tr>
<td>SPRING</td>
<td>Axial Spring</td>
<td>2</td>
<td>3</td>
<td>Curve (1D)</td>
<td>EG, RC</td>
<td>S, N</td>
</tr>
<tr>
<td>GAP</td>
<td>Gap Element</td>
<td>2</td>
<td>3</td>
<td>Curve (1D)</td>
<td>EG, RC</td>
<td>S</td>
</tr>
<tr>
<td>MASS</td>
<td>Concentrated Mass</td>
<td>1</td>
<td>6</td>
<td>Point (0D)</td>
<td>RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL3L</td>
<td>Composite 3-Node Shell</td>
<td>3</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, A</td>
</tr>
<tr>
<td>SHELL4L</td>
<td>Composite 4-Node Shell</td>
<td>4</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, T, A, N</td>
</tr>
<tr>
<td>SHELL9L</td>
<td>Composite 8/9-Node Shell</td>
<td>8 or 9</td>
<td>6</td>
<td>Region (2D)</td>
<td>EG, MP, RC</td>
<td>S, A</td>
</tr>
<tr>
<td>SHELLAX</td>
<td>Axisymmetric Shell</td>
<td>2</td>
<td>4</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, A</td>
</tr>
<tr>
<td>PIPE</td>
<td>Elastic Straight Pipe</td>
<td>2</td>
<td>6</td>
<td>Curve (1D)</td>
<td>EG, MP, RC</td>
<td>S, A</td>
</tr>
<tr>
<td>FLOW2D</td>
<td>2D Flow</td>
<td>4</td>
<td>3</td>
<td>Region (2D)</td>
<td>EG, MP</td>
<td>F</td>
</tr>
<tr>
<td>FLOW3D</td>
<td>3D Flow</td>
<td>8</td>
<td>4</td>
<td>Volume (3D)</td>
<td>MP</td>
<td>F</td>
</tr>
</tbody>
</table>

Where:
EG: Element Group
MP: Material Property
RC: Real Constant Set
T: Thermal Analysis
F: Fluid Flow Analysis
A: Linear Dynamic Analysis (ASTAR)
S: Linear Static, Frequency, Buckling and Stress Analyses
N: Static and Dynamic Nonlinear Structural Analysis (NSTAR)

Note: For thermal analysis, TRUSS3D is identical to BEAM3D, and TETRA4 is identical to TETRA4R.
Element Attributes and Element Generation

Elements may be generated in a CAD program or in the interface. In the interface, elements may be generated in one of two ways: direct definition using the Define Element command, or through meshing. The correct association must be specified for the associated entity type when using the direct definition (see Table 4.1 and help for Define Element command).

If an element is generated in the interface, then it will assume the active element attributes which are Element Group (EG), Material Property (MP), and Real Constant Set (RC). There is only one active EG, MP, and RC at any moment and the only effect they have is that whenever a new element is generated, it will assume these active attributes. The analysis routine will consider all element groups defined with their associated RC and MP. For example, you may have SHELL4, BEAM3D, TETRA4R, SPRING,...etc. in the same model. List elements to verify correct association.

An appropriate element group and real constant set must be activated before meshing. As an example you may not mesh a part if the active element group is not TETRA4, TETRA4R, or TETRA10.

If the mesh is generated in a CAD program, all element attributes will be automatically defined when the model is imported into the interface. The element group, and associated real constant, and material property sets will be readily defined with the default options. Making changes to these attributes is possible in the interface if default options are not desired.

An element group must be defined for each element before a successful analysis run can be made. Real constants and material property sets may or may not be required. Table 4.1 lists the required attributes for each element group.

In the COSMOS/M CAD Interface, 1D elements may be generated by meshing curves, or using the CR (curves) option for the associated entity if the Define Element command is used.

2D elements may be generated by meshing regions or polyhedrons, or using the RG (regions) option for the associated entity if the Define Element command is used.

3D elements may be generated by meshing parts, or using the VL (volumes) option for the associated entity if the Define Element command is used.

Figure 4.1 shows pictorial representations of elements available in the COSMOS/M CAD Interface. Other COSMOS/M elements not available in the COSMOS/M CAD Interface Element Group Menu may be defined by the Egroup command in the GEOSTAR Command mode.

For a detailed description of elements used in NSTAR (the nonlinear structural module), refer to the COSMOS/M CAD Interface Advanced Modules Manual.
### Concentrated Mass
- **Element Name:** MASS
- **Nodes:** 1

### Truss/Spar
- **Element Name:** TRUSS2D or TRUSS3D
- **Nodes:** 2

### Convection Link
- **Element Name:** CLINK
- **Nodes:** 2

### Radiation Link
- **Element Name:** RLINK
- **Nodes:** 2

### Beam
- **Element Name:** BEAM2D or BEAM3D
- **Nodes:** 3

### Spring
- **Element Name:** SPRING
- **Nodes:** 2

### Linear Gap
- **Element Name:** GAP
- **Nodes:** 2

### 4-Node Plane or Axisymmetric Quadrilateral
- **Element Name:** PLANE2D
- **Nodes:** 4

### 8-Node Plane or Axisymmetric Quadrilateral
- **Element Name:** PLANE2D
- **Nodes:** 8

### 3-Node Plane or Axisymmetric Triangle
- **Element Name:** TRIANG
- **Nodes:** 3

### 6-Node Plane or Axisymmetric Triangle
- **Element Name:** TRIANG
- **Nodes:** 6

### 3-Node Shell
- **Element Name:** SHELL3 or SHELL3T
- **Nodes:** 3

### 4-Node Shell
- **Element Name:** SHELL4 or SHELL4T
- **Nodes:** 4

### 6-Node Thin Shell
- **Element Name:** SHELL6
- **Nodes:** 6

### 8/9-Node Thin Shell
- **Element Name:** SHELL9
- **Nodes:** 8/9

### 4-Node Tetrahedral Solid
- **Element Name:** TETRA4
- **Nodes:** 4

---

**Figure 4.1** Pictorial Representation of Elements Supported by COSMOS/M CAD Interface
### 4-Node Tetrahedral Solid w/Rotational DOFs
- Element Name: TETRA4R
- Nodes: 4

### 10-Node Tetrahedral Solid
- Element Name: TETRA10
- Nodes: 10

### 2-Node Axisymmetric Shell
- Element Name: SHELLAX
- Nodes: 2

### Straight Pipe
- Element Name: PIPE
- Nodes: 2

### 3-Node Composite Shell
- Element Name: SHELL3L
- Nodes: 3

### 4-Node Composite Shell
- Element Name: SHELL4L
- Nodes: 4

### 8/9-Node Composite Shell
- Element Name: SHELL9L
- Nodes: 8/9

### Fluid Flow 2D 4-Node Isoparametric
- Element Name: FLOW2D
- Nodes: 4

### Fluid Flow 3D 8-Node Isoparametric
- Element Name: FLOW3D
- Nodes: 8

**Figure 4.1 Pictorial Representation of Elements Supported by COSMOS/M CAD Interface**

(Continued)

### Definition of Element Group

The **Define Group** option from the Element Groups submenu in the Properties Menu must be used to specify the element group. The defined group becomes active upon definition. It is suggested that you always define the element group before defining real constant sets.

The **List Groups** and **Delete Groups** options may be used to list and delete element groups respectively. If a mistake is made in defining the correct element group, repeat the **Define Group** command and give the correct label and name. You may change the active element group by the **Activate** command. Up to 20 different element groups may be defined in a given model.
**Definition of Real Constant Sets**

Real constants are defined by the **Define Real Constant** option from the Real Constants submenu in the Properties Menu. The real constant set becomes active upon definition. The **Define Real Constant** command asks for the associated element group and then prompts for corresponding entries. For example, if the associated element group was specified as 2 and element group 2 has been defined to be TRUSS3D elements, then the program will prompt for the area of the cross section. If element group 2 has been defined as SHELL3, then the program will prompt for thickness rather than area.

The **List Real Constant** and **Delete Real Constant** options may be used to list and delete real constant sets respectively. If a mistake is made in defining the proper real constant set, define it again and give the proper label and name. You may change the active real constant set by redefining the set to be activated or by using the **Activate** command. The start location index in the **Define** command may be used to redefine the values of just few real constants. Some element groups as shown in Table 4.1 do not require real constant sets. Up to 5000 real constant sets can be defined in a given model. The EPROPCHANGE command from the GEOSTAR Command mode may be used to change the real constant associations of existing elements.

**Definition of Material Property Sets**

Material properties are defined by the **Define Property** option from the Material submenu in the Properties Menu. The material property set becomes active upon definition. Materials may also be picked from the COSMOS/M material library. The **List Material** and **Delete Material** options may be used to list and delete material property sets respectively. If a mistake is made in defining the correct material property, redefine it again and give the correct label, property name and value. You may change the active material set by redefining the set to be activated or by using the **Activate** command. Up to 90 material property sets can be defined in a given mode for all modules except NSTAR where the limit is 20. The EPROPCHANGE command from the GEOSTAR Command mode may be used to change the material property set association of existing elements.

**Definition of Element Coordinate Systems**

The default element coordinate system used in **COSMOS/M CAD Interface** is ‘-1’ which means that the node connectivity is used to define the ECS. The details of how the default ECS is defined for each element group is described in the following sections. The ECS is important in two ways: 1) to define orthotropic material properties and, 2) to obtain stresses in a given coordinate system. The **Activate** command in the Properties Menu may be used to set the ECS to any existing coordinate system. The **Elem Prop Change** command from the Properties Menu may be used to change the element coordinate system association of existing elements.

**An Important Note**

When you import the mesh from a CAD program, the element group options will be set to the default ones. If you need to change one or more of these options, then you must redefine the
corresponding element group in COSMOS/M CAD Interface and input the desired changes. List the element groups to verify the changes before submitting the model for analysis.
Thermal and Linear Structural 3D 4- or 10-Node Tetrahedron Solid
element_name TETRA4 or TETRA10

General Description
TETRA4 is a 4-node and TETRA10 is a 10-node three dimensional tetrahedral solid element for structural and thermal problems. Three translational degrees of freedom per node are considered for structural analysis. Only one degree of freedom per node, representing the temperature, is used for the thermal module.

The nodal input pattern for this element is shown in Figures 4-2 and 4-3. Both clockwise and counter-clockwise node numbering are allowed. All midside nodes should fall within the middle third of the element edges.

Special Features
Buckling, Geometric stiffness consideration (inplane loading flag), Adaptive P-Method (polynomial degrees up to 4).

Default Element Coordinate System (ECS = -1)
The element x-axis goes from the first node to the second. The element y-axis lies in the plane defined by the three nodes (1, 2, 3), perpendicular to the x-axis toward node 3. The element z-axis completes a right-hand Cartesian system with x- and y-axes.

Element Group Options
- **Op. 1:** Unrelated option to this type of analysis (use default value)
- **Op. 2:** Unrelated option to this type of analysis (use default value)
- **Op. 3:** Unrelated option to this type of analysis (use default value)
- **Op. 4:** Use the default value (stress printout in the global Cartesian coordinate system)
- **Op. 5:** Use the default value (linear elastic material)
- **Op. 6:** Use the default value (small displacement formulation)
- **Op. 7:** Use the default value (material creep is not considered)
- **Op. 8:** Unrelated option to this type of analysis (use default value)

Real Constants
Nine constants have to be defined in case of using orthotropic or anisotropic models. The nine values are used to determine the coordinates of three points (1`, 2`, 3`) to define the material coordinate system. The first direction of the material coordinate system, denoted by "a" in Figure 4.2, is defined by a vector connecting point 1` to point 2`. The b-axis (second material direction) lies in the plane of the three defined points and goes from a-axis toward the third point. The c-axis (third material direction) completes a right-hand Cartesian coordinate system. The above material directions can be defined with respect to the defined element coordinate system as specified by the E Coord Sys option in the Activate command. You do not need to define these constants if material directions coincide with the global coordinate system or the active local coordinate system defined by ECS.

\[ r1 = \text{x-coordinate of point } 1` \]
Material Properties

1. Isotropic and Orthotropic Materials

(See Figure 4.2 for material directions)

- \( r_2 \) = y-coordinate of point 1
- \( r_3 \) = z-coordinate of point 1
- \( r_4 \) = x-coordinate of point 2
- \( r_5 \) = y-coordinate of point 2
- \( r_6 \) = z-coordinate of point 2
- \( r_7 \) = x-coordinate of point 3
- \( r_8 \) = y-coordinate of point 3
- \( r_9 \) = z-coordinate of point 3

- \( \text{EX} \) = Modulus of elasticity in the 1st material direction
- \( \text{EY} \) = Modulus of elasticity in the 2nd material direction
- \( \text{EZ} \) = Modulus of elasticity in the 3rd material direction
- \( \text{KX} \) = Thermal conductivity in the global X-direction
- \( \text{KY} \) = Thermal conductivity in the global Y-direction
- \( \text{KZ} \) = Thermal conductivity in the global Z-direction
- \( \nu_{xy} \) = Poisson's ratio relating the 1st and 2nd material directions (strain in the 2nd direction due to unit strain along the 1st direction)
- \( \nu_{yz} \) = Poisson's ratio relating the 2nd and 3rd material directions (strain in the 3rd direction due to unit strain along the 2nd direction)
- \( \nu_{xz} \) = Poisson's ratio relating the 1st and 3rd material directions (strain in the 3rd direction due to unit strain along the 1st direction)
- \( \alpha_{px} \) = Coefficient of thermal expansion in the 1st material direction
- \( \alpha_{py} \) = Coefficient of thermal expansion in the 2nd material direction
- \( \alpha_{pz} \) = Coefficient of thermal expansion in the 3rd material direction
- \( C \) = Specific heat
- \( \rho \) = Density
- \( G_{xy} \) = Shear modulus relating the 1st and 2nd material directions
- \( G_{yz} \) = Shear modulus relating the 2nd and 3rd material directions
- \( G_{xz} \) = Shear modulus relating the 1st and 3rd material directions
- \( \text{DAMP} \) = Material damping coefficient
- \( \text{ECONX} \) = Electric conductivity (thermal analysis only)

Note:

The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:

1. Moduli of elasticities in at least two directions are defined and are unequal.
2. Poisson's ratio in at least two planes are defined and are unequal.
3. Thermal coefficients in at least two directions are defined and are unequal.
4. Thermal conductivity in at least two directions are defined and are unequal.
5. The anisotropic material matrix is defined (MC11, MC12, ..., MC66 material properties).

The following conditions must be met for proper representation of orthotropic properties for the \( i^{th} \) and \( j^{th} \) material directions:

\[
\frac{\nu_{ij}}{E_j} = \frac{\nu_{ji}}{E_i}
\]
Where $\nu_{ij}$, $E_i$, and $E_j$ are provided as input and $\nu_{ji}$ calculated internally by the program.

2. General Anisotropic Material:

MC11, MC12, ….., MC66 (Total of 21 entries) to define either the anisotropic material stiffness or compliance matrix in the material coordinate system:

\[
\begin{bmatrix}
MC_{11} & MC_{12} & MC_{13} & MC_{14} & MC_{15} & MC_{16} \\
MC_{22} & MC_{23} & MC_{24} & MC_{25} & MC_{26} \\
MC_{33} & MC_{34} & MC_{35} & MC_{36} \\
MC_{44} & MC_{45} & MC_{46} \\
MC_{55} & MC_{56} \\
Sym. & MC_{66}
\end{bmatrix}
\]

The 21 material properties can be used also to define isotropic and orthotropic material properties.

It should be noted that the anisotropic material matrix elements overrides any other defined (related) material properties.

**Element Loadings**

- Thermal
- Gravitational
- Pressure

**Output Results**

Stress components in the global or local coordinate directions including the von Mises stress are available at the center and at the nodes of the element. Principal stresses may also be optionally requested at the element center.

---

**Figure 4.2  4-Node Tetrahedron Element**

<table>
<thead>
<tr>
<th>Predictions</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>XYZ:</td>
<td>Global Cartesian Coordinate System</td>
</tr>
<tr>
<td>xy:</td>
<td>Element Coordinate System</td>
</tr>
<tr>
<td>1 2 3 4:</td>
<td>Face numbers for Applying Loads and Boundary Conditions (pressure is positive when applied inward)</td>
</tr>
</tbody>
</table>
Figure 4.3 10-Node Tetrahedron Element

References
Linear Structural 3D 4-Node Tetrahedron Solid with Rotation  
element_name TETRA4R

General Description
TETRA4R is a 4-node three-dimensional tetrahedral solid element for the analysis of structural models. Three translational and three rotational degrees of freedom are considered per node, allowing the element to be easily connected to BEAM3D and all SHELL elements.

The nodal input pattern for this element is shown in Figure 4.4. Both clockwise and counterclockwise node numbering are allowed.

Special Features
Adaptive H-method.

Default Element Coordinate System (ECS = -1)
The element x-axis goes from the first node to the second. The element y-axis lies in the plane defined by the three nodes (1, 2, 3), perpendicular to the x-axis toward node 3. The element z-axis completes a right-hand Cartesian system with x- and y-axes.

Element Group Options
- **Op. 1 to Op. 3**: Unused options for this element
- **Op. 4**: Stress printout
  
  - \( = 0 \): Global direction
  - \( = 1 \): Local direction

- **Op. 5 to Op. 8**: Unused options for this element

Real Constants
Nine constants have to be defined in case of using orthotropic or anisotropic models. The nine values are used to determine the coordinates of three points \((1^\prime, 2^\prime, 3^\prime)\) to define the material coordinate system. The first direction of the material coordinate system, denoted by "a" in Figure 4.4, is defined by a vector connecting point \(1^\prime\) to point \(2^\prime\). The b-axis (second material direction) lies in the plane of the three defined points and goes from a-axis toward the third point. The c-axis (third material direction) completes a right-hand Cartesian coordinate system. The above material directions can be defined with respect to the defined element coordinate system as specified by the **E Coord Sys** option in the **Activate** command. You do not need to define these constants if material directions coincide with the global coordinate system or the active local coordinate system defined by ECS.

\[
\begin{align*}
  r_1 &= \text{x-coordinate of point } 1^\prime \\
  r_2 &= \text{y-coordinate of point } 1^\prime \\
  r_3 &= \text{z-coordinate of point } 1^\prime \\
  r_4 &= \text{x-coordinate of point } 2^\prime \\
  r_5 &= \text{y-coordinate of point } 2^\prime \\
  r_6 &= \text{z-coordinate of point } 2^\prime \\
  r_7 &= \text{x-coordinate of point } 3^\prime \\
  r_8 &= \text{y-coordinate of point } 3^\prime \\
  r_9 &= \text{z-coordinate of point } 3^\prime
\end{align*}
\]
Material Properties

1. Isotropic and Orthotropic Materials

*(See Figure 4.4 for material directions)*

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>Modulus of elasticity in the 1st material direction</td>
</tr>
<tr>
<td>EY</td>
<td>Modulus of elasticity in the 2nd material direction</td>
</tr>
<tr>
<td>EZ</td>
<td>Modulus of elasticity in the 3rd material direction</td>
</tr>
<tr>
<td>KX</td>
<td>Thermal conductivity in the X-direction of the global Cartesian coordinate system</td>
</tr>
<tr>
<td>KY</td>
<td>Thermal conductivity in the Y-direction of the global Cartesian coordinate system</td>
</tr>
<tr>
<td>KZ</td>
<td>Thermal conductivity in the Z-direction of the global Cartesian coordinate system</td>
</tr>
<tr>
<td>NUXY</td>
<td>Poisson's ratio relating the 1st and 2nd material directions (strain in the 2nd direction due to unit strain along the 1st direction)</td>
</tr>
<tr>
<td>NUYZ</td>
<td>Poisson's ratio relating the 2nd and 3rd material directions (strain in the 3rd direction due to unit strain along the 1st direction)</td>
</tr>
<tr>
<td>NUXZ</td>
<td>Poisson's ratio relating the 1st and 3rd material directions (strain in the 3rd direction due to unit strain along the 1st direction)</td>
</tr>
<tr>
<td>ALPX</td>
<td>Coefficient of thermal expansion in the 1st material direction</td>
</tr>
<tr>
<td>ALPY</td>
<td>Coefficient of thermal expansion in the 2nd material direction</td>
</tr>
<tr>
<td>ALPZ</td>
<td>Coefficient of thermal expansion in the 3rd material direction</td>
</tr>
<tr>
<td>DENS</td>
<td>Density</td>
</tr>
<tr>
<td>C</td>
<td>Specific Heat</td>
</tr>
<tr>
<td>GXY</td>
<td>Shear modulus relating the 1st and 2nd material directions</td>
</tr>
<tr>
<td>GYZ</td>
<td>Shear modulus relating the 2nd and 3rd material directions</td>
</tr>
<tr>
<td>GXZ</td>
<td>Shear modulus relating the 1st and 3rd material directions</td>
</tr>
<tr>
<td>DAMP</td>
<td>Material damping coefficient</td>
</tr>
<tr>
<td>ECONX</td>
<td>Electric conductivity (thermal analysis only)</td>
</tr>
</tbody>
</table>

Note:

The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:

1. Moduli of elasticities in at least two directions are defined and are unequal.
2. Poisson's ratio in at least two planes are defined and are unequal.
3. Thermal coefficients in at least two directions are defined and are unequal.
4. Thermal conductivity in at least two directions are defined and are unequal.
5. The anisotropic material matrix is defined (MC11, MC12, ..., MC66 material properties).

The following conditions must be met for proper representation of orthotropic properties for the \(i^{th}\) and \(j^{th}\) material directions:

\[ \nu_{ij}/E_i = \nu_{ji}/E_j \]

Where \(\nu_{ij}\), \(E_i\), and \(E_j\) are provided as input and \(\nu_{ji}\) calculated internally by the program.

2. General Anisotropic Material:

MC11, MC12, ..., MC66 (Total of 21 entries) to define either the anisotropic material stiffness or compliance matrix in the material coordinate system:
The 21 material properties can be used also to define isotropic and orthotropic material properties.

It should be noted that the anisotropic material matrix elements overrides any other defined (related) material properties.

**Element Loadings**
- Thermal
- Gravitational
- Pressure

**Output Result**

Stress components in the global or local coordinate directions including the von Mises stress are available at the center and at the nodes of the element. Principal stresses may also be optionally requested at the element center.

---

**Figure 4.4** 4-Node Tetrahedron Element with Rotation

---
References


Thermal and Linear Structural Triangular Thin Shell
element_name SHELL3

General Description
SHELL3 is a 3-node triangular thin shell element with membrane and bending capabilities for
the analysis of three dimensional structural and thermal models. The shear deformation effect
is neglected for this element. Six degrees of freedom per node (three translations and three
rotations) are considered for structural analysis. Only one degree of freedom per node,
representing the temperature, is used for the thermal module.

The element is assumed to be isotropic with constant thickness for structural problems and
orthotropic for thermal problems. For orthotropic structural materials, SHELL3L may be
considered.

For problems involving thick plates or shells, as determined by standard guidelines,
SHELL3T is recommended. Both of SHELL3T and SHELL3 have identical inputs which
permit exchanging one for the other by simply altering the "element_name."

Special Features

Default Element Coordinate System (ECS = -1)
The nodal input pattern for the element is shown in Figure 4.5. Both clockwise and counter-
clockwise node numbering are allowed. For element coordinate system, the x-axis goes from
the first node to the second. The element y-axis lies in the plane defined by the three nodes,
perpendicular to x-axis toward the third node. The element z-axis completes a right-hand
Cartesian system with x- and y-axes.

Other Element Coordinate Systems (ECS >= -1)
When a defined element coordinate system \((x_e y_e z_e)\), as shown in Figure 4.6, is different from
the default element coordinate system \((ECS = -1)\), the program considers a modified element
coordinate system \((xyz)\) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a
right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is greater than 45° \((\alpha
> 45°)\), the element x-axis is considered as the projection of \(x_e\) on the element plane.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is less than or equal
to 45° \((\alpha <= 45°)\), the element x-axis is considered as the projection of \(y_e\) on the element
plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with
x- and y-axes.

Element Group Options
Op. 1: Unused option for this element
Op. 2: Type of analysis
   = 0 ; Regular shell analysis (membrane + bending)
   = 1 ; Membrane analysis *
   = 2 ; Shear panel analysis (shear terms only)

Op. 3: Stress print-out option
   = 0 ; Print forces per unit length and stresses at center of element
   = 1 ; Add print-out of nodal forces
   = 2 ; Add print-out of nodal stresses

Op. 4: Stress directions
   = 0 ; Calculate stresses in the global Cartesian coordinate system
   = 1 ; Calculate stresses in the defined element local coordinate system

Op. 5: Use default value (linear elastic material type)
Op. 6: Use default value (small displacement formulation)
Op. 7: Unused option for this element
Op. 8: Unused option for this element

* For flat membrane structures with transverse loads, it is recommended that the
  problem start with an assumed deflected shape obtained from the regular shell
  analysis as defined by Op. 2. Since no equilibrium iterations are performed in
  linear analysis, results may not be accurate enough. Nonlinear analysis is
  strongly recommended.

Real Constants

  r1  = Thickness
  r2  = Temperature gradient
  r3  = Foundation stiffness (see Note 2 for SHELL4 element)
  r4  = Unused constant for this element
  r5, r6 = Unrelated constants to this type of analysis (use default value)

Material Properties

  EX  = Modulus of elasticity
  KX  = Thermal conductivity in the x-direction defined by the ECS which must
        be 0, -1, or a user created Cartesian system
  KY  = Thermal conductivity in the y-direction defined by the ECS which must
        be 0, -1, or a user created Cartesian system
  NUXY = Poisson's ratio
  C   = Specific heat
  ALPX = Coefficient of thermal expansion
  DENS = Density
  GXY  = Shear modulus
  DAMP = Material damping coefficient
  ECONX = Electric conductivity (thermal analysis only)

Element Loadings

- Thermal
- Gravitational
- Pressure (applied normal to element faces)
Output Results

Stress components including von Mises stress are available in the element coordinate system at the centroid of the element for top and bottom fibers. Principal stresses may optionally be requested (see STRESS command in the ANALYSIS menu). In addition, nodal force per unit length and stress components can also be calculated and printed (see Op. 3). The directions of force and moment components per unit length for this element are illustrated in Figure 4.7.

Figure 4.5 Triangular Thin Shell
\( x_e \ y_e \ z_e \): Defined element coordinate system (ECS ≠ -1)

\( xyz \): Modified element coordinate system

\( \alpha > 45^\circ \)

\( z = \) normal to the shell plane
\( x = \) projection of \( x_e \) - axis on the shell plane

\( \alpha \leq 45^\circ \)

\( z = \) normal to the shell plane
\( x = \) projection of \( y_e \) - axis on the shell plane

Figure 4.6 Coordinate System Modification for Shell Elements
Figure 4.7  Direction of Force and Moment Components Per Unit Length as Defined by COSMOS/M for Thin Shells

References


Thermal and Linear Structural Triangular Thick Shell

**element_name SHELL3T**

**General Description**

SHELL3T is a 3-node triangular thick shell element with membrane and bending capabilities for the analysis of three-dimensional structural and thermal models. The element accounts for shear deformation effects. Six degrees of freedom per node (three translations and three rotations) are considered for structural analysis. Only one degree of freedom per node, representing temperature, is used for the thermal module.

The element is assumed to be isotropic with constant thickness for structural problems and orthotropic for thermal problems. For orthotropic structural materials, SHELL3L may be used.

For problems involving thin plates or shells, as determined by standard guidelines, SHELL3 is recommended instead. Both of these elements (SHELL3T and SHELL3) have identical inputs which permit exchanging one for the other by simply altering the "element_name."

**Special Features**

Buckling, Inplane loading.

**Default Element Coordinate System (ECS = -1)**

The nodal input pattern for the element is shown in Figure 4.8. Both clockwise and counterclockwise node numbering are allowed for the element coordinate system. The x-axis goes from the first node to the second. The element y-axis lies in the plane defined by the three nodes, perpendicular to x-axis toward the third node. The element z-axis completes a right-hand Cartesian system with x- and y-axes.

**Other Element Coordinate Systems (ECS ≠ -1)**

When a defined element coordinate system \((x_ey_ez_e)\), as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system \((xyz)\) as follows:

- The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

- If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is greater than \(45^\circ\) \((\alpha > 45^\circ)\), the element x-axis is considered as the projection of \(x_e\) on the element plane.

- If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is less than or equal to \(45^\circ\) \((\alpha \leq 45^\circ)\), the element x-axis is considered as the projection of \(y_e\) on the element plane.

- The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.

**Element Group Options**

- **Op. 1:** Unused option for this element
Op. 2: Analysis option
  = 0 ; Regular shell analysis (membrane + bending)
  = 1 ; Membrane analysis *
  = 2 ; Shear panel analysis (shear terms only)

Op. 3: Stress print-out option
  = 0 ; Print forces per unit length and stresses at center of element
  = 1 ; Add print-out of nodal forces
  = 2 ; Add print-out of nodal stresses

Op. 4: Stress directions
  = 0 ; Calculate stresses in the global Cartesian coordinate system
  = 1 ; Calculate stresses in the defined element local coordinate system

Op. 5: Use default value (linear material type)

Op. 6: Use default value (small displacement formulation)

Op. 7: Unused option for this element

Op. 8: Unrelated option to this type of analysis (use default value)
  *
  For flat membrane structures with transverse loads, it is recommended that the
  problem start with an assumed deflected shape obtained from the regular shell
  analysis as defined by Op. 2. Since no equilibrium iterations are performed in
  linear analysis, results may not be accurate enough. Nonlinear analysis is
  strongly recommended.

Real Constants
  r1 = Thickness
  r2 = Temperature gradient
  r3 = Foundation stiffness (see Note 2 for SHELL4 element)
  r4 = Unused constant for this element
  r5, r6 = Unrelated constants for this type of analysis

Material Properties
  EX = Modulus of elasticity
  KX = Thermal conductivity in the x-direction defined by the ECS which must
       be 0, -1, or a user created Cartesian system
  KY = Thermal conductivity in the y-direction defined by the ECS which must
       be 0, -1, or a user created Cartesian system
  NUXY = Poisson's ratio
  C = Specific heat
  ALPX = Coefficient of thermal expansion
  DENS = Density
  GXY = Shear modulus
  DAMP = Material damping coefficient
  ECONX = Electrical conductivity (thermal analysis only)

Element Loadings
  - Thermal
  - Gravitational
  - Pressure (applied normal to element faces)
Output Results

Stress components including von Mises stress are available in the element coordinate systems at the centroid of the element for top and bottom fibers. Principal stresses may optionally be requested (see STRESS command in the ANALYSIS menu). In addition, nodal forces per unit length and stress components can be calculated and printed in the output file (see Op. 3). The directions of force and moment components per unit length for this element are illustrated in Figure 4.9.

Figure 4.8 Triangular Thick Shell
Figure 4.9  Direction of Force and Moment Components Per Unit Length as Defined by COSMOS/M for Thick Shells

References


Thermal and Linear Structural Quadrilateral Thin Shell
element_name SHELL4

General Description
SHELL4 is a 4-node quadrilateral thin shell element with membrane and bending capabilities for the analysis of three dimensional structural and thermal models. The shear deformation effect is neglected. Six degrees of freedom per node (three translations and three rotations) are considered for structural analysis. Only one degree of freedom per node, representing the temperature, is used for the thermal module.

The element is assumed to be isotropic with constant thickness for structural problems and orthotropic for thermal problems. For orthotropic structural materials, SHELL4L may be considered.

For problems involving thick plates or shells, as determined by standard guidelines, SHELL4T is recommended. Both of SHELL4T and SHELL4 have identical inputs which permit exchanging one for the other by simply altering the "element_name."

Special Features
Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)
The nodal input pattern for this element is shown in Figure 4.10. Both clockwise and counterclockwise node numbering are allowed. A triangular element is considered if the third and fourth nodes have the same global node number. For the element coordinate system, the x-axis goes from the first node to the second. The element y-axis lies in the plane defined by the first three nodes, perpendicular to the element x-axis toward the fourth node. The element z-axis completes a right-hand Cartesian system.

Other Element Coordinate Systems (ECS >< -1)
When a defined element coordinate system \((x_e y_e z_e)\), as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system \((xyz)\) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is greater than 45° \((\alpha > 45°)\), the element x-axis is considered as the projection of \(x_e\) on the element plane.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is less than or equal to 45° \((\alpha <= 45°)\), the element x-axis is considered as the projection of \(y_e\) on the element plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.
Element Group Options

Op. 1:  (See Note 1 below)
= 0 ; QUAD 2 element (2 triangles to form a quadrilateral)
= 1 ; QUAD 4 element (4 triangles to form a quadrilateral)
= 2 ; QUAD element (4-node quadrilateral element)
= 3 ; QM6 (4-node quadrilateral element)

Op. 2:  Analysis option
= 0 ; Regular shell analysis (membrane + bending)
= 1 ; Membrane analysis *
= 2 ; Shear panel analysis (shear terms only)

Op. 3:  Stress print-out option
= 0 ; Print forces per unit length and stresses at center of element
= 1 ; Add print-out of nodal forces
= 2 ; Add print-out of nodal stresses

Op. 4:  Stress directions
= 0 ; Calculate stresses in the global Cartesian coordinate system
= 1 ; Calculate stresses in the defined element local coordinate system

Op. 5:  Use default value (linear elastic material)
Op. 6:  Use default value (small displacement formulation)
Op. 7:  Unused option for this element
Op. 8:  Unused option for this element

* For flat membrane structures with transverse loads, it is recommended that the problem start with an assumed deflected shape obtained from the regular shell analysis as defined by Op. 2. Since no equilibrium iterations are performed in linear analysis, results may not be accurate enough. Nonlinear analysis is strongly recommended.

Real Constants

r1 = Thickness
r2 = Temperature gradient
r3 = Foundation stiffness (see Note 2)
r4 = Unused constant for this element
r5, r6 = Unrelated constants to this type of analysis

Material Properties

EX = Modulus of elasticity
KX = Thermal conductivity in the x-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
KY = Thermal conductivity in the y-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
NUXY = Poisson's ratio
C = Specific heat
ALPX = Coefficient of thermal expansion
DENS = Density
GXY = Shear modulus
DAMP = Material damping coefficient
ECONX = Electric conductivity (thermal analysis only)
Element Loadings
- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results
Stress components including von Mises stress are available in the element coordinate systems at the centroid of the element for top and bottom fibers. Principal stresses may optionally be requested (see STRESS command in the ANALYSIS menu). In addition nodal force per unit length and stress components can also be printed (see Op. 3). The directions of force and moment components per unit length for this element are illustrated in Figure 4.7.

Note 1
Quad elements are internally treated as follows:
1. QUAD2 Element
   Formed from two triangles as shown; stiffness matrix calculation is fast; accuracy is good; in symmetric problems it may display slight anti-symmetry behavior.
2. QUAD4 Element

Formed from four triangles as shown; stiffness matrix calculations are slow; accuracy is good; in symmetric problems it shows symmetrical behavior.

Note 2

Elaboration on Real Constant No. 3 (Foundation stiffness, $K_f$)

For a non-zero value of this real constant, a spring is considered at each node, applying a stiffness in the out-of-plane direction (normal to plane of the element). The stiffness for each spring is equal to:

$$K_i = \frac{A \cdot K_f}{n}$$

where:

- $K_i = \text{Normal stiffness at node } i$
- $A = \text{Area of the element}$
- $K_f = \text{Foundation stiffness}$
- $n = \text{Number of the element nodes}$

The stress program calculates the foundation pressure according to:
\[ \sigma_f = K_f \sum_{i=1}^{n} w_i / n \]

where:

\[ w_i = \text{lateral displacement at node } i \]

**Figure 4.13  Foundation Stiffness**

**References**


Thermal and Linear Structural Quadrilateral Thick Shell
element_name SHELL4T

General Description
SHELL4T is a 4-node quadrilateral thick shell element with membrane and bending
capabilities for the analysis of three dimensional structural and thermal models. The element
accounts for shear deformation effects. Six degrees of freedom per node (three translations
and three rotations) are considered for structural analysis. Only one degree of freedom per
node, representing the temperature, is used for the thermal module. A triangular element is
considered if the third and fourth node are assigned the same global node number.

The element is assumed to be isotropic with constant thickness for structural problems and
orthotropic for thermal problems. For orthotropic structural materials SHELL4L may be
used.

For problems involving thin plates or shells, as determined by standard guidelines, SHELL4 is
recommended. Both of SHELL4T and SHELL4 have identical inputs which permit
exchanging one for the other by simply altering the "element_name."

Special Features
Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)
The nodal input pattern for the element is shown in Figure 4.14. Both clockwise and counter-
clockwise node numbering are allowed. For the element coordinate system, the x-axis goes
from the first node to the second. The element y-axis lies in the plane defined by the first
three nodes, perpendicular to the element x-axis toward the fourth node. The element z-axis
completes a right-hand Cartesian system.

Other Element Coordinate System (ECS >< -1)
When a defined element coordinate system \((x_e,y_e,z_e)\), as shown in Figure 4.6, is different from
the default element coordinate system (ECS = -1), the program considers a modified element
coordinate system \((xyz)\) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a
right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is greater than \(45^\circ\) \((\alpha
> 45^\circ)\), the element x-axis is considered as the projection of \(x_e\) on the element plane.

If the angle \((\alpha)\) between the \(x_e\) axis and the normal to the surface \((z)\) is less than or equal
to \(45^\circ\) \((\alpha =< 45^\circ)\), the element x-axis is considered as the projection of \(y_e\) on the element
plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with
x- and y-axes.
Element Group Options

**Op. 1:** Type of shell element (see SHELL4)
- 0: QUAD2 element (2 triangles to form a quadrilateral)
- 1: QUAD4 element (4 triangles to form a quadrilateral)
- 2: QUAD element (4-node quadrilateral element)
- 3: QM6 (4-node quadrilateral element)

**Op. 2:** Analysis option
- 0: Regular shell analysis (membrane + bending)
- 1: Membrane analysis *
- 2: Shear panel analysis (shear terms only)

**Op. 3:** Stress print-out option
- 0: Print forces per unit length and stresses at center of element
- 1: Add print-out of nodal forces
- 2: Add print-out of nodal stresses

**Op. 4:** Stress directions
- 0: Calculate stresses in the global Cartesian coordinate system
- 1: Calculate stresses in the defined element local coordinate system

**Op. 5:** Use default value (linear material type)

**Op. 6:** Use default value (small displacement formulation)

**Op. 7:** Unused option for this element

**Op. 8:** Unrelated option to this type of analysis (use default value)

* For flat membrane structures with transverse loads, it is recommended that the problem start with an assumed deflected shape obtained from the regular shell analysis as defined by Op. 2. Since no equilibrium iterations are performed in linear analysis, results may not be accurate enough. Nonlinear analysis is strongly recommended.

Real Constants

- \( r_1 \) = Thickness
- \( r_2 \) = Temperature gradient
- \( r_3 \) = Foundation stiffness (see Note 2 for SHELL4 element)
- \( r_4 \) = Unused constants for this element
- \( r_5, r_6 \) = Unrelated constants for this type of analysis

Material Properties

- \( EX \) = Modulus of elasticity
- \( KX \) = Thermal conductivity in the x-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
- \( KY \) = Thermal conductivity in the y-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
- \( NUXY \) = Poisson's ratio
- \( C \) = Specific heat
- \( ALPX \) = Coefficient of thermal expansion
- \( DENS \) = Density
- \( GXY \) = Shear modulus
- \( DAMP \) = Material damping coefficient
- \( ECONX \) = Electric conductivity (thermal analysis only)
Element Loadings
- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results
Stress components including von Mises stress are available in the element coordinate system at the centroid of the element for top and bottom fibers. Principal stresses may optionally be requested (see STRESS command in the ANALYSIS menu). In addition, nodal force per unit length and stress components can also be calculated and printed in the output file (see Op. 3). The directions of force and moment components per unit length are illustrated in Figure 4.9.

Figure 4.14 Quadrilateral Thick Shell

References

Thermal and Linear Structural 6-Node Triangular Thin Shell

element_name SHELL6

General Description
SHELL6 is a 6-node triangular thin shell element with membrane and bending capabilities for the analysis of three dimensional structural and thermal models. The shear deformation effect is neglected for this element. Six degrees of freedom per node (three translations and three rotations) are considered for structural analysis.

The element is assumed to be isotropic with constant thickness for structural problems and orthotropic for thermal problems.

Special Features
Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)
The nodal input pattern for the element is shown in Figure 4.15. Both clockwise and counterclockwise node numbering are allowed. For element coordinate system, the x-axis goes from the first node to the second. The element y-axis lies in the plane defined by the three nodes, perpendicular to x-axis toward the third node. The element z-axis completes a right-hand Cartesian system with x- and y-axes.

Other Element Coordinate Systems (ECS $\neq$ -1)
When a defined element coordinate system ($x_e$,$y_e$,$z_e$) is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system (xyz) similar to that described in Figure 4.6 for SHELL3.

Element Group Options

Op. 1: Unused option for this element
Op. 2: Integration type
  = 0 ; Reduced integration
  = 1 ; Full integration
Op. 3: Unused option for this element
Op. 4: Stress directions
  = 0 ; Calculate stresses in the global Cartesian coordinate system
  = 1 ; Calculate stresses in the defined element local coordinate system
Op. 5: Unused option for this element
Op. 6: Unused option for this element
Op. 7: Unused option for this element
Op. 8: Unused option for this element

Real Constants
r1  = Thickness
r2  = Temperature gradient
Material Properties

EX = Modulus of elasticity
KX = Thermal conductivity in the x-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
KY = Thermal conductivity in the y-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
NUXY = Poisson's ratio
C = Specific heat
ALPX = Coefficient of thermal expansion
DENS = Density
GXY = Shear modulus
DAMP = Material damping coefficient

Element Loadings

- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results

Stress components including von Mises stress are available in the element coordinate system at the centroid of the element for top and bottom fibers. Principal stresses may optionally be requested (see STRESS command in the ANALYSIS menu).

Figure 4.15 6-Node Triangular Thin Shell

References


Linear Structural 8/9-Node Isoparametric Shell

element_name SHELL9

General Description

SHELL9 is an 8- or 9-node quadrilateral shell element with membrane, bending and shear capabilities for the analysis of three-dimensional structural models. Six degrees of freedom (three translations and three rotations) are considered per node.

The element is assumed to be isotropic with constant thickness. For orthotropic materials, SHELL9L may be considered.

The nodal input pattern can be specified in both clockwise and counter-clockwise directions. The ninth (9th) node (if any) must be placed at the center of the element.

Special Features

Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)

The element is a curved lamina in three dimensional space. At any point of the element curved surface the z-axis of the local coordinate system is normal to the element mid-plane and the x- and y-axes lie in the tangent plane. The direction of the element axes (x and y) are related to the natural coordinates (ξ, η) as shown in Figure 4.16. Given in the figure also, an example of the element coordinate system at node 3. A special case of the element coordinate system for a cylindrical surface is shown in Figure 4.17.

Other Element Coordinate Systems (ECS >= -1):

When a defined element coordinate system (xe,ye,ze), as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system (xyz) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle (α) between the xe axis and the normal to the surface (z) is greater than 45° (α > 45°), the element x-axis is considered as the projection of xe on the element plane.

If the angle (α) between the xe axis and the normal to the surface (z) is less than or equal to 45° (α <= 45°), the element x-axis is considered as the projection of ye on the element plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.

Element Group Options

Op. 1: Number of nodes per element
  = 0 ; Use 9-node element (default)
  = 1 ; Use 8-node element
Op. 2: Integration method (see References)
\[ = 0 ; \text{ Using GAMA-PHI method} \]
\[ = 1 ; \text{ Using reduced integration} \]

Op. 3: Unused option for this element

Op. 4: Stress directions
\[ = 0 ; \text{ Calculate nodal stresses in the global Cartesian coordinate system} \]
\[ = 1 ; \text{ Calculate nodal stresses in the default element coordinate system} \]

Op. 5 to Op. 8: Unused options for this element

Real Constants
\[ r_1 = \text{ Thickness} \]

Material Properties
\[ EX = \text{ Modulus of elasticity} \]
\[ NUXY = \text{ Poisson's ratio} \]
\[ DENS = \text{ Density} \]
\[ GXY = \text{ Shear modulus} \]
\[ DAMP = \text{ Material damping coefficient} \]

Element Loadings
- Thermal
- Gravitational
- Pressure (applied to element faces)

Output Results
Forces and stress components are available in the nodal coordinate system for each node of the element. Stresses for top and bottom fibers are also calculated.

**Figure 4.16 9-Node Isoparametric Shell Element**
Figure 4.17 SHELL9 as part of a Cylindrical Surface

References

Thermal and Linear Structural 2D 3- to 6-Node Triangular Plane Stress, Plane Strain, and Body of Revolution
element_name TRIANG

General Description
TRIANG is a 3- to 6-node triangular, two-dimensional element for plane stress, plane strain, or axisymmetric structural and thermal models. All elements have to be defined in the X-Y plane. Axisymmetric structures have to be modeled in the positive X half plane in which X represents the radial direction and Y refers to the axis of symmetry. Only two translational degrees of freedom per node are considered for structural analysis. One degree of freedom per node, representing temperature, is used for the thermal module.

The nodal input pattern for this element is shown in Figure 4.18. Both clockwise and counter-clockwise node numbering are allowed. For transitional elements, missing nodes are issued zeros (0) at their location during the element connectivity definition (Define Element command).

Special Features
Buckling, Inplane loading, Adaptive P- and HP-methods for the 6-node structural elements (polynomial degrees up to 10), Adaptive H-Method.

Default Element Coordinate System (ECS = -1)
The element x-axis goes from the first node to the second. The element y-axis is normal to the x-axis toward the third node.

Element Group Options

Op. 1: Unused option for this element

Op. 2: (See note on Numerical Integration below)
  =0 ;  1-Point Integration
  =1 ;  3-point integration

Op. 3:
  = 0 ;  Plane Stress (default)
  = 1 ;  Axisymmetric
  = 2 ;  Plane strain

Op. 4:
  = 0 ;  Stress printout in the global Cartesian coordinate system (default)
  = 1 ;  Stress printout in the defined element local coordinate system. Stress at midside nodes are obtained by averaging the stresses of corresponding vertex nodes

Op. 5: Use default value (linear elastic material)

Op. 6: Use default value (small displacement formulation)

Op. 7: Use default value (material creep is not considered)

Op. 8: Unrelated option to this type of analysis (use default value)
Real Constants

- $r_1$ = Thickness (only for plane stress analysis)
- $r_2$ = Material angle ($\beta$)

The material angle is measured with respect to the element coordinate system, as shown in Figure 4.18.

Material Properties

(See Figure 4.18 for material directions)

- $E_X$ = Modulus of elasticity in the 1st material direction
- $E_Y$ = Modulus of elasticity in the 2nd material direction
- $E_Z$ = Modulus of elasticity in the global Z-direction
- $K_X$ = Thermal conductivity in the x-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
- $K_Y$ = Thermal conductivity in the y-direction defined by the ECS which must be 0, -1, or a user created Cartesian system
- $\nu_{XY}$ = Poisson's ratio relating the 1st and 2nd material directions (strain in the 2nd direction due to unit strain along the 1st direction)
- $\nu_{YZ}$ = Poisson's ratio relating the 2nd material direction and global Z-direction (strain in the Z-direction due to unit strain along the 2nd direction)
- $\nu_{XZ}$ = Poisson's ratio relating the 1st material direction and global Z-direction (strain in the Z-direction due to unit strain along the 1st direction)
- $C$ = Specific heat
- $\alpha_X$ = Coefficient of thermal expansion in the 1st material direction
- $\alpha_Y$ = Coefficient of thermal expansion in the 2nd material direction
- $\alpha_Z$ = Coefficient of thermal expansion in the global Z-direction
- $\rho$ = Density
- $\gamma$ = Material damping coefficient
- $E_{CONX}$ = Electric conductivity (thermal analysis only)

Note:
The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:

1. Moduli of elasticities in two directions are defined and are unequal.
2. Poisson's ratio in two planes are defined and are unequal.
3. Thermal coefficients in two directions are defined and are unequal.
4. Thermal conductivity in two directions are defined and are unequal.

The following condition has to be satisfied for proper representation of orthotropic properties in the $i^{th}$ and $j^{th}$ material direction:

$$\nu_{ij}/E_i = \nu_{ji}/E_j$$

Where $\nu_{ij}$, $E_i$, and $E_j$ are provided as input and $\nu_{ji}$ calculated internally by the program.

Element Loadings

- Thermal
- Gravitational
- Pressure (applied normal to element faces)
Output Results
Stress components in the global or local coordinate systems and von Mises stress are available at all nodes and the center of the element.

Principal stresses may also be optionally requested at the element center.

Note on Numerical Integration
Option 2 defines the integration scheme where one or three points may be used as follows:
Figure 4.19

References
Thermal and Linear Structural 2D 4- to 8-Node Plane Stress, Plane Strain and Body of Revolution

**element_name PLANE2D**

**General Description**

PLANE2D is a 4- to 8-node two dimensional element for plane stress, plane strain, or axisymmetric structural with symmetric and non-symmetric (asymmetric) loading and thermal problems. All elements have to be defined in the X-Y plane. Axisymmetric structures have to be modeled in the positive X half plane, in which X represents the radial direction and Y refers to the axis of symmetry for axisymmetric structures with symmetric or asymmetric loading conditions. Only two translational degrees of freedom per node are considered for structural analysis. In the case of axisymmetric structures with asymmetric loading conditions or frequency and buckling analysis of axisymmetric structures for different circumferential harmonic numbers, an additional translational degree of freedom along the Z-axis to account for asymmetric conditions and modes is required. One degree of freedom, representing temperature, is used for the thermal module.

The nodal input pattern is shown in Figure 4.20 for an 8-node element illustrating its local node numbering. The element however can be used with 4- to 8-nodes by assigning zeros (0) at the locations of missing nodes during element connectivity definition (Define Element command). Triangular shaped elements can also be considered. In this case, the third and fourth nodes (in case of 4-node elements) and the third, fourth and seventh nodes (in case of 5- to 8-node elements) will be assigned the same global node number, as shown in Figure 4.20. Both clockwise and counter-clockwise node numbering are allowed.

**Special Features**

Buckling, Inplane Loading, Fluid-solid interaction, Adaptive P-Method for the 8-node structural elements with axisymmetric loading (polynomial order up to 10).

**Default Element Coordinate System (ECS= -1)**

The element x-axis goes from the first node to the second, and the element y-axis is normal to the x-axis toward the fourth node.

**Element Group Options**

Op. 1:

- 0 ;  Regular structural or thermal element (default)
- 1 ;  4-node incompressible fluid element

**For structural or thermal elements (Op. 1 = 0), the other options are:**

Op. 2:

- 0 ;  Reduced integration
- 1 ;  QM6 incompatible element; full integration for 8-node elements (default)
- 2 ;  Full integration
- 3 ;  Unrelated option for this type of analysis

Op. 3:

- 0 ;  Plane Stress (default)
- 1 ;  Axisymmetric (a one radian sector is considered, and hence, forces should be applied based on one radian)
= 2 ;  Plane Strain
= 3 ;  Axisymmetric structure with non-symmetric loading

Op. 4:
= 0 ;  Stresses calculated in global Cartesian coordinate system
= 1 ;  Stresses calculated in the defined local element coordinate system

Op. 5:  Use default value (linear elastic material)
Op. 6:  Use default value (small displacement formulation)
Op. 7:  Use default value (material creep is not considered)
Op. 8:  Unrelated option to this type of analysis (use default value)

*For fluid elements (Op. 1 = 1), the other options are:*

Op. 2:  Unused option for this element
Op. 3:
= 1 ;  Axisymmetric with symmetric loading
= 2 ;  Plane Strain (default)

Op. 4 to Op. 8:  Unused options for this element

**Real Constants**

r1 = Thickness (only for plane stress analysis)
r2 = Material angle (β)
The material angle is measured with respect to the element coordinate system,
as shown in Figure 4.20.

**Material Properties**

1. For structural or thermal elements (Op. 1 = 0)
   *(See Figure 4.20 for material directions)*

   EX =  Modulus of elasticity in the 1st material direction
   EY =  Modulus of elasticity in the 2nd material direction
   EZ =  Modulus of elasticity in the global Z-direction
   KX =  Thermal conductivity in the x-direction defined by the ECS which must
         be 0, -1, or a user created Cartesian system
   KY =  Thermal conductivity in the y-direction defined by the ECS which must
         be 0, -1, or a user created Cartesian system
   NUXY = Poisson's ratio relating the 1st and 2nd material directions (strain in the
          2nd direction due to unit strain along the 1st direction)
   NUYZ = Poisson's ratio relating the 2nd material direction and global Z-direction
          (strain in the Z-direction due to unit strain along the 2nd direction)
   NUXZ = Poisson's ratio relating the 1st material direction and global Z-direction
          (strain in the Z-direction due to unit strain along the 1st direction)
   C =  Specific heat
   ALPX = Coefficient of thermal expansion in the 1st material direction
   ALPY = Coefficient of thermal expansion in the 2nd material direction
   ALPZ = Coefficient of thermal expansion in the global Z-direction
   GXY =  Shear modulus relating the 1st and 2nd material directions
   DENS =  Density
   DAMP =  Material damping coefficient
   ECONX = Electrical conductivity (thermal analysis only)
Note:
The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:
1. Moduli of elasticities in two directions are defined and are unequal.
2. Poisson's ratio in two planes are defined and are unequal.
3. Thermal coefficients in two directions are defined and are unequal.
4. Thermal conductivity in two directions are defined and are unequal.
The following conditions must be met for proper representation of orthotropic properties for the \(i\)th and \(j\)th material directions:
\[
\nu_{ij}/E_i = \nu_{ji}/E_j
\]
Where \(\nu_{ij}\), \(E_i\), and \(E_j\) are provided as input and \(\nu_{ji}\) calculated internally by the program.

2. For fluid elements (Op. 1 = 1)
   \[
   EX = \text{Fluid elastic (bulk) modulus}
   \]
   \[
   GXY = 10^{-19}EX; \text{ an arbitrarily small number to give element some shear stability}
   \]

Element Loadings
- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results
Stress components including the von Mises stress are available at all nodes and the center of the element in either global or element coordinate directions.

Principal stresses may also be optionally requested at the element center (see STRESS command in the ANALYSIS menu).

For the fluid option, pressure is printed at the center of each element.

Note on Numerical Integration
Option 2 defines the numerical integration scheme which may be specified as one of the following options:
1. Reduced Integration
   - For 4-node elements:
     - 2 x 2 Gauss integration for bending terms
     - 1 x 1 Gauss integration for shear terms
     - Overcomes parasitic shear effects; handles nearly incompressible materials; not available for orthotropic models.
   - For 8-node elements:
     - 2 x 2 Gauss integration for bending terms
2 x 2 Gauss integration for shear terms

2. QM6 (Available for 4-node elements only)
   2 x 2 Gauss integration for all terms including the effect of bubble functions which introduce additional internal degrees of freedom.
   Overcomes parasitic shear effects, handles nearly incompressible materials, in general more stable with better accuracy, but more costly in terms of solution time.

3. Full Integration
   - For 4-node elements:
     2 x 2 Gauss integration for all terms.
     Fastest and simplest solution option, does not overcome parasitic shear effects.
   - For 8-node elements:
     3 x 3 Gauss integration for all terms.
     This option should not be used for 8-node elements if the value of the Poisson's ratio is greater than 0.48. For this special case reduced integration should be used instead.
References


Thermal and Linear Structural 2D Spar/Truss

**element_name TRUSS2D**

**General Description**

TRUSS2D is a 2-node uniaxial element for two dimensional structural and thermal models. All elements have to be defined in the X-Y plane as shown in Figure 4.21. Only two translational degrees of freedom per node are considered for structural analysis. Temperature is the only degree of freedom for the thermal module.

**Special Features**

Buckling, Inplane loading.

**Default Element Coordinate System (ECS = -1)**

The nodal input pattern shown in Figure 4.21 specifies the direction of the element axis. The x-axis goes from the first node to the second. The element y-axis is perpendicular to the x-axis and lies in the X-Y plane.

**Element Group Options**

- **Op. 1 to Op. 4:** Unused options for this element
- **Op. 5:** Use default value (linear elastic material type)
- **Op. 6:** Use default value (small displacement formulation)
- **Op. 7:** Use default value (material creep is not considered)
- **Op. 8:** Unused option for this element

**Real Constants**

- r1 = Cross-sectional area
- r2 = Perimeter (thermal analysis only)

**Material Properties**

- EX = Modulus of elasticity
- KX = Thermal conductivity
- ALPX = Coefficient of thermal expansion
- C = Specific heat
- DENS = Density
- DAMP = Material Damping coefficient
- ECONX = Electrical conductivity (thermal analysis only)

**Element Loadings**

- Thermal
- Gravitational

**Output Results**

Forces and stresses are available in the element coordinate system.
Figure 4.21  2D Truss
Thermal and Linear Structural 3D Spar/Truss

**element_name** TRUSS3D

**General Description**
TRUSS3D is a 2-node uniaxial element for three dimensional structural and thermal models. Only three translational degrees of freedom are considered per node for structural analysis. Temperature is the only degree of freedom for the thermal module.

**Special Features**
Buckling, Inplane loading.

**Default Element Coordinate System (ECS = -1)**
The nodal input pattern shown in Figure 4.22 specifies the direction of the element axis. The x-axis goes from the first node to the second. The element y- and z-axes lie in a plane perpendicular to the x-axis.

**Element Group Options**
- **Op. 1 to Op. 4:** Unused options for this element
- **Op. 5:** Use default value (linear elastic material type)
- **Op. 6:** Use default value (small displacement formulation)
- **Op. 7:** Use default value (material creep is not considered)
- **Op. 8:** Unused option for this element

**Real Constants**
- \( r_1 \) = Cross-sectional area
- \( r_2 \) = Perimeter (thermal analysis only)

**Material Properties:**
- \( E_X \) = Modulus of elasticity
- \( K_X \) = Thermal conductivity
- \( A_LP_X \) = Coefficient of thermal expansion
- \( C \) = Specific heat
- \( DENS \) = Density
- \( D Amp \) = Material damping coefficient
- \( ECONX \) = Electrical conductivity (thermal analysis only)

**Element Loadings**
- Thermal
- Gravitational

**Output Results**
Forces and stresses are available in the element coordinate system.
XYZ: Global Cartesian Coordinate System
xyz: Element Coordinate System

Figure 4.22 3D Truss
Thermal and Linear Structural 2D Elastic Beam
element_name BEAM2D

General Description
BEAM2D is a 2-node uniaxial element for two-dimensional structural and thermal models. The element has three degrees of freedom (two translations and one rotation) per node for structural analysis. Temperature is the only degree of freedom for the thermal module. All elements have to be defined in the X-Y plane as shown in Figure 4.23.

Special Features
Buckling, Inplane loading, ASME code check (see STRESS command in the ANALYSIS menu).

Default Element Coordinate System (ECS = -1)
The nodal input pattern shown in Figure 4.23 specifies the direction of the element axis. The x-axis goes from the first node to the second. The element z-axis is parallel to and in the positive sense of the global Cartesian Z-axis. The element y-axis is perpendicular to the x-axis and completes, with x- and z-axes, a right-hand Cartesian coordinate system.

Element Group Options
- Op. 1 to Op. 3: Unused options for this element
- Op. 4: Unrelated option to this type of analysis (use default value)
- Op. 5: Use default value (linear elastic material)
- Op. 6: Use default value (small displacement formulation)
- Op. 7: Unused option for this element
- Op. 8: Unused option for this element

Real Constants
There are two different ways to define the section properties:
A. Using Define RC Set option in Real Constants Menu

\[
\begin{align*}
r_1 &= \text{Cross-sectional area} \\
r_2 &= \text{Moment of inertia} \\
r_3 &= \text{Depth (diameter for circular cross-sections)} \\
r_4 &= \text{End-release code (node 1)}* \\
r_5 &= \text{End-release code (node 2)}* \\
r_6 &= \text{Shear factor in the element y-axis} \\
r_7 &= \text{Temperature difference in the element y-axis} \\
r_8 &= \text{Perimeter (thermal analysis only)}
\end{align*}
\]

* The end-release code for each end is specified by a six-digit number with combinations of zeros and ones. If a zero (0) is placed in a particular location, the corresponding force is not known and will be calculated by the program, but if a one (1) is placed in that location, the force or moment corresponding to that direction is known to be zero due to a hinge or roller, and the program will remove the force. The six-digit code corresponds in order to the six degrees of freedom at each end of the beam element. For example, end release code 000001 represents a
condition in which the forces in the x and y directions are to be calculated and the moment about the Z-axis is zero, i.e., a hinge condition.

B. Using Define Beam Sections Menu

There are five cross sections available for the BEAM section library as shown in Figure 4.24.

1. Solid Rectangular Section:
   - Constant_1 = Height of the beam (H)
   - Constant_2 = Width of the beam (B)

2. Solid Circular Section:
   - Constant_1 = Radius of the beam (R)

3. Circular Hollow Section (Pipe):
   - Constant_1 = Outside diameter (D)
   - Constant_2 = Thickness (T)

4. Hollow Rectangular Section (Box):
   - Constant_1 = Height of the beam (H)
   - Constant_2 = Width of the beam (B)
   - Constant_3 = Thickness associated with the height (TB)
   - Constant_4 = Thickness associated with the width (TH)

5. Symmetric I-Section:
   - Constant_1 = Height of the beam (H)
   - Constant_2 = Flange width (B)
   - Constant_3 = Flange thickness (TH)
   - Constant_4 = Web thickness (TB)

After entering values for the constants described above, values corresponding to real constants r4, r5, r6, r7, and r8 also have to be input for each cross section defined.

Material Properties

EX = Modulus of elasticity
KX = Thermal conductivity
ALPX = Coefficient of thermal expansion
C = Specific heat
NUXY = Poisson's ratio
DENS = Density
DAMP = Material Damping Coefficient
ECONX = Electrical conductivity (thermal analysis only)

Element Loadings

- Uniform pressure (in terms of force per unit length)
- Thermal
- Gravitational
- Beam loading

Output Results

Forces, moments, and stresses are available in the element coordinate system.
Figure 4.23  2D Elastic Beam

Figure 4.24  Identification Parameters of the Built-In Section Library for BEAM2D
References

Thermal and Linear Structural 3D Elastic Beam

**element_name BEAM3D**

**General Description**

BEAM3D is a 2-node uniaxial element for three-dimensional structural and thermal models. For structural analysis, six (6) degrees of freedom (three translations and three rotations) are considered per node. One (1) degree of freedom per node, representing the temperature is used for the thermal module. A third node or an orientation angle is required only for the element orientation as shown in Figure 4.25.

This element permits using unsymmetric cross-section (Figure 4-26) when the shear center is not coincident with the center of gravity; e.g., channel and L-shape cross-sections can be considered. For the element two nodes (1 and 2), an offset is allowed from the centroidal axis.

**Special Features**

Buckling, Inplane loading, Node offset, Unsymmetric cross-sections, ASME code check (see STRESS command in the ANALYSIS menu).

**Default Element Coordinate System (ECS = -1)**

The element x-axis is defined by a vector starting from the first node towards the second. The orientation of the cross-section is defined by a third node or an angle. The third node, if specified, defines the element y-axis such that it is orthogonal to the element x-axis, starts from the first node towards the third, and lies in the plane defined by the three nodes.

If a third node is not specified, the orientation of the cross-section is defined through real constants r13 or r21 as the angle between $y_o$ and the element y-axis as shown in Figure 4.25(c). The axis $y_o$ is parallel to the global X-Y plane and normal to the element x-axis. A positive angle is measured from the $y_o$ axis using the right-hand rule such that the thumb points to the positive direction of the element x-axis. Real constant r13 and r21 are ignored if a third node exists.

The element z-axis completes a right-handed Cartesian coordinate system defined by the element x- and y-axes.

**Element Group Options**

- **Op. 1:** Section type
  - 0 ; Symmetric (default)
  - 1 ; Unsymmetric
  - 2 ; Symmetric tapered

- **Op. 2 and Op. 3:** Unused options for this element

- **Op. 4:** Unrelated option for this type of analysis (use default value)

- **Op. 5:** Use default value (linear elastic material)

- **Op. 6:** Use default value (small displacement formulation)

- **Op. 7:** Unused option for this element

- **Op. 8:** Unused option for this element
Real Constants
There are two different ways to define the section properties:

A. Using Define RC Set option in Real Constants Menu

_Symmetric and Unsymmetric Sections (Op. 1 = 0 or 1)_

- \( r_1 \) = Cross-sectional area
- \( r_2 \) = Moment of inertia about the element y-axis
- \( r_3 \) = Moment of inertia about the element z-axis
- \( r_4 \) = Depth of beam (y-axis) or the diameter for a circular cross-section
- \( r_5 \) = Width of beam (z-axis) or the diameter for a circular cross-section
- \( r_6 \) = End-release code (node 1)*
- \( r_7 \) = End-release code (node 2)*
- \( r_8 \) = Torsion constant J (Polar moment of inertia for circular sections)*****
- \( r_9 \) = Shear factor in the element y-axis (Asy/Area)**
- \( r_{10} \) = Shear factor in the element z-axis (Asz/Area)**
- \( r_{11} \) = Temperature difference in the element y-axis
- \( r_{12} \) = Temperature difference in the element z-axis
- \( r_{13} \) = Orientation angle (degrees)
- \( r_{14} \) = CTOR ; Constant for maximum shear stress calculation*****

_Unsymmetric Section only (Op. 1 = 1)_

- \( r_{15} \) = DX1 ; x-distance of the section centroid relative to the nodal point location at node 1 ***
- \( r_{16} \) = DX2 ; x-distance of the section centroid relative to the nodal point location at node 2 ***
- \( r_{17} \) = DY1 ; y-distance of the section centroid relative to the nodal point location at node 1 ***
- \( r_{18} \) = DY2 ; y-distance of the section centroid relative to the nodal point location at node 2 ***
- \( r_{19} \) = DZ1 ; z-distance of the section centroid relative to the nodal point location at node 1 ***
- \( r_{20} \) = DZ2 ; z-distance of the section centroid relative to the nodal point location at node 2 ***
- \( r_{21} \) = DYSC1 ; y-distance of the shear center relative to the section centroid at node 1 ****
- \( r_{22} \) = DZSC1 ; z-distance of the shear center relative to the section centroid at node 1 ****
- \( r_{23} \) = DYSC2 ; y-distance of the shear center relative to the section centroid at node 2 ****
- \( r_{24} \) = DZSC2 ; z-distance of the shear center relative to the section centroid at node 2 ****
- \( r_{25} \) = Ty ; y-distance of the point where stresses are to be calculated ****
- \( r_{26} \) = Tz ; z-distance of the point where stresses are to be calculated ****
- \( r_{27} \) = Iyz ; Centroidal product of inertia of the element cross-section

_Symmetric Tapered (Op. 1 = 2)_

(Sec. 1) and (Sec. 2) will refer to sectional properties of the beam at nodes 1 and 2.

- \( r_1 \) = Cross-sectional area for (Sec. 1)
- \( r_2 \) = Cross-sectional area for (Sec. 2)
- \( r_3 \) = Moment of inertia about the element y-axis at (Sec. 1)
- \( r_4 \) = Moment of inertia about the element y-axis at (Sec. 2)
- \( r_5 \) = Moment of inertia about the element z-axis at (Sec. 1)
Chapter 4   Element Library

r6  =  Moment of inertia about the element z-axis at (Sec. 2)
r7  =  Depth of beam (y-axis) at (sec. 1) or the diameter in case of a circular cross-section
r8  =  Depth of beam (y-axis) at (sec. 2) or the diameter in case of a circular cross-section
r9  =  Depth of beam (z-axis) at (sec. 1) or the diameter in case of a circular cross-section
r10 =  Depth of beam (z-axis) at (sec. 2) or the diameter in case of a circular cross-section
r11 =  End-release code at (node 1)*
r12 =  End-release code at (node 2)*
r13 =  Torsion constant J of (sec. 1)*****
r14 =  Torsion constant J of (sec. 2)*****
r15 =  Shear factor in the element y-axis**
r16 =  Shear factor in the element z-axis**
r17 =  Temperature difference in the element y-axis
r18 =  Temperature difference in the element z-axis
r19 =  Perimeter (thermal analysis only)
r20 =  CTOR; Constant for maximum shear stress calculation*****
r21 =  Orientation angle (degrees)

* The end-release code for each end is specified by a six-digit number with combinations of zeros and ones. If a zero (0) is placed in a particular location, the corresponding force is not known and will be calculated by the program, but if a one (1) is placed in that location, the force or moment corresponding to that direction is known to be zero due to a hinge or roller, and the program will remove the force. The six-digit code corresponds in order to the six degrees of freedom at each end of the beam element. For example, end release code 101100 for a 3D beam element represents a condition in which the forces in the x- and z-directions and the moment about the x-axis are zero, and the force in the y-direction and moments about y- and z-axes are to be calculated.

** Asy = beam cross-sectional area effective in shear in y-direction.
Asz = beam cross-sectional area effective in shear in z-direction.

*** Offset distances (Dy, Dz) shown in Figure 4.26 are measured positive from the nodal point in the positive element coordinate directions.

**** Stress point (Ty, Tz) and shear center distances (DYSC, DZSC) shown in Figure 4.26 are measured positive from the center of gravity in the positive element coordinate directions.

***** J and CTOR given below for some beam cross-sections shown:

\[ \tau_{\text{max}} = \frac{(T)(\text{CTOR})}{J} \]

where T is the Torsional moment.
### Cross-Section

<table>
<thead>
<tr>
<th>Cross-Section</th>
<th>CTOR and J</th>
<th>Shear Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td><img src="image1" alt="Round Cross-Section" /></td>
<td>( \text{CTOR} = r ) ( J = I_p = \frac{\pi r^4}{2} )</td>
<td>( \frac{9}{10} = 0.90 )</td>
</tr>
<tr>
<td><img src="image2" alt="Rectangular Cross-Section" /></td>
<td>( \text{CTOR} = r ) ( J = I_p = I_{yy} + I_{zz} )</td>
<td>( \frac{1}{2} = 0.50 )</td>
</tr>
</tbody>
</table>
| ![Trapezoidal Cross-Section](image3) | \( \text{CTOR} = \frac{(3a + 1.8b)}{8a^2 b^2} \cdot J \) | \( \frac{5}{6} = 0.8333 \)

Where:

\[
J = ab^3 \left[ \frac{16}{3} - 3.36 \left( \frac{b}{a} \right) \left( 1 - \frac{b^4}{12a^4} \right) \right]
\]

### B. Using Define Beam Section Menu

This command replaces some of the real constants defined before. The values corresponding to the following real constants also have to be input by this command:

- \( r_6, r_7, r_9, r_{10}, r_{11}, \) and \( r_{12} \) (for symmetric sections)
- \( r_6 \) to \( r_{12}, r_{15} \) to \( r_{20}, r_{25} \) and \( r_{26} \) (for unsymmetric sections)
- \( r_{11}, r_{12}, r_{15}, r_{16}, r_{17}, r_{18}, r_{19}, \) and \( r_{20} \) (for symmetric tapered sections)

There are 10 cross-sections available for the BEAM3D section library as shown in Figure 4.28.

1. **Solid Rectangular Section:**
   - **Symmetric/Unsymmetric Beams:**
     - Constant_1 = Height of the beam (H)
     - Constant_2 = Width of the beam (B)
   - **Symmetric Tapered Beam:**
     - Constant_1 = Height of the beam at node 1 (H1)
     - Constant_2 = Width of the beam at node 1 (B1)
     - Constant_3 = Height of the beam at node 2 (H2)
     - Constant_4 = Width of the beam at node 2 (B2)

2. **Solid Circular Section:**
   - **Symmetric/Unsymmetric Beams:**
     - Constant_1 = Radius (R)
   - **Symmetric Tapered Beam:**
     - Constant_1 = Radius at node 1 (R1)
     - Constant_2 = Radius at node 2 (R2)
3. Circular Hollow Section (Pipe):
   - Symmetric/Unsymmetric Beams:
     Constant_1 = Outside Diameter (D)
     Constant_2 = Thickness (T)
   - Symmetric Tapered Beam:
     Constant_1 = Outside Diameter at node 1 (D1)
     Constant_2 = Thickness at node 1 (T1)
     Constant_3 = Outside Diameter at node 2 (D2)
     Constant_4 = Thickness at node 2 (T2)

4. Hollow Rectangular Section (Box):
   - Symmetric/Unsymmetric Beams:
     Constant_1 = Height of the beam (H)
     Constant_2 = Width of the beam (B)
     Constant_3 = Thickness associated with the height (TB)
     Constant_4 = Thickness associated with the width (TH)
   - Symmetric Tapered Beam:
     Constant_1 = Height of the beam at node 1 (H1)
     Constant_2 = Width of the beam at node 1 (B1)
     Constant_3 = Thickness associated with the height at node 1 (TB1)
     Constant_4 = Thickness associated with the width at node 1 (TH1)
     Constant_5 = Height of the beam at node 2 (H2)
     Constant_6 = Width of the beam at node 2 (B2)
     Constant_7 = Thickness associated with the height at node 2 (TB2)
     Constant_8 = Thickness associated with the width at node 2 (TH2)

5. I-Section:
   - Symmetric/Unsymmetric Beams:
     Constant_1 = Height of the beam (H)
     Constant_2 = Flange width (B)
     Constant_3 = Flange thickness (TH)
     Constant_4 = Web thickness (TB)
   - Symmetric Tapered Beam:
     Constant_1 = Height of the beam at node 1 (H1)
     Constant_2 = Flange width at node 1 (B1)
     Constant_3 = Flange thickness at node 1 (TH1)
     Constant_4 = Web thickness at node 1 (TB1)
     Constant_5 = Height of the beam at node 2 (H2)
     Constant_6 = Flange width at node 2 (B2)
     Constant_7 = Flange thickness at node 2 (TH2)
     Constant_8 = Web thickness at node 2 (TB2)

6. Trapezoidal Solid Section:
   Constant_1 = Height of the beam (H)
   Constant_2 = Bottom width of the beam (B1)
   Constant_3 = Top width of the beam (B2)
   (Note that H>B1>B2)

7. Thin-Walled Channel Section:
   Constant_1 = Height of the beam (H)
   Constant_2 = Flange width (B)
   Constant_3 = Flange thickness (TH)
   Constant_4 = Web thickness (TB)
8. Thin-Walled Z-Section:
   Constant_1 = Height of the beam (H)
   Constant_2 = Flange width (B)
   Constant_3 = Flange thickness (TH)
   Constant_4 = Web thickness (TB)

9. Thin-Walled T-Section:
   Constant_1 = Height of the beam (H)
   Constant_2 = Flange width (B)
   Constant_3 = Flange thickness (TH)
   Constant_4 = Web thickness (TB)

10. Thin-Walled L-Section:
    Constant_1 = Height of the beam (H)
    Constant_2 = Width (B)
    Constant_3 = Thickness associated with the height (TB)
    Constant_4 = Thickness associated with the width (TH)

Material Properties

    EX = Modulus of elasticity
    KX = Thermal conductivity
    ALPX = Coefficient of thermal expansion
    C = Specific heat
    NUXY = Poisson's ratio
    DENS = Density
    DAMP = Material damping coefficient
    ECONX = Electric conductivity (thermal analysis only)

Element Loadings

- Uniform lateral pressure (in terms of force per unit length): offset effect is not considered.
- Thermal
- Gravitational
- Beam loading

Output Results

Forces, moments, and stresses are available in the element coordinate system shown in Figure 4.27.
(a) 2-node element with axial third node defining alignment

(b) 2-node element with orientation angle defining alignment

(c) If node K is omitted and $\theta = 0^\circ$, the element y-axis is parallel to the global X-Y plane

Figure 4.25 3D Symmetric Elastic Beam
Chapter 4   Element Library

Figure 4.26   3D Unsymmetric Elastic Beam

Figure 4.27   3D Elastic Beam

| CG: | Center of gravity (Centroid) of cross section |
| SC: | Shear center of cross section |
| I:  | Nodal point |
| SP: | Point at which the stress is required |

y and z axes define the element coordinate system. The positive values are in the direction of the arrows.
Figure 4.28 Identification Parameters of the Built-In BEAM3D Section Library (viewed by looking into the negative x-direction)
References
Thermal Convection Link  
**element_name CLINK**

**General Description**

CLINK is a 2-node element to model the heat flow due to convection between two nodes. One degree of freedom per node is used in two- or three-dimensional thermal models.

The nodal input pattern for this element is shown in Figure 4.29. The two nodes may or may not be coincident. Temperature boundary conditions must be specified at the node which is not directly connected to the model. This temperature boundary condition represents the convection source temperature.

**Special Features**  (None)

**Element Group Options**  (None)

**Real Constants**

\[ r_1 = \text{Area of the convection surface} \]

**Material Properties**

\[ HC = \text{Film coefficient} \]

**Element Loadings**

- Thermal

**Output Results**

Heat flow due to convection is available for each element.

![Figure 4.29 Convection Link](image-url)
Thermal Radiation Link

element_name RLINK

General Description

RLINK is a 2-node element to model the heat flow between two nodes due to radiation. One degree of freedom for each node is used in two- or three-dimensional thermal models.

The nodal input pattern for this element is shown in Figure 4.30. The two nodes may or may not be coincident. Temperature boundary condition must be specified at the node which is not directly connected to the model. This temperature boundary condition represents the radiation source temperature.

Special Features  (None)

Element Group Options  (None)

Real Constants

    r1 = Area of the radiating surface
    r2 = View factor
    r3 = Emissivity
    r4 = Stefan-Boltzman constant

Material Properties  (None)

Element Loadings

- Thermal

Output Results

Heat flow due to radiation is available for each element.

Figure 4.30  Radiation Link
Linear Structural Spring Element

**element_name SPRING**

**General Description**

SPRING is a 2-node uniaxial element for structural models. Two degrees of freedom (one translation and one rotation) are considered for each node in the element local coordinate system. The element has the capability to perform as a longitudinal and/or torsional spring in one-, two-, or three-dimensional applications.

**Special Features**  (None)

**Default Element Coordinate System (ECS = -1)**

The nodal input pattern shown in Figure 4.31 specifies the direction of the element axes which also correspond to the directions of both axial and torsional degrees of freedom. For the element coordinate system, the x-axis goes from the first node to the second. The element y- and z-axes lie in a plane perpendicular to the x-axis.

**Element Group Options**

**Op. 1:**
- 0 ; Axial spring
- 1 ; Torsional spring
- 2 ; Both axial and torsional

**Op. 2:**
- 1 ; 1-node element (global Cartesian coordinate system is taken as the local element coordinate system)
- 2 ; 2-node element (default)

**Op. 3:** Degrees of freedom for the 1-node element (prompted only if Op. 2 = 1)
- 1 ; UX, translational degree of freedom along global Cartesian X-direction
- 2 ; UY, translational degree of freedom along global Cartesian Y-direction
- 3 ; UZ, translational degree of freedom along global Cartesian Z-direction
- 4 ; ROTX, rotational degree of freedom about global Cartesian X-direction
- 5 ; ROTY, rotational degree of freedom about global Cartesian Y-direction
- 6 ; ROTZ, rotational degree of freedom about global Cartesian Z-direction
- 7 ; UX and ROTX
- 8 ; UY and ROTY
- 9 ; UZ and ROTZ

**Op. 4:** Unused option for this element

**Op. 5:** Use default value (linear behavior)

**Op. 6:** Use default value (small displacement formulation)

**Op. 7:** Unused option for this element

**Op. 8:** Unused option for this element

**Real Constants**

- \( r1 \) = Axial stiffness
- \( r2 \) = Rotational stiffness
**Material Properties**  (None)

**Element Loadings**  (None)

**Output Results**
Axial forces and torsional moments are available in the element coordinate system.

---

**Figure 4.31  3D Spring**
Linear Structural Gap-Friction

element_{name} GAP

General Description

GAP is a 2-node element for two- or three-dimensional interface problems in structural models. The element behaves similar to a rigid link which can resist either compression or tension in the direction normal to the interface:

A compressive gap resists compression once the relative contraction between the two nodes exceeds the defined gap distance.

A tensile gap limits the relative expansion between the two nodes to the gap distance.

Static friction effects, when present, are accounted for by the product of the gap's normal force and the coefficient of friction. Sliding friction is not supported by the linear Gap-Friction elements. Static friction can be considered only for two-dimensional problems.

Default Element Coordinate System (ECS = -1)

The direction of the gap goes from the first to the second node. The contact surface is normal to the gap direction.

Element Group Options

Op. 1 and Op. 2: Unrelated options to this type of analysis (use default values)
Op. 3: Use default value (friction can be considered only in the X-Y plane)
Op. 4: Use default value (node to node element)
Op. 5: Unused option for this element
Op. 6: Gap distance calculations for compressive gap elements
  = 0 ; User-calculated (default)
  = 1 ; Automatic calculations such that the two nodes contact each other (not applicable for tensile gaps)
Op. 7: Unused option for this element
Op. 8: Unused option for this element

Real Constants

r1 = Gdist - Allowable relative displacement between 2 nodes with no gap resistance.
  > 0.0 ; gap resists compression
  = 0.0 ; gap is originally closed and resists compression
  < 0.0 ; gap resists tension

r2 = Gfric - Coefficient of friction used for defining the static friction force.

r3 to r7 = Unrelated constants to this type of analysis.

Material Properties (None)

Element Loadings (None)
Output Results
For every load case, the gap forces are available in the global Cartesian directions.

Figure 4.32  Node-to-Node Gap Element
Thermal and Linear Structural General Mass

element_name MASS

General Description
MASS is a one node concentrated mass element to define a lumped mass at a node in structural and thermal models. Up to six (6) degrees of freedom can be considered for each element, defined in the global coordinate system.

Special Features (None)

Default Element Coordinate System (ECS = -1)
The global Cartesian coordinate system is always considered as the default element coordinate system.

Element Group Options (None)

Real Constants
Only r7 is to be specified for the Thermal module.

\[
\begin{align*}
  r1 & = \text{Mass in X-direction} \\
  r2 & = \text{Mass in Y-direction} \\
  r3 & = \text{Mass in Z-direction} \\
  r4 & = \text{Rotary inertia about X-axis} \\
  r5 & = \text{Rotary inertia about Y-axis} \\
  r6 & = \text{Rotary inertia about Z-axis} \\
  r7 & = \text{Thermal Capacity defined in units of heat energy (thermal analysis only)}
\end{align*}
\]

Material Properties (None)

Element Loadings (None)

Output Results (None)
**Linear Structural Composite Triangular Plate and Shell**

element_name SHELL3L

**General Description**

SHELL3L is a 3-node multi-layer triangular shell element with membrane and bending capabilities for the analysis of three-dimensional structural models. Up to fifty (50) layers can be used. Each layer can be associated with different isotropic or orthotropic material properties. Six degrees of freedom (three translations and three rotations) are considered per node.

The nodal input pattern for this element is shown in Figure 4.33. Both clockwise and counterclockwise node numbering are allowed.

**Special Features**

Buckling, Inplane loading.

**Default Element Coordinate System (ECS = -1)**

The element x-axis goes from the first node to the second, and the element y-axis lies in the plane defined by the three nodes, perpendicular to the x-axis toward the third node. The element z-axis completes a right-hand Cartesian system with the x and y axes.

**Other Element Coordinate Systems (ECS ≠ -1)**

When a defined element coordinate system $(x_e y_e z_e)$, as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system (xyz) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle ($\alpha$) between the $x_e$ axis and the normal to the surface (z) is greater than 45° ($\alpha > 45^\circ$), the element x-axis is considered as the projection of $x_e$ on the element plane.

If the angle ($\alpha$) between the $x_e$ axis and the normal to the surface (z) is less than or equal to 45° ($\alpha \leq 45^\circ$), the element x-axis is considered as the projection of $y_e$ on the element plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.

**Element Group Options**

**Op. 1:** unused option for this element

**Op. 2:**

$NL = \text{Number of layers in the element (1 to 50)}$

**Op. 3:**

$= 0 ; \text{ stresses calculated in the global Cartesian coordinate system (default)}$

$= 1 ; \text{ stresses calculated in the defined element local coordinate system}$
= 2 ;  Stresses calculated in the material defined coordinate system (see Real
Constants)

Op. 4:  Unused option for this element

Op. 5:  Use default value (linear elastic material)

Op. 6:  Use default value (small displacement formulation)

Op. 7:  Sandwich plate option (prompted only if NL = 3 in Op. 2)
    = 0 ;  Layered input
    = 1 ;  Sandwich input

Op. 8:  Unused option for this element

Real Constants

Figure 4.34 shows the convention for thickness definition and temperature distribution of a
multi-layer composite shell element. The material angle ($\beta$) for each layer is defined relative
to the element coordinate system as shown in Figure 4.33.

(If Op. 2 $<$ 3 or Op. 2 = 3 but Op. 7 = 0)

r1 =  Distance from reference plane to upper surface (r1 is positive when upper
surface is above the reference plane)

r2 =  Temperature gradient

r3 =  Thickness of layer 1

r4 =  Material set number associated with layer 1

r5 =  Material angle ($\beta$) for layer 1

r6 =  Thickness of layer 2

r7 =  Material set number associated with layer 2

r8 =  Material angle ($\beta$) for layer 2

r[3(NL - 1) + 3] =  Thickness of layer NL

r[3(NL - 1) + 4] =  Material set number associated with layer NL

r[3(NL - 1) + 5] =  Material angle ($\beta$) or layer NL

(If Op. 2 = 3 and Op. 7 = 1)

r1 =  Distance from reference plane to upper surface (r1 is positive when upper
surface is above the reference plane)

r2 =  Temperature gradient

r3 =  Thickness of layers 1 and 3

r4 =  Material set number of layers 1 and 3

r5 =  Thickness of layer 2

r6 =  Material set number of layer 2

Material Properties

EX =  Modulus of elasticity in the 1st material direction

EY =  Modulus of elasticity in the 2nd material direction

NUXY =  Poisson's ratio relating the 1st and 2nd material directions (strain in the
2nd direction due to unit strain along the 1st direction)

NUYZ =  Poisson's ratio relating the 2nd and 3rd material directions (strain in the
3rd direction due to unit strain along the 2nd direction)

NUXZ =  Poisson's ratio relating the 1st and 3rd material directions (strain in the
3rd direction due to unit strain along the 1st direction)

ALPX =  Coefficient of thermal expansion in the 1st material direction
ALPY = Coefficient of thermal expansion in the 2nd material direction
GXY = Shear modulus relating the 1st and 2nd material directions
GXZ = Shear modulus relating the 1st and 3rd material directions
GYZ = Shear modulus relating the 2nd and 3rd material directions
DENS = Density
SIGXT = Tensile strength in the 1st material direction
SIGXC = Compressive strength in the 1st material direction
SIGYT = Tensile strength in the 2nd material direction
SIGYC = Compressive strength in the 2nd material direction
SIGXY = Shear strength in the plane defined by 1st and 2nd material directions
DAMP = Material damping coefficient

Note:
The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:
1. Moduli of elasticities in two directions are defined and are unequal.
2. Poisson's ratio in two planes are defined and are unequal.
3. Thermal coefficients in two directions are defined and are unequal.
The following conditions must be met for proper representation of orthotropic properties for the $i^{th}$ and $j^{th}$ material directions:

$$\frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j}$$

Where $\nu_{ij}$, $E_i$, and $E_j$ are provided as input and $\nu_{ji}$ calculated internally by the program.

Element Loadings
- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results
Stress components and von Mises stress are available in the element coordinate directions at the center of the top and bottom surfaces of each layer. Principal stresses may also be optionally requested.
Global Cartesian Coordinate System

Element Coordinate System

First material direction
Second material direction
Third material direction

Face numbers for Applying Loads and Boundary Conditions (pressure is positive when applied inward)

Figure 4.33 3-Node Layered Shell Element

 Convention for Thickness Definition and Temperature Distribution.

Figure 4.34 Composite Triangular Plate and Shell
References


Thermal and Linear Structural Composite Quadrilateral Plate and Shell

element_name SHELL4L

General Description

SHELL4L is a 4-node multi-layer quadrilateral shell element with membrane and bending capabilities for the analysis of three-dimensional structural and thermal models. Up to fifty (50) layers can be used. Six degrees of freedom (three translations and three rotations) are considered per node. Only one degree of freedom per node, representing the temperature, is used for the thermal module. Each layer can be associated with different isotropic or orthotropic material properties.

The nodal input pattern for this element is shown in Figure 4.35. Both clockwise and counterclockwise node numbering are allowed. A triangular element is assumed if the third and fourth nodes have the same global node number.

Special Features

Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)

The element x-axis goes from the first node to the second, and the element y-axis lies in the plane defined by the three nodes, perpendicular to the x-axis toward the third node. The element z-axis completes a right-hand Cartesian system with the x and y axes.

Other Element Coordinate Systems (ECS => -1)

When a defined element coordinate system (xₑyₑzₑ), as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system (xyz) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle (α) between the xₑ axis and the normal to the surface (z) is greater than 45° (α > 45°), the element x-axis is considered as the projection of xₑ on the element plane.

If the angle (α) between the xₑ axis and the normal to the surface (z) is less than or equal to 45° (α <= 45°), the element x-axis is considered as the projection of yₑ on the element plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.

Element Group Options

Op. 1: Type of shell element (see SHELL4)
  = 0 ; QUAD2 element (2 triangles to form a quadrilateral)
  = 1 ; QUAD4 element (4 triangles to form a quadrilateral)
  = 2 ; QUAD element (4-node quadrilateral element)
Op. 2:
   NL = Number of layers in the element (1 to 50)

Op. 3:
   Nodal stress directions
   = 0 ; Stresses calculated in the global Cartesian coordinate system (default)
   = 1 ; Stresses calculated in the defined element local coordinate system
   = 2 ; Stresses calculated in the material direction

Op. 4:
   Unused option for this element

Op. 5:
   Use default value (linear elastic material)

Op. 6:
   Use default value (small displacement formulation)

Op. 7:
   Sandwich plate option (prompted only if NL = 3 in Op. 2)
   = 0 ; Layered input
   = 1 ; Sandwich input

Op. 8:
   Unused option for this element

Real Constants

Figure 4.34 shows the convention for thickness definition and temperature distribution of a multi-layer composite shell element. The material angle ($\beta$) for each layer is defined relative to the element coordinate system as shown in Figure 4.35.

(If Op. 2 $\neq$ 3 or Op. 2 = 3 but Op. 7 = 0)

   r1 = Distance from reference plane to upper surface (r1 is positive when upper surface is above the reference plane)
   r2 = Temperature gradient
   r3 = Thickness of layer 1
   r4 = Material set number associated with layer 1
   r5 = Material angle ($\beta$) for layer 1
   r6 = Thickness of layer 2
   r7 = Material set number associated with layer 2
   r8 = Material angle ($\beta$) for layer 2
   r[3(NL - 1) + 3] = Thickness of layer NL
   r[3(NL - 1) + 4] = Material set number associated with layer NL
   r[3(NL - 1) + 5] = Material angle ($\beta$) or layer NL

(If Op. 2 = 3 and Op. 7 = 1)

   r1 = Distance from reference plane to upper surface (r1 is positive when upper surface is above the reference plane)
   r2 = Temperature gradient
   r3 = Thickness of layers 1 and 3
   r4 = Material set number of layers 1 and 3
   r5 = Thickness of layer 2
   r6 = Material set number of layer 2

Material Properties

(See Figure 4.35 for material directions)

   EX = Modulus of elasticity in the 1st material direction
   EY = Modulus of elasticity in the 2nd material direction
KX = Thermal conductivity in the global X-direction
K = Thermal conductivity in the global Y-direction
NUXY = Poisson's ratio relating the 1st and 2nd material directions (strain in the 2nd direction due to unit strain along the 1st direction)
NUYZ = Poisson's ratio relating the 2nd and 3rd material directions (strain in the 3rd direction due to unit strain along the 2nd direction)
NUXZ = Poisson's ratio relating the 1st and 3rd material directions (strain in the 3rd direction due to unit strain along the 1st direction)
ALPX = Coefficient of thermal expansion in the 1st material direction
ALPY = Coefficient of thermal expansion in the 2nd material direction
GXY = Shear modulus relating the 1st and 2nd material direction
GXZ = Shear modulus relating the 1st and 3rd material direction
GYZ = Shear modulus relating the 2nd and 3rd material direction
DENS = Density
SIGXT = Tensile strength in the 1st material direction
SIGXC = Compressive strength in the 1st material direction
SIGYT = Tensile strength in the 2nd material direction
SIGYC = Compressive strength in the 2nd material direction
SIGXY = Shear strength in the plane defined by 1st and 2nd material directions
DAMP = Material damping coefficient

Note:
The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:
1. Moduli of elasticities in two directions are defined and are unequal.
2. Poisson's ratio in two planes are defined and are unequal.
3. Thermal coefficients in two directions are defined and are unequal.
4. Thermal conductivity in two directions are defined and are unequal.

The following conditions must be met for proper representation of orthotropic properties for the i\(^{th}\) and j\(^{th}\) material directions:

\[ \nu_{ij}/E_i = \nu_{ji}/E_j \]

Where \( \nu_{ij}, E_i, \) and \( E_j \) are provided as input and \( \nu_{ji} \) calculated internally by the program.

Element Loadings
- Thermal
- Gravitational
- Pressure (applied normal to element faces)

Output Results
Stress components and von Mises stress are available in the element coordinate directions at the center of the top and bottom surfaces of each layer. Principal stresses may also be optionally requested.
Figure 4.35  Composite Quadrilateral Plate and Shell

References


Linear Structural Composite 8/9-Node Isoparametric Shell

element_name SHELL9L

General Description

SHELL9L is an 8- or 9-node quadrilateral shell element with membrane, bending and shear capabilities for the analysis of three-dimensional structural models, constructed of fiber-reinforced laminated composites. Six degrees of freedom (three translations and three rotations) are considered per node.

The element is assumed to have constant thickness with isotropic and orthotropic material properties for each layer.

The nodal input pattern can be specified in both clockwise and counter-clockwise directions. The ninth (9th) node (if any) must be placed at the center of the element.

Special Features

Buckling, Inplane loading, P-Method (polynomial degrees up to 5).

Default Element Coordinate System (ECS = -1)

The element is a curved lamina in three dimensional space. At any point of the element curved surface the z-axis of the local coordinate system is normal to the element mid-plane and the x- and y-axes lie in the tangent plane. The direction of the element axes x and y are related to the natural coordinates (ξ, η) as shown in Figure 4.16. Given in the figure also, an example of the element coordinate system at node 3. A special case of the element coordinate system for a cylindrical surface is shown in Figure 4.17.

Other Element Coordinate Systems (ECS >-1)

When a defined element coordinate system (xe yez), as shown in Figure 4.6, is different from the default element coordinate system (ECS = -1), the program considers a modified element coordinate system (xyz) as follows:

The element z-axis is normal to the shell plane. The positive direction of z follows a right-hand rule (as directed by nodes 1, 2 and 3) as shown in Figure 4.6.

If the angle (α) between the xe axis and the normal to the surface (z) is greater than 45° (α > 45°), the element x-axis is considered as the projection of xe on the element plane.

If the angle (α) between the xe axis and the normal to the surface (z) is less than or equal to 45° (α =< 45°), the element x-axis is considered as the projection of ye on the element plane.

The element y-axis is defined so that z-axis completes a right-hand Cartesian system with x- and y-axes.

Element Group Options

**Op. 1:** Number of nodes per element
- = 0 ; Use 9-node element (default)
- = 1 ; Use 8-node element

---

*COSMOS/M CAD Interface • 12/97*
Op. 2: Integration method (see references)
   = 0 ; Using GAMA-PHI method
   = 1 ; Using reduced integration

Op. 3:  
   NL = Number of layers in the element (1 to 50)

Op. 4: Unused option for this element

Op. 5: Stress directions
   = 0 ; Calculate nodal stresses in the global Cartesian coordinate system
   = 1 ; Calculate nodal stresses in the default element coordinate system

Op. 6 to Op. 8: Unused options for this element

Real Constants

Figure 4.34 shows the convention for thickness definition and temperature distribution of a multi-layer composite shell element. For each layer, the material coordinate system (abc) is shown in Figure 4.36. The material angle ($\beta$) is defined relative to the element coordinate system.

\[ \begin{align*}
   r_1 & = \text{Distance from reference plane to upper surface (r1 is positive when upper surface is above the reference plane)} \\
   r_2 & = \text{Temperature gradient} \\
   r_3 & = \text{Thickness of layer 1} \\
   r_4 & = \text{Material set number associated with layer 1} \\
   r_5 & = \text{Material angle ($\beta$) for layer 1} \\
   r_6 & = \text{Thickness of layer 2} \\
   r_7 & = \text{Material set number associated with layer 2} \\
   r_8 & = \text{Material angle ($\beta$) for layer 2} \\
   r_{3(NL - 1) + 3} & = \text{Thickness of layer NL} \\
   r_{3(NL - 1) + 4} & = \text{Material set number associated with layer NL} \\
   r_{3(NL - 1) + 5} & = \text{Material angle ($\beta$) or layer NL}
\end{align*} \]

Material Properties

*Material properties associated with each element layer are:*

\[ \begin{align*}
   EX & = \text{Modulus of elasticity in the 1st material direction} \\
   EY & = \text{Modulus of elasticity in the 2nd material direction} \\
   NUXY & = \text{Poisson's ratio relating the 1st and 2nd material direction (strain in the 2nd direction due to unit strain along the 1st direction)} \\
   NUYZ & = \text{Poisson's ratio relating the 2nd and 3rd material direction (strain in the 3rd direction due to unit strain along the 2nd direction)} \\
   NUXZ & = \text{Poisson's ratio relating the 1st and 3rd material direction (strain in the 3rd direction due to unit strain along the 1st direction)} \\
   ALPX & = \text{Coefficient of thermal expansion in the 1st material direction} \\
   ALPY & = \text{Coefficient of thermal expansion in the 2nd material direction} \\
   GXY & = \text{Shear modulus relating the 1st and 2nd material directions} \\
   GXZ & = \text{Shear modulus relating the 1st and 3rd material directions} \\
   GYZ & = \text{Shear modulus relating the 2nd and 3rd material directions} \\
   DENS & = \text{Density} \\
   DAMP & = \text{Material damping coefficient}
\end{align*} \]
Note:
The element is assigned orthotropic material properties if at least one of the following conditions is satisfied:

1. Moduli of elasticities in two directions are defined and are unequal.
2. Poisson's ratio in two planes are defined and are unequal.
3. Thermal coefficients in two directions are defined and are unequal.

The following conditions must be met for proper representation of orthotropic properties for the \( i \)th and \( j \)th material directions:

\[ \frac{\nu_{ij}}{E_i} = \frac{\nu_{ji}}{E_j} \]

Where \( \nu_{ij} \), \( E_i \), and \( E_j \) are provided as input and \( \nu_{ji} \) calculated internally by the program.

Element Loadings
- Thermal
- Gravitational
- Pressure (applied to element faces)

Output Results
Forces per unit length and stress components are available in the element coordinate systems for each layer.

---

![Diagram of Composite 9-Node Isoparametric Shell Element]

**Figure 4.36** Composite 9-Node Isoparametric Shell Element
References


Linear Structural Axisymmetric Shell

element_name SHELLAX

General Description

SHELLAX is a 2-node straight conical shell element with constant thickness for the analysis of axisymmetric structural models. The shell element is geometrically symmetric, but the loading could be either axisymmetric or non-axisymmetric. Also in the case of frequency and buckling analysis, the mode shapes can be either axisymmetric or non-axisymmetric. In the formulation, both bending and "in plane" or "membrane" forces are considered. The shear deformation, however, is neglected making the element suitable for thin shell problems.

Structures have to be modeled in the positive X half-plane, where X represents the radial direction and Y refers to the axis of axial symmetry.

In general, four degrees of freedom are considered per node: three translations along the global X, Y and Z axes and one rotation about the global Z-axis. For static analysis with axisymmetric loading, only three degrees of freedom are considered; these are: two translations along the X and Y axes and one rotation about the Z-axis. In the case of non-axisymmetric (asymmetric) loading, and frequency and buckling analyses, however, different circumferential harmonic numbers may require the translational degree of freedom along the Z-axis to account for asymmetric conditions and modes.

The element can be used for both isotropic and orthotropic material models.

Special Features

Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)

The nodal input pattern for this element is shown in Figure 4.37. For the element coordinate system, the x-axis goes from the first node to the second. The element y-axis is perpendicular to the x-axis, in the outward direction. The element z-axis (circumferential direction) is parallel to the global Z-axis and completes a right-hand Cartesian coordinate system.

Element Group Options  (None)

Real Constants

r1 = Shell thickness

Material Properties

EX = Modulus of elasticity in the meridian direction (element x-direction)
EY = Modulus of elasticity in the circumferential direction (global Z-direction)
NUXY = Poisson's ratio
DENS = Density
DAMP = Material damping coefficient

Element Loadings

- Gravitational (in axial direction)
- Nodal loads per unit radian (both symmetric and asymmetric)
- Pressure (applied normal to element face) (both symmetric and asymmetric)

**Output Results**

Forces and moments per unit length are available for each element in the element coordinate system at the two end nodes as shown in Figure 4.37. The stress components are also calculated and printed in the element coordinate system at the center of the element.

![Element Coordinate System](image)

**Figure 4.37 Axisymmetric Shell**

**References**

Linear Structural Elastic Straight Pipe

element_name PIPE

General Description

PIPE is a 2-node uniaxial element for three-dimensional structural models. Six degrees of freedom (three translations and three rotations) are considered per node. The element can be regarded as a special case of the 3D ELASTIC BEAM for which input requirements are reduced due to the tubular cross section geometry.

Special Features

Buckling, Inplane loading.

Default Element Coordinate System (ECS = -1)

The nodal input pattern is shown in Figure 4.38. The x-axis goes from the first node to the second. A third node is defined internally by the program along a line parallel to and in the positive direction of the global Y-axis. For the special case when the element is parallel to the Y-axis, the third node is defined along a line parallel to and in the positive direction of the global Z-axis. The element y-axis lies in the plane defined by the three nodes, perpendicular to x-axis towards the third node. The element z-axis completes a right-hand Cartesian system.

Element Group Options

(None)

Real Constants

\[
\begin{align*}
  r_1 &= \text{Outer diameter} \\
  r_2 &= \text{Wall thickness} \\
  r_3 &= \text{Internal pressure}
\end{align*}
\]

Material Properties

\[
\begin{align*}
  \text{EX} &= \text{Modulus of elasticity} \\
  \text{EY} &= \text{Coefficient of thermal expansion} \\
  \text{NUXY} &= \text{Poisson's ratio} \\
  \text{DENS} &= \text{Density} \\
  \text{DAMP} &= \text{Material damping coefficient}
\end{align*}
\]

Element Loadings

- Thermal
- Gravitational
- Internal pressure

Output Results

Forces and stresses are available in the element coordinate system. The direction of forces are similar to those of the 3D BEAM element shown in Figure 4.27.
Figure 4.38 Elastic Straight Pipe

References
Fluid Flow 2D 4-Node Isoparametric

\textbf{element\_name FLOW2D}

\textbf{General Description}

FLOW2D is a 4-node quadrilateral element. It can be used to model both two dimensional and axisymmetric fluid flow problems. All the elements must be defined in the X-Y plane as shown in Figure 4.39. Axisymmetric models have to be defined in the positive X half-plane in which X represents the radial direction and Y refers to the axis of symmetry.

Each node has three (3) degrees of freedom: \( u \) and \( v \) (which are the components of velocity in X and Y directions) in addition to the temperature. For the nodal input pattern both clockwise and counter-clockwise numbering are allowed.

\textbf{Special Features} \hspace{1cm} (None)

\textbf{Element Group Options}

\textbf{Op. 1:}
\begin{itemize}
  \item \( = 0 \) ; PLANE2D element (default option)
  \item \( = 1 \) ; axisymmetric element
\end{itemize}

\textbf{Op. 2 to Op. 8:} Unused options for this element

\textbf{Real Constants} \hspace{1cm} (None)

\textbf{Material Properties}

\begin{itemize}
  \item \textbf{VISC} = Dynamic viscosity of the fluid
  \item \textbf{DENS} = Density of the fluid
  \item \textbf{KX} = Thermal conductivity of the fluid
  \item \textbf{C} = Specific heat of the fluid
  \item \textbf{BETA} = Coefficient of volumetric expansion
\end{itemize}

For compressible fluid flow analysis, only the following material properties are to be specified:

\begin{itemize}
  \item \textbf{C} = Specific heat at constant pressure
  \item \textbf{GAMMA} = Ratio of specific heat at constant pressure to specific heat at constant volume
\end{itemize}

\textbf{Element Loadings}

\begin{itemize}
  \item Thermal convection
  \item Pressure (applied normal to element faces)
  \item Internal heat generation
  \item Applied heat flux
\end{itemize}

\textbf{Output Results}

Velocity, pressure and temperature values are printed at each nodal point. The nodal pressure is an average over the elements that are connected to the node.
Figure 4.39  4-Node Isoparametric Quadrilateral Element

References
Fluid Flow 3D 8-Node Isoparametric

**element_name FLOW3D**

**General Description**
FLOW3D is an 8-node element for three-dimensional fluid flow problems. The nodal input pattern for this element is shown in Figure 4.40. Each node has four degrees of freedom: u, v, and w (which are the velocity components in X, Y and Z directions) in addition to the temperature.

**Special Features** (None)

**Element Group Options** (None)

**Real Constants** (None)

**Material Properties**
- VISC = Dynamic viscosity of the fluid
- DENS = Density of the fluid
- KX = Thermal conductivity of the fluid
- C = Specific heat of the fluid
- BETA = Coefficient of volumetric expansion

**Element Loadings**
- Thermal convection
- Pressure (applied normal to element faces)
- Internal heat generation
- Applied heat flux

**Output Results**
Velocity, pressure and temperature values are printed at each nodal point. The nodal pressure is calculated by averaging the pressure values at the same node from different elements.
References
Chapter 5

Command Reference
(This page is intentionally left blank)
Introduction

This chapter contains descriptions of all menus, submenus, and commands in the COSMOS/M CAD Interface in hierarchical order according to their appearance in the menu system. Descriptions of all commands and their syntax and usages are given.

Chapters 2 and 3 should have hopefully helped the user to get familiar with the commands so that finding the command that performs a certain function would be natural.

The commands in the COSMOS/M CAD Interface are designed to work with and without mesh from CAD programs. If you use a CAD program for meshing, loading and boundary condition specifications, then you may or may not need to create any geometric entities or elements.

Start-Up Menu

Once COSMOS/M CAD Interface has been started, the Start-Up menu appears. The following is a list of the items in that menu. (The Windows version has an additional option of changing your working directory.)

<table>
<thead>
<tr>
<th>MAIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>CAD Program</td>
</tr>
<tr>
<td>Read Geofile</td>
</tr>
<tr>
<td>Read IGES file</td>
</tr>
<tr>
<td>COSMOS/M Dbase</td>
</tr>
<tr>
<td>Exit</td>
</tr>
</tbody>
</table>

Figure 5.1 COSMOS/M CAD Interface Start-Up Menu

CAD Program

The CAD Program command transfers control to a CAD program. Control goes back to COSMOS/M CAD Interface upon exiting the CAD program.

Read GEO File

The Read GEO File command loads an existing GEOSTAR file to COSMOS/M CAD Interface. The file extension must be GEO and its contents must be in GEOSTAR format. Such GEO files may be obtained by using the G_FORM OUT command inside GEOSTAR. GEOSTAR session files may also be read through this command.
Read GEO File

Input Description:
- Name of model without extension.

Read IGES File

The Read IGES File command inputs an IGES file.

Input Description:
- Name of the source CAD program used to generate the IGES file.
- Name of IGES file.

COSMOS/M Dbase

The COSMOS/M Dbase command loads an existing COSMOS/M database.

Input Description:
- Database name. (Problem name without extension.)

Exit

Exit closes the interface and returns control to the system window upon confirmation.

Input Description:
- Y(es) or N(o).
  (default is N)
Main Menu

Once the new database is created or an existing one is loaded, the Main Menu is displayed.

<table>
<thead>
<tr>
<th>MAIN</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>CAD Program</td>
<td>Geometry</td>
</tr>
<tr>
<td>Properties</td>
<td>Meshing</td>
</tr>
<tr>
<td>Loads/BConds</td>
<td>Viewing</td>
</tr>
<tr>
<td>Analysis</td>
<td>Postprocessing</td>
</tr>
<tr>
<td>Exit</td>
<td>Selection</td>
</tr>
<tr>
<td>Utilities</td>
<td>Flow Chart</td>
</tr>
<tr>
<td>What Next?</td>
<td>Import/Export</td>
</tr>
<tr>
<td>GEOSTAR Command</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.2 COSMOS/M CAD Interface Main Menu

CAD Program

Runs the CAD program upon confirmation.

Input Description:
- Y(es) or N(o)
  (default is N)

Geometry Menu

Opens the menu for geometric entities. Commands to create, list, and delete geometric entities are contained in this menu.

Properties Menu

Opens the menu for defining element groups and their attributes like material and real constant sets.
Meshing Menu
Opens the menu for meshing entities and manipulating nodes and elements.

Loads/BConds Menu
Opens the menu for defining loads and boundary conditions. Loads and boundary conditions may be applied directly to nodes and elements or through association with geometric entities.

Viewing Menu
Opens the menu for setting of view and display options.

Analysis Menu
Opens the menu for specifying various analysis options and submitting the model for solution.

Postprocessing Menu
Opens the menu for the examination of solution results in both text and graphic modes.

Exit
Closes the interface and returns to the system window upon confirmation.

Input Description:
- Y(es) or N(o)
  (default is N)

Selection Menu
Opens the menu for initializing, creating, and manipulating selection lists.

Utilities Menu
Opens the menu for general utility commands and hardcopy devices.

Flow Chart
A utility to guide the user throughout the steps of performing a successful analysis.
What Next?
Displays a screen of help that tells the user about the status of the model and what the next step might be.

Import/Export
Menu to import or export geometry. Currently not available.

GEOSTAR Command
Gives access to all GEOSTAR commands (no menu or help facilities are available). This option is not intended for extensive use and the user must be familiar with GEOSTAR or should refer to the COSMOS/M User Guide and Command Reference Manual in order to be able to use it to get access to the options that are available in GEOSTAR but are not available in the interface. The GEOSTAR Command Section in Chapter 3 gives help for some important commands that the user may need. A menu is also opened with the following options:

<table>
<thead>
<tr>
<th>GEOSTAR Command</th>
<th>Option &amp; Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Snap On</td>
<td>Allows keypoints to be snapped to a defined grid. (default is on)</td>
</tr>
<tr>
<td>Snap Off</td>
<td>Suppresses the snapping capability.</td>
</tr>
<tr>
<td>Pick On</td>
<td>Allows entities to be picked by the mouse. (default is on)</td>
</tr>
<tr>
<td>Pick Off</td>
<td>Suppresses the picking capability.</td>
</tr>
<tr>
<td>Repeat On</td>
<td>After the execution of a command, the same command can be repeated by hitting the enter key or left hand mouse button. (default is on)</td>
</tr>
<tr>
<td>Repeat Off</td>
<td>Disables the repeat facility.</td>
</tr>
<tr>
<td>Exit GEOSTAR Mode</td>
<td>Returns to the CAD Interface.</td>
</tr>
</tbody>
</table>
Point Submenu

Keypoints are the basic geometric entity and will be created as part of the geometry transfer from CAD programs. Selecting this option opens the menu for keypoint commands containing:
Choose the operation type, **All** or **Pattern**, and **Accept** or **Quit**. The **All/Pattern** option is used with **Plot**, **List**, **Delete**, and **Merge** only.

**Define Point**

Creates a keypoint by defining its coordinates in the active coordinate system. The **All/Pattern** option is ignored.

**Input Description:**
- X-coordinate.  
  (default is 0.0)
- Y-coordinate.  
  (default is 0.0)
- Z-coordinate.  
  (default is 0.0)

**Plot Point**

Plots all or a selected pattern of keypoints. If a selection list exists, then only keypoints in the selection list will be plotted. The pattern information is only prompted if **Pattern** is selected.

**Input Description:**
- First keypoint in the pattern.  
  (default is 1)
- Last keypoint in the pattern.
  (default is highest keypoint)

- Increment in the pattern.
  (default is 1)

List Point

Lists all or a selected pattern of keypoints. If a selection list exists, then only keypoints in the selection list will be listed. The pattern information is only prompted if Pattern is selected.

<table>
<thead>
<tr>
<th>List Point</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Description:</strong></td>
</tr>
<tr>
<td>- First keypoint in the pattern.</td>
</tr>
</tbody>
</table>
  (default is 1) |

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Last keypoint in the pattern.</td>
</tr>
</tbody>
</table>
  (default is highest keypoint) |

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Increment in the pattern.</td>
</tr>
</tbody>
</table>
  (default is 1) |

Delete Point

Deletes all or a selected pattern of keypoints. If a selection list exists, then only keypoints in the selection list will be deleted. The pattern information is only prompted if Pattern is selected.

<table>
<thead>
<tr>
<th>Delete Point</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Description:</strong></td>
</tr>
<tr>
<td>- First keypoint in pattern.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Last keypoint in pattern.</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Increment in the pattern.</td>
</tr>
</tbody>
</table>
  (default is 1.) |

**Example:** 1, 5, 2 deletes keypoints number 1, 3, and 5.

Delete Pick Point

Delete a keypoint by picking. The All/Pattern option is ignored. After choosing Accept, pick the keypoint to be deleted, choose again to confirm, or use the right button to choose other keypoints.
Delete Pick Point

Identify Point

Highlights the keypoint selected by the mouse and displays its coordinates. The keypoint must have been plotted on the screen. The All/Pattern option is ignored.

Input Description:
- Pick the required keypoint. Use the right button of the mouse to identify additional keypoints in the neighborhood.

Merge Point

Merges all or a selected pattern of keypoints. Any selected keypoints whose X-, Y-, and Z-coordinates are within the specified tolerance from each other are merged.

Input Description:
- First keypoint in pattern.
  (default is last in pattern - first)
- Last keypoint in pattern.
- Increment in the pattern.
- Merge tolerance.
  (default is 0.0001)

All

Operate on all keypoints. Used with the List, Delete, and Merge options. Pattern information will not be prompted.

Pattern

Operate on the pattern of keypoints to be specified. Used with the List, Delete, and Merge options. After choosing Accept, the user will be prompted to specify beginning, ending, and increment of the pattern.
Pattern

Accept
Accept command options.

Accept

Quit
Returns to the Geometry menu.

Quit

Curve Submenu
Curves are 1-D parametric entities and are created as part of the geometry transfer. This selection opens the menu for curve commands containing:

<table>
<thead>
<tr>
<th>CURVE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete</td>
</tr>
<tr>
<td>Delete Pick</td>
</tr>
<tr>
<td>Identify</td>
</tr>
<tr>
<td>Merge</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Pattern</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.5  Curve Submenu

Choose the operation type, All or Pattern, and Accept or Quit. The All/Pattern option is used with Plot, List, Delete, and Merge only.

Define Curve
Creates a new straight line or redefines an existing one. The All/Pattern option is ignored.
Define Curve

Input Description:
- Curve label.
  (default is highest defined + 1)
- First keypoint label.
- Second keypoint label.

Plot Curve

Plots all or a selected pattern of curves. If a selection list exists for curves, then only curves in the selection list will be plotted. The pattern information is only prompted if Pattern is selected.

List Curve

Lists all or a selected pattern of curves. If a selection list exists for curves, then only curves in the selection list will be listed. The pattern information is only prompted if Pattern is selected.

Delete Curve

Deletes the selected pattern of curves. If a selection lists exists for curves, then only curves in the selection lists will be deleted.
- Last curve in pattern.
- Increment in the pattern.
  (default is 1)

**Example:** 2, 9, 3 deletes curves number 2, 5, and 8.

## Delete Pick Curve
Delete a curve by picking. The **All/Pattern** option is ignored. After choosing **Accept**, pick the curve to be deleted, choose again to confirm, or use the right button to choose other curves.

### Delete Pick Curve

## Identify Curve
Highlights the selected curve and displays its label. The curve must have been plotted on the screen. The **All/Pattern** option is ignored.

### Identify Curve

**Input Description:**
- Pick the required curve. Use the right button of the mouse to identify additional curves in the neighborhood.

## Merge Curve
Merges all or a selected pattern of curves. Any selected curves whose corresponding end, one third and two-thirds points are within the specified tolerance from each other are merged. The pattern information is only prompted if **Pattern** is selected.

### Merge Curve

**Input Description:**
- First curve in pattern.
- Last curve in pattern.
- Increment in the pattern.
  (default is last in pattern - first)
- Merge tolerance.
  (default is 0.0001)
All
Operate on all curves. Used with the **List**, **Delete**, and **Merge** options. Pattern information will not be prompted.

Pattern
Operate on the pattern of curves to be specified. Used with the **List**, **Delete**, and **Merge** options. After choosing **Accept**, the user will be prompted to specify beginning, ending, and increment of the pattern.

Accept
Accept command options.

Quit
Returns to Geometry menu.

Contour Submenu
Contours are a closed-loop set of curves. The curves must be joined. One curve is sufficient to define the contour if no alternative path exists, otherwise more curves are needed to define a unique path. These entities are created as part of the geometry transfer but the user may modify them in the interface if desired. Selecting this option opens the menu for contour commands containing:
Choose the operation type, All or Pattern, and Accept or Quit. The All/PATTERN option is used with Plot, List and Delete only.

**Define Contour**

Creates a new contour, or redefines an existing one. The All/PATTERN option is ignored.

### Define Contour

**Input Description:**
- Reference number for the contour.
  (default is highest defined + 1)
- Meshing flag.
  = 0    Mesh with average element size.
  = 1    Mesh with number of elements on boundary.
  (default is 0)
- Average element size if meshing flag = 0 or approximate number of elements along the contour boundary if meshing flag = 1.
- Number of reference curves to define the contour (limit is 250).
  (default is 1)
- Pick the ith curve (i = 1, 2, 3, ..., 250).
- Use selection set (not yet supported).
  = 0    No
  = 1    Yes
  (default is 0)
- Redefinition criteria for the common curves of adjoining contours:
  = 0    Use previous definition.
  = 1    Redefine the mesh at the common curve.
= 2 Use the definition that gives the maximum number of elements on the curve.
= 3 Use the definition that gives the minimum number of elements on the curve.

**Plot Contour**

Plots all or a selected pattern of contours. If a selection list exists, then only contours in the selection list are plotted. The pattern information is only prompted if Pattern is selected.

**Plot Contour**

**Input Description:**
- First contour in the pattern.
  (default is 1)
- Last contour in the pattern.
  (default is highest contour)
- Increment in the pattern.
  (default is 1)

**List Contour**

Lists all or a selected pattern of contours. If a selection list exists, then only the contours in the selection list are listed. The pattern information is only prompted if Pattern is selected.

**List Contour**

**Input Description:**
- First contour in the pattern.
  (default is 1)
- Last contour in the pattern.
  (default is highest contour)
- Increment in the pattern.
  (default is 1)

**Delete Contour**

 Deletes all or a selected pattern of contours. If a selection list exists, then only the contours in the selection list are deleted. The pattern information is only prompted if Pattern is selected.
Input Description:
- First contour in pattern.
- Last contour in pattern.
- Increment in the pattern.
  (default is 1)

Example: 9, 16, 3 deletes contours 9, 12, and 15.

Delete Pick Contour

Delete a contour by picking. The All/Pattern option is ignored. After choosing Accept, pick the contour to be deleted, choose again to confirm, or use the right button to choose other contours.

Identify Contour

Highlights the selected contour and displays its label. The contour must have been plotted on the screen. The All/Pattern option is ignored.

Input Description:
- Pick the required contour. Use the right button of the mouse to identify additional contours in the neighborhood.

Modify Contour

Modifies an existing contour by replacing one of its curves by one or more other existing curves. The command can also be used to redefine the number of elements on a curve by replacing it with itself and specifying the desired number of elements for it. The All/Pattern option is ignored.

Input Description:
- Pick contour to be modified.
- Pick curve to be modified.
- Number of replacing curves (limit 10).
  (default is 1)
- Pick ith replacing curve.
- Number of elements on ith curve.
  (default is 4)

**All**

Operate on all contours. Used with the List and Delete options. Pattern information will not be prompted.

**Pattern**

Operate on the pattern of contours to be specified. Used with the List and Delete options. After choosing Accept, the user will be prompted to specify beginning, ending, and increment of the pattern.

**Accept**

Accept command options.

**Quit**

Returns to Geometry menu.

**Region Submenu**

A region is an area definition composed of outer and inner contours. The region is the area enclosed between the outer and inner contours. Regions are automatically created as part of the geometry transfer from a CAD program. Curved regions imported from a CAD program are automatically defined by underlying surfaces. A maximum of 100 contours can be used to define a region. Selecting this item opens the menu for region commands containing:
Choose the operation type, **All** or **Pattern**, and **Accept** or **Quit**. The **All/Pattern** option is used with **Plot**, **List** and **Delete** only.

### Define Region

Creates a region from 1 outer contour and up to 99 inner contours. A curved region must be defined with an underlying surface from which it references the curvature of the desired region. No underlying surface is needed for a flat region. The **All/Pattern** option is ignored.

**Input Description:**
- Reference number for new region.
  (default is highest defined + 1)
- Number of contours needed to define the region.
  (default is 1)
- Pick the outer contour.
- Pick the ith inner contour (i = 1, 2, 3, ..., 99).
- Underlying surface to define curvature. Use default for flat regions.
  (default is 0)

### Plot Region

Plots all or a selected pattern of regions. If a selection list exists, only the regions in the selection list are plotted. The pattern information is only prompted if **Pattern** is selected.
Input Description:
- First region in the pattern.
  (default is 1)

- Last region in the pattern.
  (default is highest region)

- Increment in the pattern.
  (default is 1)

List Region
Lists all or a selected pattern of regions. If a selection list exists, only the regions in the selection list are listed. The pattern information is only prompted if Pattern is selected.

Delete Region
Deletes all or a selected pattern of regions. If a selection list exists, only the regions in the selection list are deleted. The pattern information is only prompted if Pattern is selected.

Example: 10, 20, 4 deletes regions number 10, 14, and 18.
Delete Pick Region

Delete a region by picking. The All/Pattern option is ignored. After choosing Accept, pick the region to be deleted, choose again to confirm, or use the right button to choose other regions.

Identify Region

Highlights the selected region and displays its label. The region must have been plotted on the screen. The All/Pattern option is ignored.

Input Description:
- Pick the desired region. Use the right button of the mouse to select neighboring regions.

Reorient Region

Reverses the direction of the normal for the region and its associated elements. This is particularly important in the case of shell elements since it determines the bottom and top faces.

Input Description:
- Pick region.

All

Operate on all regions. Used with the List and Delete options. Pattern information will not be prompted.

Pattern

Operate on the pattern of regions to be specified. Used with the List and Delete options. After choosing Accept, the user will be prompted to specify beginning, ending, and increment of the pattern.
Accept

Accept command options.

Quit

Returns to Geometry menu.

Polyhedron Submenu

A polyhedron in COSMOS/M CAD Interface is a continuous air-tight multi-sided closed boundary defined by a group of regions. Polyhedrons are created as part of the geometry transfer. The user may wish to redefine them as to produce a different mesh density. The command performs a closed loop check so the polyhedron will not be created if a unique closed loop is not found within the specified tolerance. The polyhedron is not created if free regions are connected to the set of regions forming the closed loop. Use commands in the Select menu to unselect undesired regions. Selecting this item opens the menu for polyhedron commands containing:

<table>
<thead>
<tr>
<th>POLYHEDRON</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete</td>
</tr>
<tr>
<td>Delete Pick</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Pattern</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.8 Polyhedron Submenu

Choose the operation type, All or Pattern, and Accept or Quit. The All/Pattern option is used with Plot, List and Delete only.
Define Polyhedron

Creates a polyhedron from a closed loop of regions. The **All/Pattern** option is ignored.

**Input Description:**
- Label of polyhedron to be created.
- Pick a region to be used for defining the polyhedron. Only one region is needed as the software performs a closed loop check. Use commands in the Select menu to unselect undesired regions. The maximum number of regions that can be used for a polyhedron is 1000.
- Input the average element size. This is the average height of the Tetrahedral elements. The length of each side will be between 2/3rd's and 4/3rd's of this value.
- Input the tolerance for the closed loop check. (default is 0.0001)
- Redefine the element size on the boundary.
  - = 0 Use contour definition.
  - = 1 Redefine. (default is 1)

Plot Polyhedron

Plots all or a selected pattern of polyhedra. The pattern information is only prompted if **Pattern** is selected.

**Input Description:**
- First polyhedra in the pattern. (default is 1)
- Last polyhedra in the pattern. (default is highest polyhedra)
- Increment in the pattern. (default is 1)

List Polyhedron

Lists all polyhedra.
Input Description:
- First polyhedron in the pattern.
  (default is 1)

- Last polyhedron in the pattern.
  (default is highest polyhedra)

- Increment in the pattern.
  (default is 1)

Delete Polyhedron

Deletes all or a selected pattern of polyhedra. The pattern information is only prompted if Pattern is selected.

Delete Pick Polyhedron

Delete a polyhedron by picking. The All/Pattern option is ignored. After choosing Accept, pick the polyhedron to be deleted, choose again to confirm, or use the right button to choose other polyhedra.

All

Operate on all polyhedra. Used with the List and Delete options. Pattern information will not be prompted.
Pattern
Operate on the pattern of polyhedra to be specified. Used with the List and Delete options. After choosing Accept, the user will be prompted to specify beginning, ending, and increment of the pattern.

Accept
Accept command options.

Quit
Returns to Geometry menu.

Part Submenu
A part is the volumetric representation of the 3-D space enclosed by a single polyhedron or a group of polyhedra. The space enclosed by the outer polyhedron and the inner ones forms the part. Polyhedra forming a part should not intersect with one another. Parts are automatically created as part of the geometry transfer but will need to be redefined when changing the mesh size on a polyhedron. This selection opens the menu for parts containing:

<table>
<thead>
<tr>
<th>PART</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete</td>
</tr>
<tr>
<td>Delete Pick</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Pattern</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.9 Part Submenu
Choose the operation type, **All** or **Pattern**, and **Accept** or **Quit**. The **All/Pattern** option is used with **Plot**, **List** and **Delete** only.

### Define Part

Creates a part from a group of polyhedra. The **All/Pattern** option is ignored.

#### Define Part

**Input Description:**
- Label of part to be created.
  (default is highest defined + 1)
- Enter ith polyhedron to be used in the definition (maximum 100).

### Plot Part

Plots all or a selected pattern of parts. The pattern information is only prompted if **Pattern** is selected.

#### Plot Part

**Input Description:**
- First part in the pattern.
  (default is 1)
- Last part in the pattern.
  (default is highest part)
- Increment in the pattern.
  (default is 1)

### List Part

Lists all or a selected pattern of parts. The pattern information is only prompted if **Pattern** is selected.

#### List Part

**Input Description:**
- First part in the pattern.
  (default is 1)
- Last part in the pattern.
  (default is highest part)
- Increment in the pattern.
  (default is 1)

**Delete Part**

Deletes all or a selected pattern of parts.

**Delete Part**

**Input Description:**
- Enter the first part in pattern.
- Enter last part in pattern.
  (default is first part label)
- Increment in the pattern.
  (default is 1)

**Example:** 3, 9, 2 deletes parts number 3, 5, 7, and 9.

**Delete Pick Part**

Delete a part by picking. The All/Pattern option is ignored. After choosing Accept, pick the part to be deleted, choose again to confirm, or use the right button to choose other parts.

**Delete Pick Part**

**All**

Operate on all parts. Used with the List and Delete options. Pattern information will not be prompted.

**Pattern**

Operate on the pattern of parts to be specified. Used with the List and Delete options. After choosing Accept, the user will be prompted to specify beginning, ending, and increment of the pattern.
Accept
Accept command options.

Quit
Returns to Geometry menu.

Troubleshooting

Polyhedron Not Created
If the Polyhedron is not created when using Polyhedron Define, repeat the command specifying a higher tolerance until the polyhedron is formed. Do not specify a tolerance greater than $1/10^\text{th}$ of the element size. Start with tolerances of $1/100^\text{th}$ or smaller than the element size.

If the polyhedron is not closed, the curves at the free edges are plotted on the screen. The normal case will be that two regions are not properly joined and there will be two curves overlapping at the join. Using the Curve Merge command, pick the two curves to be merged and use the default increment. The tolerance to be used for merging should be carefully chosen since too large a value will destroy the geometry. Repeat the curve merging for all occurrences of free edges (all curves plotted while defining the polyhedron).

If a message such as "Invalid Contour" appears at the time of meshing, it means that some of the curves may have become degenerate or improper curves are merged due to the use of large tolerances during Curve Merge. If this occurs examine the contours, curves, and keypoints. If the end points of the curves of the contour do not share a single keypoint, use the Point Merge command and merge the keypoints two at a time. If any of the contours appear to have missing curves or be disjointed, you must exit, delete the database and read in the input file <partname>.GEO.

Coord System Submenu
This submenu contains commands to Define, Plot, List, or Delete coordinate systems. The Cartesian/Cylindrical/Spherical option is used with Define only. Refer to Figure 3.2 in Chapter 3 for more details on the types of coordinate systems.
Define Coordinate System

To define a coordinate system, choose the type and you will be prompted for the label and 3 keypoints to make a right-handed coordinate system. The label may be 3-500. Sine systems 0, 1, and 2 are reserved for the global predefined Cartesian, cylindrical, and spherical system respectively. The All/PATTERN option is ignored.

Input Description:
- Label of the coordinate system.
- Keypoint at the origin.
- Keypoint on the x-axis.
- Keypoint on the x-y plane.

Plot Coordinate System

Plots all or a selected pattern of coordinate systems. The pattern information is only prompted if PATTERN is selected.

Input Description:
- Beginning coordinate system.
- Ending coordinate system.
- Increment.
**List Coordinate System**

Lists all or a selected pattern of coordinate systems. The pattern information is only prompted if **Pattern** is selected.

**Input Description:**
- Beginning coordinate system.
- Ending coordinate system.
- Increment.

**Delete Coordinate System**

Deletes all or a selected pattern of coordinate systems. The pattern information is only prompted if **Pattern** is selected.

**Input Description:**
- Beginning coordinate system.
- Ending coordinate system.
- Increment.

**All**

Operate on all coordinate systems. Used with the **Plot**, **List**, and **Delete** options. Pattern information will not be prompted.

**Pattern**

Operate on the pattern of coordinate systems to be specified. Used with the **Plot**, **List**, and **Delete** options. After choosing **Accept**, the user will be prompted to specify beginning, ending, and increment of the pattern.
Accept
Accept command options.

Quit
Returns to Geometry menu.
PROPERTIES Menu

This menu contains commands to define, list, and delete, element groups, material property sets, and real constant sets.

Figure 5.11 Properties Menu
Element Groups Submenu

This menu provides commands to define, list or delete the type of element to be used in the analysis.

<table>
<thead>
<tr>
<th>ELEMENT GROUPS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define Group</td>
</tr>
<tr>
<td>List Groups</td>
</tr>
<tr>
<td>Delete Groups</td>
</tr>
</tbody>
</table>

Figure 5.12   Element Groups Submenu

Define Group Submenu

Specifies the element group to be used in the analysis. You will be prompted with options related to selected element groups. Refer to Chapter 4 for description of options and details of capabilities of various elements.
### Define Group

#### Input Description:
- Reference number for the group.
  (default is 1)
- Name of the element to be used.
  - **Tetra4**: 4-node tetrahedral solid.
  - **Tetra4R**: 4-node tetrahedral solid with rotational degrees of freedom.
  - **Tetra10**: 10-node tetrahedral element.
  - **Shell3**: 3-node thin shell.
  - **Shell3T**: 3-node thick shell.
  - **Shell4**: 4-node thin shell.
  - **Shell4T**: 4-node thick shell.
  - **Shell9**: 8- or 9-node shell.
  - **Triang**: 3-6 node plane.
  - **Plane2D**: 4- to 8-node plane element.
  - **Truss2D**: 2-node, 2D truss.

![Define Group Submenu](image-url)
Truss3D  2-node, 3-D truss.
Beam2D   2-node, 2D beam.
Beam3D   3-node, 3D beam.
Clink    2-node convection link.
Rlink    2-node radiation link.
Spring   2-node spring.
Gap      2-node gap.
Mass     1-node concentrated mass.
Shell3L  3-node composite shell element.
Shell4L  4-node composite shell element.
Shell9L  8- or 9-node composite shell element.
Shellax  2-node axisymmetric shell element.
Pipe     2-node uniaxial beam element with tubular cross-section.
Flow2D   4-node quadrilateral fluid flow element.
Flow3D   8-node element for three-dimensional fluid flow problems.

List Groups

Lists all defined groups and shows active options.

Delete Groups

Deletes the specified pattern of element groups.

Input Description:
- Enter first group for deletion.
- Enter last group in pattern.
  (default is first group input)
- Increment in pattern.
  (default is 1)

Material Submenu

This submenu contains commands to define, list and delete material property sets. The defined material set becomes the active material set.
Define Set Submenu

This opens the menu shown below.

![Define Set Menu](image)

You should highlight the material properties you wish to define then select **Accept**. Input is transferred to the Message Window where you should enter the material set number you wish to define and the values of the properties you have selected.

List Material Sets

Lists all of the defined material sets and the property values assigned.
Delete Material Sets

Deletes the specified material sets.

**Delete Material Sets**

**Input Description:**
- Enter first material set in pattern.
- Enter last material set in pattern.
  (default is first set input)
- Increment in pattern.
  (default is 1)

Pick Material

Allows the user to select a material from the library.

**Pick Material**

**Input Description:**
- Set number to be defined.
  (default is 1)
- Name of material.
  
<table>
<thead>
<tr>
<th>Code</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>A_STEEL</td>
<td>ALLOY STEEL</td>
</tr>
<tr>
<td>AIR</td>
<td>Air</td>
</tr>
<tr>
<td>ACRYLIC</td>
<td>ACRYLIC (MEDIUM-HIGH IMPACT)</td>
</tr>
<tr>
<td>ALUMINUM</td>
<td>ALUMINUM ALLOY</td>
</tr>
<tr>
<td>AL_1345</td>
<td>ALUMINUM 1345 ALLOY</td>
</tr>
<tr>
<td>AL_1350</td>
<td>ALUMINUM 1350 ALLOY</td>
</tr>
<tr>
<td>AL_2014</td>
<td>ALUMINUM 2014 ALLOY</td>
</tr>
<tr>
<td>AL_2018</td>
<td>ALUMINUM 2018 ALLOY</td>
</tr>
<tr>
<td>AL_2024</td>
<td>ALUMINUM 2024 ALLOY</td>
</tr>
<tr>
<td>AL_3003</td>
<td>ALUMINUM 3003 ALLOY</td>
</tr>
<tr>
<td>AL_6061</td>
<td>ALUMINUM 6061 ALLOY</td>
</tr>
<tr>
<td>AL_7079</td>
<td>ALUMINUM 7079 ALLOY</td>
</tr>
<tr>
<td>AL_BRONZE</td>
<td>ALUMINUM BRONZE</td>
</tr>
<tr>
<td>BRASS</td>
<td>SILICON BRASS &amp; BRONZE</td>
</tr>
<tr>
<td>BRONZE</td>
<td>BRONZE</td>
</tr>
<tr>
<td>CA_STEEL</td>
<td>CAST ALLOY STEEL (BELOW 8% CONTENT)</td>
</tr>
<tr>
<td>COBALT</td>
<td>COBALT</td>
</tr>
<tr>
<td>COPPER</td>
<td>COPPER</td>
</tr>
<tr>
<td>CS_STEEL</td>
<td>CAST STAINLESS STEEL (CF-8M OR CF-20)</td>
</tr>
<tr>
<td>GC_IRON</td>
<td>GRAY CAST IRON (ASTM - CLASS40)</td>
</tr>
<tr>
<td>GLASS</td>
<td>GLASS</td>
</tr>
<tr>
<td>GOLD</td>
<td>PURE GOLD</td>
</tr>
<tr>
<td>IRON</td>
<td>IRON</td>
</tr>
<tr>
<td>LEAD</td>
<td>PURE LEAD</td>
</tr>
</tbody>
</table>
MAGNES | MAGNESIUM ALLOY - WROUGHT OR CAST  
MC_IRON | MALLEABLE CAST IRON (ASTM - A220)  
MN_BRONZE | MANGANESE_BRONZE  
MOLYBDENUM | MOLYBDENUM  
MONEL | MONEL 400  
NICKEL | NICKEL  
D_NICKEL | DURANICKEL 301  
NYLON | NYLON 6/10  
PC_STEEL | PLAIN CARBON STEEL  
PORCELAIN | CERAMIC PORCELAIN  
RUBBER | RUBBER  
SILVER | PURE SILVER  
STEEL | STEEL  
ST_1020 | STEEL, AISI C1020 (HOT WORKED)  
ST_304 | STEEL, AISI 304 (SHEET)  
ST_ST | STEEL, STAINLESS  
T_BRONZE | TIN BEARING BRONZE  
TITANIUM | TITANIUM  
TUNGSTEN | TUNGSTEN  
VANADIUM | VANADIUM  
WATER | Water  
W_COPPER | WROUGHT COPPER  
WS_STEEL | WROUGHT STAINLESS STEEL  
ZIRCONIUM | ZIRCONIUM  
(default is A_STEEL)

- Units to be used.
  FPS = (inch, pound, second)  
  SI = (meter, kilogram, second)  
  MKS = (centimeter, kilogram, second)

Note:

1. The user can add materials to the library by editing the file PICKMAT.LIB. Appendix C lists the contents of the material library.

Material Library

The Material Library command temporarily creates a new window and runs a material browser which may be used to define material property sets. The browser provides a large library of materials and associated temperature curves compiled by InfoDex. After selecting the desired materials and exiting the browser, the selected materials and associated temperature curves will be automatically loaded into the database. The Additional Prop command may be used to add or modify any material properties in the set. The browser is an add-on utility that must be acquired before executing this command. Contact SRAC for more information.
# Additional Prop

Defines additional properties for a material set. The types of properties that can be defined are listed below. The defined value for some properties is associated with the active time curve, or temperature curve. The defined material property set becomes the default and stays so until another material property set is created, or the another set is activated. Refer to the Units appendix for consistent sets of units.

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Material property set number (between 1 and 90 except for NSTAR where limit is 20). (default is highest defined + 1)</td>
</tr>
<tr>
<td>- Name of the material property. The admissible property names are:</td>
</tr>
<tr>
<td>ALPH1</td>
</tr>
<tr>
<td>ALPH2</td>
</tr>
<tr>
<td>ALPH3</td>
</tr>
<tr>
<td>ALPH4</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>ALPY</td>
</tr>
<tr>
<td>ALPZ</td>
</tr>
<tr>
<td>BETA</td>
</tr>
<tr>
<td>C</td>
</tr>
<tr>
<td>COHESN</td>
</tr>
<tr>
<td>CREEPC</td>
</tr>
<tr>
<td>CREEPX</td>
</tr>
<tr>
<td>DAMP</td>
</tr>
<tr>
<td>DC_{ij} i=1,3 and j=i,3. A total of 6 constants. To define the upper half of the dielectric material matrix (used for Piezoelectric applications in STAR only).</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>EMIS</td>
</tr>
<tr>
<td>ETAN</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>EY</td>
</tr>
<tr>
<td>EZ</td>
</tr>
<tr>
<td>FRCANG</td>
</tr>
<tr>
<td>G1 TO G8</td>
</tr>
<tr>
<td>GAMMA</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>GXZ</td>
</tr>
<tr>
<td>GYZ</td>
</tr>
<tr>
<td>HC</td>
</tr>
<tr>
<td>K1 to K8</td>
</tr>
<tr>
<td>KX</td>
</tr>
</tbody>
</table>
KY  Thermal conductivity in the y-direction of the coordinate system defined by ECS (for each element), which must be 0, -1, or user-defined. For 3-D elements, the global Cartesian system is always used.

KZ  Thermal conductivity in the z-direction of the coordinate system defined by ECS (for each element), which must be 0, -1, or user-defined. For 3-D elements, the global Cartesian system is always used.

Mcij  i=1,6 and j=i,6. A total of 21 constants. For STAR: To define the upper half of the anisotropic material stiffness matrix.

MOONEY_A  First material constant for Mooney-Rivlin hyperelastic material model.

MOONEY_B  Second material constant for Mooney-Rivlin hyperelastic material model.

MOONEY_C  Third material constant for Mooney-Rivlin hyperelastic material model.

MOONEY_D  Fourth material constant for Mooney-Rivlin hyperelastic material model.

MOONEY_E  Fifth material constant for Mooney-Rivlin hyperelastic material model.

MOONEY_F  Sixth material constant for Mooney-Rivlin hyperelastic material model.

MU1  First constant for the Ogden material model.

MU2  Second constant for the Ogden material model.

MU3  Third constant for the Ogden material model.

MU4  Fourth constant for the Ogden material model.

NUXY  Poisson's ratio, relating strain in the second material direction to strain in the first material direction.

NUXZ  Poisson's ratio, relating strain in the third material direction to strain in the first material direction.

NUYZ  Poisson's ratio, relating strain in the third material direction to strain in the second material direction.

Pcij  i=1,6 and j=1,3. A total of 18 constants to define the Piezoelectric material matrix (used in STAR only).

REFTMP  Glassy transition temperature for visco-elasticity.

SIGXC  Compressive strength in the first material direction.

SIGXT  Tensile strength in the first material direction.

SIGXYC  Compressive shear strength in the plane defined by the first and second material directions.

SIGXYT  Tensile shear strength in the plane defined by the first and second material directions.

SIGYC  Compressive strength in the second material direction.

SIGYLD  Yield stress.

SIGYT  Tensile strength in the second material direction.

TAUG1 to TAUG8  Time values associated with G1 through G8, respectively and are used in visco-elasticity.

TAUK1 to TAUK8  Time values associated with K1 through K8, respectively and are used in visco-elasticity.

VC1 and VC2  Constants for the Williams-Landel-Ferry equation used in viscoelasticty (refer to the COSMOS/M CAD Interface Advanced Modules Manual).

VISC  Dynamic viscosity.
Value(s) of the material property. CREEPC needs three values to be entered, CREEPX needs seven.
(default is 0.0)

You may continue to define material properties. A blank entry for material property name will terminate the command.

Notes:

1. Some material properties are associated with the active temperature curve. The following is a listing of these properties for different modules:
   a. For STAR, DSTAR, and ASTAR:
      ALPX  ALPY  ALPZ  DENS  EX  EY
      EZ    GXY   GYZ   GXZ   NUXY NUYZ
      NUXZ  MCij  Dcij  PCij  DAMP SIGXT
      SIGYT SIGXC SIGYC SIGXYT SIGXYC
   b. For NSTAR:
      ALPX  ALPY  ALPZ  EX  EY
      EZ    GXY   GYZ   GXZ   NUXY
      NUYZ  NUXZ  GYZ   GXZ   NUXY
      ETAN  SIGYLD
      G1, G2, ..., G8 AND K1, K2, ..., K8 are used in the viscoelastic material model in NSTAR and are temperature-dependent using the William-Landel-Ferry function, and time-dependent based on TAUG1 through TAUG8, and TAUK1 through TAUK8.  (Refer to the COSMOS/M Advanced Modules Manual.)
   c. For HSTAR:
      C      DENS  KX  KY  KZ  EMIS  HC  ECONX
      EMIS and HC are also associated with the active time curve. The input value is multiplied by corresponding values from both of the time and temperature curves. Note that HSTAR uses the average of the element face temperature and the ambient temperature to enter the temperature curve for HC.

2. It is good practice to use the following sequence of commands to associate material properties with a desired temperature, time curve, or both:
   a. Activate the curve to be used for association:
      Activate a temperature or time curve.
   b. Define material properties:
      Material properties defined here will be associated with the active curve(s), if that capability is supported.
   c. Activate temperature or time curve zero.
   d. Define material properties:
      Material properties defined here have fixed values and will not be associated with any temperature or time curves.

3. For NSTAR, the following table shows the temperature dependent properties used for different material models in association with certain element groups:
4. For the Mooney-Rivlin material model:
   a. The sum of MOONEY_A and MOONEY_B must be greater than zero.
   b. The Mooney-Rivlin model should be used if Poisson’s ratio is greater than 0.48.
   c. The Mooney-Rivlin strain energy density function is expressed as:
      \[ W = W_1 + W_2 \]
      \[ W_1 = A(I-3) + B(II-3) + X[1/(III)**2 -1] + Y(III -1)**2 \]
      \[ W_2 = C(I-3)(II-3) + D(I-3)**2 + E(II-3)**2 + F(I-3)**3 \]
      where I, II, III are invariants of the right Cauchy-Green deformation tensor which can
      be expressed in terms of principal stretch ratios. A, B, C, D, E, and F are Mooney-
      Rivlin material constants; and:
      \[ X = A/2. + B \]
      \[ Y = (A(5*nu -2.0) +B(11*nu -5.0))/(2.0(1.0 - 2*nu)) \]
      \[ nu = \text{Poisson’s ratio} \]
5. For orthotropic materials, the following assumptions are made:
   a. NUXY defaults to 0.3, NUXZ and NUYZ default to 0.0.
   b. EY and EZ default to EX.
   c. GXY, if not explicitly given, will be calculated from GXY = EX/(2(1+NUXY)) for
      isotropic materials, and from GXY = (EX.EY)/(EX + EY + 2.EY.NUXY) for
      orthotropic materials.
   d. GYZ and GXZ default to GXY.

**Material Curve Submenu**

Material curves are defined by a series of strain-stress values. This submenu contains commands to **Define, Activate, Plot, List, and Delete** material curves.
Define Material Curve

Define a material property set by defining a stress-strain curve. Values of strain and stress may be typed in or may be read from a file. Up to 20 curves may be defined with a maximum of 200 strain-stress data points for each curve.

**Input Description:**
- Material property set number.
- Curve type.
  - 0 Elastic
  - 1 Plastic
    (default is 0)
- Input flag.
  - 0 Read from a file.
  - 1 Type in values.
    (default is 1)
- File name (only if input flag = 2).
- Reference temperature (currently not used).
  (default is 0.0)

*The following entries are not prompted if data is to be read from a file:*

- Index of starting point in curve definition.
  (default is highest defined +1)
- Strain(i).
  (default is ith strain value)
- Stress(i).
  (default is corresponding ith stress value)
Note:

1. The file must be an ASCII file formatted in one of two ways:
   a. The first row lists the number of data points to be read, 0 (for the format type), and the increment value of the independent variable. The dependent variable values are listed one value per row from second row onwards. The independent variable will start from zero.
   b. The first row lists the number of data points to be read and 1 (for the format type). The independent and dependent variables values are then listed one pair per row from the second column onwards.

Activate Material Curve

Activate a material curve (similar to activating a material property set).

<table>
<thead>
<tr>
<th>Activate Material Curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input Description:</td>
</tr>
<tr>
<td>- Enter a material property set number.</td>
</tr>
</tbody>
</table>

Plot Material Curve

Plot the active stress-strain curve.

<table>
<thead>
<tr>
<th>Plot Material Curve</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input Description:</td>
</tr>
<tr>
<td>- Graph number.</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
<tr>
<td>- Graph color.</td>
</tr>
<tr>
<td>(default is 16 (white))</td>
</tr>
<tr>
<td>- Line style.</td>
</tr>
<tr>
<td>= 0 Solid line.</td>
</tr>
<tr>
<td>= 1 Dotted line.</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
<tr>
<td>- Symbol style.</td>
</tr>
<tr>
<td>= 0 No symbol</td>
</tr>
<tr>
<td>= 1 Circle</td>
</tr>
<tr>
<td>= 2 Filled circle</td>
</tr>
<tr>
<td>= 3 Square</td>
</tr>
<tr>
<td>= 4 Filled square</td>
</tr>
<tr>
<td>= 5 Upward triangle</td>
</tr>
<tr>
<td>= 6 Filled upward triangle</td>
</tr>
<tr>
<td>= 7 Downward triangle</td>
</tr>
<tr>
<td>= 8 Filled downward triangle</td>
</tr>
</tbody>
</table>
= 9 Star
= 10 Filled star
(default is 1)

List Material Curve

List all defined stress-strain material curves.

Delete Material Curve

Delete a pattern of stress-strain material curves.

Input Description:
- Enter first curve number.
- Enter last curve number.
- Increment.
  (default is 1)

Real Constants Submenu

This submenu contains commands for defining, listing and deleting real constant sets. The defined set becomes the active real constant set. Real constants define geometric (e.g., thickness of shells) and other element properties.

<table>
<thead>
<tr>
<th>REAL CONSTANTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define Beam Section</td>
</tr>
<tr>
<td>Define RC Set</td>
</tr>
<tr>
<td>List RC Sets</td>
</tr>
<tr>
<td>List Beam Section</td>
</tr>
<tr>
<td>Delete RC Sets</td>
</tr>
</tbody>
</table>

Figure 5.17  Real Constants Submenu

Define Beam Section

This submenu helps the user in calculating section properties for beam elements.
Choose Section Type, and respond to prompts. The program uses the section type and dimensions to calculate real constants for the cross-section. The active element group must be BEAM3D. The Cross-sectional area, moments of inertia, center of gravity, and the shear center are automatically calculated by the program. It should be noted that the command must be used to define the real constant set for non-rectangular beam elements with material nonlinearity, since the general Define RC Set assumes a rectangular cross section in this case. For beam elements with geometric nonlinearity (large displacement), the user has the option to use the Define RC Set, or Define Beam Section.

**Define Beam Section**

**Input Description:**
- Associated element group.
  (default is active group)
- Label for the real constant set to be defined.
- Number of points to define the user-defined section. Prompted only if "0" is selected for "shape".
- Starting location for storing section parameters.
- Number of beam section constants to be entered.
- Geometric dimensions. The user is prompted for the required constants depending on the selected section type. The description of the constants for various sections is given below.
- Non-geometric real constants. Refer to the linear and nonlinear BEAM2D and BEAM3D elements in the Element Library chapter for detailed description of the definition of non-geometric real constants for various sections. Note that the command prompts for all the properties that are not automatically calculated. Gaps in numbering the r(..) may exist.
since the command may calculate non-consecutive real constants. For example the command calculates values corresponding to real constants 1 through 5, and 8 for BEAM3D elements, but the values corresponding to real constants 6, 7, 9, etc., should be entered.

The sections available are shown in the figure below.

1. Rectangular cross-section:
   a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
      c(10) Height of the beam (H).
      c(2) Width of the beam (B).
   b. Symmetric tapered BEAM3D elements:
      c(1) Height of the beam at node-1 (H1).
      c(2) Width of the beam at node-1 (B1).
      c(3) Height of the beam at node-2 (H2).
      c(4) Width of the beam at node-2 (B2).

2. Circular cross-section:
   a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
      c(1) Radius of the beam (R).
   b. Symmetric tapered BEAM3D elements:
      c(1) Radius of the beam at node-1 (R1).
      c(2) Radius of the beam at node-2 (R2).

3. Pipe cross-section:
   a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
      c(1) Outside diameter of the beam (D).
      c(2) Thickness of the beam (T).
   b. Symmetric tapered BEAM3D elements:
      c(1) Outside diameter of the beam at node-1 (D1).
      c(2) Thickness of the beam at node-1 (T1).
      c(3) Outside diameter of the beam at node-2 (D2).
      c(4) Thickness of the beam at node-2 (T2).

4. Box cross-section:
   a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
      c(1) Height of the beam (H).
      c(2) Width of the beam (B).
      c(3) Thickness associated with height (TB).
      c(4) Thickness associated with width (TH).
   b. Symmetric tapered BEAM3D elements:
      c(1) Height of the beam at node-1 (H1).
      c(2) Width of the beam at node-1 (B1).
      c(3) Thickness of the web at node-1 (TB1).
      c(4) Thickness of the flange at node-1 (TH1).
      c(5) Height of the beam at node-2 (H2).
      c(6) Width of the beam at node-2 (B2).
      c(7) Thickness of the web at node-2 (TB2).
      c(8) Thickness of the flange node-2 (TH2).

5. I-section:
   a. Symmetric and unsymmetrical BEAM2D and BEAM3D elements:
      c(1) Height of the beam (H).
      c(2) Width of the beam (B).
      c(3) Thickness of the flange (TH).
c(4) Thickness of the web (TB).

b. Symmetric tapered BEAM3D:
  c(1) Height of the beam at node-1 (H1).
  c(2) Width of the beam at node-1 (B1).
  c(3) Thickness of the flange at node-1 (TH1).
  c(4) Thickness of the web at node-1 (TB1).
  c(5) Height of the beam at node-2 (H2).
  c(6) Width of the beam at node-2 (B2).
  c(7) Thickness of the flange at node-2 (TH2).
  c(8) Thickness of the web at node-2 (TB2).

6. Trapezoidal (unsymmetrical BEAM3D):
  c(1) Height of the beam (H).
  c(2) Bottom width of the beam (B1).
  c(3) Top width of the beam (B2).

7. Open channel (unsymmetrical BEAM3D):
  c(1) Height of the beam (H).
  c(2) Width of the beam (B).
  c(3) Thickness of the flange (TH).
  c(4) Thickness of the web (TB).

8. Z-section (unsymmetrical BEAM3D):
  c(1) Height of the beam (H).
  c(2) Width of the beam (B).
  c(3) Thickness of the flange (TH).
  c(4) Thickness of the web (TB).

9. T-section (unsymmetrical BEAM3D):
  c(1) Height of the beam (H).
  c(2) Width of the beam (B).
  c(3) Thickness of the flange (TH).
  c(4) Thickness of the web (TB).

10. L-section (unsymmetrical BEAM3D):
    c(1) Height of the beam (H).
    c(2) Width of the beam (B).
    c(3) Thickness associated with height (TB).
    c(4) Thickness associated with (TH).

The beam section constants shown above are required to define the geometry of the cross-
section. Beside the geometric constants, some other real constants are also required to fully
define the element. The following is a list of the required non-geometric constants:

1. BEAM2D elements:
   r(4) End-release code (node-1).
   r(5) End-release code (node-2).
   r(6) Shear factor in the element y-axis.
   r(7) Temperature difference in the element y-axis.

2. Symmetric BEAM3D elements:
   r(6) End-release code (node-1).
   r(7) End-release code (node-2).
   r(9) Shear factor in the element y-axis.
   r(10) Shear factor in the element z-axis.
   r(11) Temperature difference in the element y-axis.
   r(12) Temperature difference in the element z-axis.
3. Unsymmetrical BEAM3D elements:
   \( r(6) \) End-release code (node-1).
   \( r(7) \) End-release code (node-2).
   \( r(9) \) Shear factor in the element y-axis.
   \( r(10) \) Shear factor in the element z-axis.
   \( r(11) \) Temperature difference in the element y-axis.
   \( r(12) \) Temperature difference in the element z-axis.
   \( r(15) \) x-distance of section centroid relative to nodal point location at node-1.
   \( r(16) \) x-distance of section centroid relative to nodal point location at node-2.
   \( r(17) \) y-distance of section centroid relative to nodal point location at node-1.*
   \( r(18) \) y-distance of section centroid relative to nodal point location at node-2.*
   \( r(19) \) z-distance of section centroid relative to nodal point location at node-1.*
   \( r(20) \) z-distance of section centroid relative to nodal point location at node-2.*
   \( r(25) \) y-distance of the point where stresses are to be calculated.
   \( r(26) \) z-distance of the point where stresses are to be calculated.

* \( r(17) \) through \( r(20) \) are input for predefined sections (shape-1 to shape-10) only.

4. Symmetric tapered BEAM3D elements:
   \( r(11) \) End-release code (node-1).
   \( r(12) \) End-release code (node-2).
   \( r(15) \) Shear factor in the element y-axis.
   \( r(16) \) Shear factor in the element z-axis.
   \( r(17) \) Temperature difference in the element y-axis.
   \( r(18) \) Temperature difference in the element z-axis.
<table>
<thead>
<tr>
<th>Section Type</th>
<th>Identification Parameters</th>
</tr>
</thead>
<tbody>
<tr>
<td>Solid Rectangular Section</td>
<td></td>
</tr>
<tr>
<td>Solid Circular Section</td>
<td></td>
</tr>
<tr>
<td>Circular Section</td>
<td></td>
</tr>
<tr>
<td>Rectangular Box Section</td>
<td></td>
</tr>
<tr>
<td>Symmetric I Section</td>
<td></td>
</tr>
<tr>
<td>Trapezoidal Solid Section</td>
<td>H&gt;B1&gt;B2</td>
</tr>
<tr>
<td>Open Channel Section</td>
<td></td>
</tr>
<tr>
<td>Z Section</td>
<td></td>
</tr>
<tr>
<td>T Section</td>
<td></td>
</tr>
<tr>
<td>L Section</td>
<td></td>
</tr>
</tbody>
</table>

*Figure 5.19 Identification Parameters for the Built-in BEAM3D Section Library*
Define RC Set

This command creates a real constant set for an existing element group. The command may be used to modify the value(s) in one or more locations by specifying the first location number and the number of values to be changed.

**Define RC Set**

**Input Description:**
- Number of the set to be created.  
  (default is highest set defined + 1)
- Number of associated element group.  
  (default is active set)
- Start location for this entry.  
  (default is highest location defined + 1)
- Number or real constants to be input.  
  (default is dependent on type of element)
- Input ith constant prompted based on associated element group.

List RC Sets

Lists all real constant sets defined.

List Beam Section

Lists real constant sets defined by Define Beam Section.

Delete RC Sets

Deletes a pattern of real constant sets previously defined.

**Delete RC Sets**

**Input Description:**
- First set to be deleted.
- Final set to be deleted.  
  (default is first set)
- Increment in pattern.  
  (default is 1)

**Activate Submenu**

A submenu to activate an element group, material set, real constant set, coordinate system, load case, or element coordinate system.

The usages of these options are explained below:

a. Element Group, Material Set, Real Constant Set, and Element Coordinate System:
   These are called element attributes and together with the nodes, they completely define the element in COSMOS/M. Whenever an element is generated in the interface, it assumes the active attributes. Use this menu to activate the appropriate attributes prior to meshing.

The Element coordinate system is important for two reasons: It defines the coordinate system to be used in stress calculations if the local system is selected in the Element group options, and it determines the directions for material properties particularly useful in cases of non-isotropic materials.

b. Coordinate system:
   Some loads and boundary conditions get associated with the active coordinate system. If you want to fix a node in the radial direction of cylindrical coordinate system 1, activate coordinate system 1 using this menu and then fix the node in the X direction. Refer to the chapter, *Exploring ENGINEER*, for details.

c. Load Case:
   For linear static analysis, you may solve for multiple load cases. To construct a load case, use this menu to activate the desired load case number. Whenever a load is defined, it will be associated with the active load case. Refer to the chapter, *Exploring ENGINEER*, for details.

<table>
<thead>
<tr>
<th>ACTIVATE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Element Group</td>
</tr>
<tr>
<td>Material Set</td>
</tr>
<tr>
<td>Real Constant</td>
</tr>
<tr>
<td>Coord Sys</td>
</tr>
<tr>
<td>Load Case</td>
</tr>
<tr>
<td>E Coord Sys</td>
</tr>
<tr>
<td><strong>Quit</strong></td>
</tr>
</tbody>
</table>

![Figure 5.20  Activate Submenu](image-url)
Option & Description:

Element Group
   Element group.

Material Set
   Material property set.

Real Constant
   Real constant set.

Coord Sys
   Coordinate system.

Load Case
   Load case.

E Coord Sys
   Element coordinate system.

Once you select an option, you will be prompted to specify the label for the selected item.

Input Description:
- Label of the set to be activated.
  For Element group: 1 to 20
  For Material set: 1 to 90
  For Real constant: 1 to 5000
  For coordinate system:
    = 0    Global Cartesian
    = 1    Global cylindrical
    = 2    Global spherical
    = 3 - 500    Local coordinate system
  For Element coordinate system:
    = -1    Local element coordinate system
    = 0    Local element coordinate system 0
    = N    Local coordinate system N

Element Prop Change Submenu

A submenu to change associated element attributes like element group, material property, real constants, or element coordinate system.
Figure 5.21 Element Prop Change Submenu

Element Prop Change

Option & Description:

- **Elem Group**
  - Change associated element group.

- **Material Prop**
  - Change associated material property set.

- **Real Constants**
  - Change associated real constant set.

- **Elem Coord Sys**
  - Change associated element coordinate system.

Input Description:

- Label of the set to be assigned.

- Color of elements.
Mesh Curve

Meshes a curve by generating 1-D elements like SPRING, TRUSS3D, and BEAM3D. BEAM3D elements must be defined by 3 nodes to define the orientation of the cross-section. An appropriate element group must be active before meshing.
Mesh Curve

**Input Description:**
- Pick curve to be meshed.
- Number of nodes per element. Use 2 for SPRING and TRUSS3D elements and 3 for BEAM3D elements.
- Number of elements on curve.
- Pick keypoint to define principal axis. (not prompted if number of nodes is 2)

**Mesh Region Submenu**

Meshes a single region, a pattern of regions, or all regions by generating triangular elements. An appropriate element group must be activated before meshing. Choose **Single**, **Pattern**, or **All**. The generated elements are associated with the active element group which must be SHELL3, SHELL3T or TRIANG. The meshing process can be aborted using the "Escape" key.

![Mesh Region Submenu](image)

**Figure 5.23  Mesh Region Submenu**

**Mesh Region**

**Option & Description:**

- **Single**
  - Mesh a single region.

- **Pattern**
  - Mesh a pattern of regions.

- **All**
  - Mesh all regions.

- **Triang**
  - Generate automatic triangular mesh.
Quad
Generate automatic quadrilateral mesh.

Input Description:
- Beginning region in the pattern if All or Single is selected.
- Ending region in the pattern. (prompted only if Pattern is selected)
  (default is beginning region)
- Increment between regions in the pattern. (prompted only if Pattern is selected)
  (default is 1)
- Element order.
  = 0 Low
  = 1 High
  (default is 0)
- Push flag.
  = 0 No push
  = 1 Push
  (default is 0)
- Number of smoothing iterations. Use '0' for non-flat regions.
  (default is 0)
- Meshing method.
  = 0 Sweeping advancing front technique. (meshing advances from the boundaries to the inside)
  = 1 Hierarchical advancing front technique. (parallel layers in one direction)

Notes:
1. The smoothing iterations option is used for the fine adjustment of the generated nodes so that every node surrounded by a group of elements, is approximately, at their center. Usually, no more than 5 iterations are needed.
2. The average element size will be as specified on the contour level or mesh density control commands.
3. Problems could happen in creating the mesh if non-realistic requirements are specified. In such cases change the requirements or divide the region into 2 or more regions.
4. If the hierarchical technique fails, the user may try the sweeping technique.

Mesh Polyhedron Submenu
Meshes a polyhedron by generating 3- or 6-node shell elements. The average element size used is that defined by the Polyhedron Define command or by commands in the Mesh Density submenu. An appropriate element group must be active before meshing.
Mesh Polyhedron

Input Description:
- Pick polyhedron.
- Element order.
  
  \[
  \begin{array}{c|c}
  \text{Value} & \text{Order} \\
  0 & \text{Low} \\
  1 & \text{High} \\
  \end{array}
  \]

(default is 0)

Mesh Part Submenu

Creates a finite element mesh of 4- or 10-node tetrahedral elements. Upon selecting this command the user is presented with the following menu:

Quadratic Mesh Part

Selecting this option results in meshing the part with 10-node tetrahedral elements. The active element group should be TETRA10.

Input Description:
- Pick the part to be meshed.
Linear Mesh Part

Selecting this option results in meshing the part with 4-node tetrahedral elements. The active element group should be TETRA4 or TETRA4R.

**Input Description:**
- Pick the part to be meshed.

Quit

Returns to Meshing menu.

Mesh Density Submenu

This submenu provides commands for redefining mesh densities for curves, regions and polyhedra. The commands are as follows:

<table>
<thead>
<tr>
<th>MESH DENSITY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curve</td>
</tr>
<tr>
<td>Contour</td>
</tr>
<tr>
<td>Region</td>
</tr>
<tr>
<td>Polyhedron</td>
</tr>
<tr>
<td>Pick</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

*Figure 5.26  Mesh Density Submenu*

Curve Mesh Density

This command allows the user to specify a new average element size for a curve to be used in subsequent meshing. Mesh density is also changed for curves lying within the specified tolerance.
**Input Description:**
- Pick the curve you wish to use.
- Input new average element size.
- Specify tolerance to be used for locating coincident curves.  
  (default is 0.0001)

**Contour Mesh Density**

Defines the average element size for all curves associated with the contours in the specified pattern. The purpose is to provide local mesh control for meshing regions, polyhedra, and parts.

**Input Description:**
- Pick the contour you wish to use.
- Average element size to be used in subsequent meshing.
- Tolerance. Used to modify mesh specification for neighboring curves of contours. The user may need to use a higher tolerance for proper 2-D and 3-D meshing, specially if geometry is imported from a CAD program.  
  (default is 0.0001)
- Meshing redefinition flag. Prompted only if the contour being defined shares curves with other existing contours for mesh compatibility.
  
  - 0 For curves common to other contours, use same element size or number of elements as previously specified.
  - 1 For curves common to other contours, change the element size or number of elements to comply with the new element size.
  - 2 For curves common to other contours, compare old and new mesh specifications and select the option that specifies a finer mesh (more elements).
  - 3 For curves common to other contours, compare old and new mesh specifications and select the option that specifies a coarser mesh (less elements).  
  (default is 1)

**Region Mesh Density**

This command allows the user to specify a new average element size for a region to be used in subsequent meshing. Mesh density is also changed for curves lying within the specified tolerance from the curves of the specified region.

**Input Description:**
- Pick the region you wish to use.
- Input new average element size.
- Specify tolerance to be used for locating coincident curves. (default is 0.0001)

**Polyhedron Mesh Density**

This command allows the user to specify a new average element size for a polyhedron to be used in subsequent meshing.

<table>
<thead>
<tr>
<th>Polyhedron Mesh Density</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Description:</strong></td>
</tr>
<tr>
<td>- Pick the polyhedron you wish to use.</td>
</tr>
<tr>
<td>- Average element size.</td>
</tr>
</tbody>
</table>

**Mesh Options Submenu**

Specifies option of geometry type from CAD packages. The set option is used during meshing and should be compatible with that of the particular imported CAD geometry.

<table>
<thead>
<tr>
<th>MESH OPTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>XYZ</td>
</tr>
<tr>
<td>UV</td>
</tr>
<tr>
<td>Iterate</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.27  Mesh Options Submenu

**Mesh Options**

Option & Description:

- **XYZ**
  - Global XYZ geometry.

- **UV**
  - Parametric UV.

- **Iterate**
  - Iterate boundary nodes for placement.
Mesh Tolerance Submenu

Defines tolerances to be used in meshing and other purposes. Point tolerance is used to specify the minimum difference in at least one of the coordinates of two keypoints. If the difference is smaller than the tolerance for any new keypoints defined after the tolerance has been specified, the keypoints will be merged. Contour tolerance defines the largest allowable gap between ends of curves making a contour. Polyhedra tolerance specifies the tolerance for closing polyhedra and is used for meshing purposes. Increasing polyhedra tolerance may be useful when meshing fails with the given message "probable free edges".

<table>
<thead>
<tr>
<th>MESH TOLERANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Point</td>
</tr>
<tr>
<td>Contour</td>
</tr>
<tr>
<td>Polyhedron</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.28  Mesh Tolerance Submenu

Mesh Tolerance

Option & Description:

**Point**
- Point tolerance.

**Contour**
- Contour gap tolerance.

**Polyhedron**
- Polyhedron gap tolerance.

Input Description:
- Input tolerance.

Node Submenu

Opens the menu of node commands. Choose the operation type, All or Pattern and Accept to perform the operation or Quit to abandon it. The All or Pattern options are ignored by Delete Pick and Identify.
Figure 5.29 Node Submenu

Define Nodes

Creates a new node or modifies an existing one using existing nodes.

**Input Description:**

- Label of node to be created.
  (default is highest defined + 1)

- X coordinate value.
  (default is 0)

- Y coordinate value.
  (default is 0)

- Z coordinate value.
  (default is 0)

- Associated keypoint.
  (default is 0)

- Associated curve.
  (default is 0)

- Associated surface.
  (default is 0)

- Associated volume.
  (default is 0)

- Associated contour.
  (default is 0)
- Associated region.  
  (default is 0)

**Plot Nodes**

Plots all or a pattern of nodes. If a selection list exists, then only nodes in the selection list will be plotted. Pattern information is not prompted if All is selected.

**Input Description:**
- Beginning node in the pattern.
- Ending node in the pattern.  
  (default is beginning node)  
- Increment in the pattern.  
  (default is 1)

**List Nodes**

Lists all or a pattern of nodes. If a selection list exists, then only nodes in the selection list will be listed. Pattern information is not prompted if All is selected.

**Input Description:**
- Beginning node in the pattern.
- Ending node in the pattern.  
  (default is beginning node)  
- Increment in the pattern.  
  (default is 1)

**Delete Nodes**

Deletes all or the selected pattern of nodes. If a selection list exists, then only nodes in the selection list will be deleted. Pattern information is not prompted if All is selected.

**Input Description:**
- First node in the pattern.
- Ending node in the pattern.  
  (default is first node)
- Increment in the pattern.
  (default is 1)

**Delete Pick Nodes**
Deletes nodes by picking one node at a time. The **All/Pattern** option is ignored.

**Identify Nodes**
Highlights the selected node and displays its coordinates. The nodes must have been plotted on the screen. The **All/Pattern** option is ignored.

**Merge Nodes**
This command causes coincident nodes or those lying within the specified tolerance to be merged. Pattern information is not prompted if **All** is selected.

**Input Description:**
The following entries are not prompted if **All** is selected.

- Enter beginning node of the pattern.
  (default is 1)

- Enter ending node of the pattern.
  (default is max. node defined)

- Increment in the pattern.
  (default is 1)

- Tolerance to be used for merging.
  (default is 0.0001)

- Consider all nodes for merging with the pattern or merge only among the nodes in the pattern.
  
  = 0           All
= 1 Among
  (default is 0)

- Echo the merging nodes to the message window.
  = 0 Off
  = 1 On
  (default is 1)

- Which node to keep.
  = 0 Lower label.
  = 1 Higher label.
  (default is 0)

Show Merge
Displays the free edges of a model, or the nodes that will be merged using a given tolerance. The command does not merge any nodes but will show the nodes that will be merged if the Merge command is issued with the same tolerance. The command is very useful in showing disconnected parts of a model.

Input Description:
- Echo flag.
  = 0 Off
  = 1 On

Accept
Accept command options.

Quit
Quit and return to the Meshing menu.

Element Submenu
Opens the menu for element commands. Choose the operation type, All or Pattern and Accept to perform the operation or Quit to abandon it. The All or Pattern options are ignored by Delete Pick and Identify.
Define Element

Creates a new element or modifies an existing one using existing nodes. The element is associated with the active element group, material property, and real constant sets. To define the correct connectivity for the element, the command asks for the associated geometric entity. For example, the program through this entry can differentiate between a 3-node beam element and a 3-node shell element. The shell element must be associated with RG (region), or SF (surface), and the beam element should be associated with CR (curve). In general, 1-node elements like MASS should be associated with PT (points). Uniaxial (1-D) elements like TRUSS3D, BEAM3D, and SPRING, should be associated with CR. Area (2-D) elements like SHELL3, SHELL4, and PLANE2D, should be associated with RG or SF. Volume (3-D) elements like TETRA4R, and TETRA10 should be associated with VL (volume). If a label is given to the associated element geometry then this association can be used in applying loads and boundary conditions, otherwise a '0' should be entered.

**Input Description:**

- Label of element to be created. (default is highest defined + 1)
- Type of associated entity. Use:
  - PT for 1-node elements.
  - CR or CT for 1-D elements.
  - SF or RG for 2-D elements.
  - and VL for 3-D elements. (default is SF)
- Label of the associated entity. Use zero if the element is not to be associated with a particular label. (default is 0)
- Number of nodes in the element.

- Pick the nodes, refer to Chapter 4 for element numbering.
  (default is 4)

- Label of geometric entity associated with face i of the element (i = 1,2,....,6). Use 0 unless
  you intend to use association in applying Loads/BConds, or selections.
  (default is 0)

Note:

1. The face association entries represent geometric association of element faces. The type of
   the geometric entity could be a curve, a surface or a region depending on the element
   group and the face number. For example faces 5 and 6 of shell elements are associated
   with a surface or a region while the other faces are associated with curves. An input of 0
   means no geometric association for the faces of the element to be created. Refer to
   Chapter 4 for details on element faces.

Plot Element

Plots all or a pattern of elements. If a selection list exists, then only elements in the selection
list will be plotted. Pattern information is not prompted if All is selected.

<table>
<thead>
<tr>
<th>Plot Element</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Description:</strong></td>
</tr>
<tr>
<td>- Beginning element in the pattern.</td>
</tr>
<tr>
<td>- Ending element in the pattern.</td>
</tr>
<tr>
<td>(default is beginning element)</td>
</tr>
<tr>
<td>- Increment in the pattern.</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
</tbody>
</table>

List Element

Lists all elements in the element selection list. If a selection list exists, then only elements in
the selection list will be listed.

<table>
<thead>
<tr>
<th>List Element</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Input Description:</strong></td>
</tr>
<tr>
<td>- Beginning element in the pattern.</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
<tr>
<td>- Ending element in the pattern.</td>
</tr>
<tr>
<td>(default is highest element)</td>
</tr>
<tr>
<td>- Increment in the pattern.</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
</tbody>
</table>
**Delete Element**

Deletes all or a pattern of elements. If a selection list exists, then only elements in the selection list will be deleted. Pattern information is not prompted if All is selected.

**Delete Element**

**Input Description:**
- Beginning element in the pattern.
- Ending element in the pattern.
  (default is beginning element)
- Increment in the pattern.
  (default is 1)

**Delete Pick Element**

 Deletes elements by picking one element at a time. The All/Pattern option is ignored.

**Delete Pick Element**

**Identify Element**

Highlights an element plotted on the screen and displays its label. The All/Pattern option is ignored.

**Identify Element**

**Input Description:**
- Pick the element required.

**Element Order**

This command changes all the elements from linear to quadratic or from quadratic to linear by creating or deleting midside nodes. Selecting the higher order, for example, changes 10-node tetrahedral elements to 4-node elements.

**Element Order**

**Input Description:**
- Make higher order or lower order.
  
<table>
<thead>
<tr>
<th>Value</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>0</td>
<td>Low</td>
</tr>
<tr>
<td>1</td>
<td>High</td>
</tr>
</tbody>
</table>

  (default is 1)
- Push new nodes to the geometry boundary.
  = 0 No push
  = 1 Push
  (default is 0)

**Element Check**

Checks the elements specified in the pattern. Aspect ratios and connectivity are checked. A message is issued if the aspect ratio of an element exceeds the specified ratio. The command automatically deletes degenerate elements.

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Beginning element in the pattern.</td>
</tr>
<tr>
<td>- Ending element in the pattern.</td>
</tr>
</tbody>
</table>
  (default is beginning element)
| - Increment in the pattern. |
  (default is 1)
| - Aspect ratio. |
  (default is 5.0)
| - Checks element node connectivity order. |
  = 1 Yes
  = 0 No
| - Element check option. |
  = 0 None
  = 1 Draw
  = 2 List
  = 3 Both
  (default is 1)

**Element Refine**

Refines an existing mesh of 3-node area elements like TRIANG, SHELL3, or SHELL3T, or 4-node volume elements like TETR4 and TETRA4R. The elements on which refinement is to be performed must be placed in the element selection list (refer to Selection menu) prior to calling this command, otherwise no refinement will take place. Note that elements in the neighborhood of the elements in the selection list may also be refined for compatibility purposes.
Align Shell

Align shell elements with respect to the first element.

Accept

Accept command options.

Quit

Quit and return to Meshing menu.

Troubleshooting

Error Messages during Mesh Polyhedron

a. Convergence not achieved for Region#

   Note the region labels this message displays and mesh them individually using Mesh Region setting the method as Postup, using the same element size as for the Polyhedron and set the hierarchy check flag to 1. If the error persists use Region Reorient and try meshing it again. If this still produces the error use Mesh Density Region and specify a smaller element size. Alternate between these last two commands until the error during Mesh Region disappears. Delete all of the elements and repeat the command Mesh Polyhedron.

b. Probable free edges

   Use the command Curve Merge as described in the Geometry Menu Troubleshooting section and then mesh the associated regions individually setting the method as Postup with an element size the same as for the Polyhedron and set the hierarchy check flag to 1. If a message of "Convergence Not Achieved" is obtained, use Region Reorient for that region and remesh it. Delete all of the elements and repeat the command Mesh Polyhedron.

c. Merged facet on Region#

   With this error, plot the two regions stated. This message is generated when the angle between two adjacent regions is less than 1 degree. These regions should be modified in a CAD program or COSMOS/M CAD Interface. They usually occur at the common regions
when two parts are merged in an assembly. If the two regions are not overlapping or the angle between them is more than 1 degree this message can be ignored.

**Error in Generating Mesh during Mesh Part**

This message is followed by the number of facets left; use either **Mesh Density Region** (if you have used local refinement) or **Polyhedron Define**. In both cases use a smaller element size, between 85% - 95% of the current size is recommended. If the error persists repeat the element size reduction. If there is an abnormal program termination, delete this database and retrieve the one you saved prior to **Mesh Part**, reduce the element size as described above, save the database to a new name and repeat **Mesh Part**.
LOADS/BCONDS Menu

This menu contains commands to define, list, and delete loads and boundary conditions. Loads and boundary conditions are either applied to nodes (e.g. displacement), or element faces (e.g. pressure). If loads or boundary conditions are applied to curves or regions, then the program applies them to the associated nodes or element faces. Loads and boundary conditions cannot be applied to geometric entities prior to meshing. Some important load options are given in the Load Options submenu.

---

**DISPLACEMENT**
- Define
- Plot
- List
- Delete Select
- Delete All

**Nodes**
- Curves
- Regions

**TEMPERATURE**
- Define
- Plot
- List
- Delete Select
- Delete All

**Nodal Heat**
- Element Heat
- Pressure
- Convection
- Heat Flux
- Radiation
- Fluid Flow

**FORCE MOMENT**
- Define
- Plot
- List
- Delete Select
- Delete All

**Nodes**
- Curves
- Regions

**LOAD OPTIONS**
- See following figure

**HEAT FLUX**
- See following figure

**FLUID FLOW**
- See following figure

**LOADS/BCONDS**
- Displacement
- Force Moment
- Temperature
- Nodal Heat
- Element Heat
- Pressure
- Convection
- Heat Flux
- Radiation
- Fluid Flow
- Load Options

**ELEMENT HEAT**
- Define
- Plot
- List
- Delete Select
- Delete All

**Curves**
- Regions

**CONVECTION**
- See following figure

**RADIATION**
- See following figure

**PRESSURE**
- See following figure

**LOAD OPTIONS**
- Accept
- Quit

**NODAL HEAT**
- Define
- Plot
- List
- Delete Select
- Delete All

**Nodes**
- Curves
- Regions

---

Figure 5.31  Loads/BConds Menu
Figure 5.31 Loads/BConds Menu (Continued)
Figure 5.31  Loads/BConds Menu  (Continued)
Figure 5.31  Loads/BConds Menu  (continued)
The following table illustrates the type and use of various loads and boundary conditions.

<table>
<thead>
<tr>
<th>LOAD OPTIONS</th>
<th>LOAD OPTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Acceleration</td>
<td>Angular Velocity</td>
</tr>
<tr>
<td>Times</td>
<td>Reference Temp</td>
</tr>
<tr>
<td>Uniform Temp</td>
<td>Read Temp</td>
</tr>
<tr>
<td>Initial Conditions</td>
<td>Temperature</td>
</tr>
<tr>
<td>Displacement</td>
<td>Velocity</td>
</tr>
<tr>
<td>Acceleration</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.31 Loads/BConds Menu (Continued)
<table>
<thead>
<tr>
<th>Load/BConds</th>
<th>Applied to</th>
<th>Used in</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacements</td>
<td>nodes</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Force moment</td>
<td>nodes</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Temperature</td>
<td>nodes</td>
<td>Structural &amp; thermal analyses</td>
</tr>
<tr>
<td>Pressure</td>
<td>element faces</td>
<td>Structural analysis</td>
</tr>
<tr>
<td>Convection</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>Heat Flux</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
<tr>
<td>Radiation</td>
<td>element faces</td>
<td>Thermal analysis</td>
</tr>
</tbody>
</table>

Displacement Submenu

Selecting this option opens the Displacement submenu shown below:

![Displacement Submenu Diagram]

The user should select one item from the top group to select the desired operation. After one option is selected from each group of options, picking the Accept option executes the desired operation, and picking the Quit option quits the submenu without any action. The second group “Nodes, Curves, Regions” is only used with the Define operation. The fourth group "Pick, Selected" is not used with the Delete Select and Delete All operations.
Define Displacement

Defines displacement boundary conditions on nodes, nodes associated with curves, or nodes associated with regions. It should be noted that displacement and boundary conditions specified on curves or regions will only take place if the curves or regions were previously meshed. To specify displacement, highlight the Define option, and choose one option from each group. The options are as follows:

<table>
<thead>
<tr>
<th>Nodes</th>
<th>Apply the displacement boundary condition to nodes.</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curves</td>
<td>Apply the displacement condition to nodes associated with curves.</td>
</tr>
<tr>
<td>Regions</td>
<td>Apply the displacement condition to nodes associated with regions.</td>
</tr>
<tr>
<td>UX-restraint</td>
<td>Specify translation in the X direction of the active coordinate system.</td>
</tr>
<tr>
<td>UY-restraint</td>
<td>Specify translation in the Y direction of the active coordinate system.</td>
</tr>
<tr>
<td>UZ-restraint</td>
<td>Specify translation in the Z direction of the active coordinate system.</td>
</tr>
<tr>
<td>RX-restraint</td>
<td>Specify rotation about the X direction of the active coordinate system in radians.</td>
</tr>
<tr>
<td>RY-restraint</td>
<td>Specify rotation about the Y direction of the active coordinate system in radians.</td>
</tr>
<tr>
<td>RZ-restraint</td>
<td>Specify rotation about the Z direction of the active coordinate system in radians.</td>
</tr>
<tr>
<td>Pin-Joint</td>
<td>Fix all translations (set UX, UY, and UZ to zero).</td>
</tr>
<tr>
<td>No-Rotation</td>
<td>Fix all rotations (set RX, RY, and RZ to zero).</td>
</tr>
<tr>
<td>Rigid-Joint</td>
<td>Fix all translations and rotations.</td>
</tr>
<tr>
<td>Pick</td>
<td>Pick the entity to which the displacement boundary conditions should be applied by the mouse.</td>
</tr>
<tr>
<td>Selected</td>
<td>Apply the displacement boundary conditions to all entities of the type specified above (nodes, curves, or regions) that are in the selected list. Refer to the Selection menu for more information on selection lists.</td>
</tr>
<tr>
<td>Accept</td>
<td>Accept command options.</td>
</tr>
<tr>
<td>Quit</td>
<td>Quit the command.</td>
</tr>
</tbody>
</table>

After selecting Accept, you will be prompted to select the members of the entity type selected above (nodes, curves, or regions), if the Pick option is highlighted. You will also be prompted to enter the value of displacement which may be zero or any other value. Note that you will not be prompted to pick the entity if the Selected option is highlighted, and you will not be prompted for the value if Pin-Joint, No-Rotation, or Rigid-Joint is highlighted.

The command may be repeated as needed to specify displacement boundary conditions.

---

**Define Displacement**

**Input Description:**
- Pick node, curve, or region if the Pick option is highlighted.
- Value of displacement to be applied. (default is 0.0)
Example: If Define, Regions, No Rotations, Selected, and Accept are selected, then all nodes associated with regions in the active region selection set will be assigned a value of zero for RX, RY, and RZ. If Pick is highlighted instead of Selected, the region should be picked by the mouse.

Plot Displacement

Plots the specified displacement constraints for nodes in the selection list. If the user has not defined a node selection list, then the requested constraints are plotted for all nodes. The "Node, Curve, Region", and the "Pick, Selected" options are not used for plotting. One arrow indicates translation, two arrows indicate rotation and three arrows indicate both.

List Displacement

Lists all displacement constraints for all nodes if no selection list exists for nodes. If a selection list exists for nodes, then only the selected nodes are listed. No options are used for listing, all 6 degrees of freedom are always listed.

Delete Select Displacement

Deletes the selected displacement boundary conditions on the specified entity type. Use the Pick option to delete boundary conditions for a specific entity.

Input Description:
- Pick the entity requested (node, curve or region) or use the select option to operate on all entities in the selection list.

Delete All Displacement

Deletes all displacement boundary conditions for the active set of nodes irrespective of other options.
Force Moment Submenu

This menu system is used in relation to point forces or bending moments at nodes. Its use is the same as described for displacements. Forces and moments are applied directly to nodes, or through node association with curves and regions.

<table>
<thead>
<tr>
<th>FORCE MOMENT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete Select</td>
</tr>
<tr>
<td>Delete All</td>
</tr>
<tr>
<td>Nodes</td>
</tr>
<tr>
<td>Curves</td>
</tr>
<tr>
<td>Regions</td>
</tr>
<tr>
<td>Fx - Force</td>
</tr>
<tr>
<td>Fy - Force</td>
</tr>
<tr>
<td>Fz - Force</td>
</tr>
<tr>
<td>Mx - Moment</td>
</tr>
<tr>
<td>My - Moment</td>
</tr>
<tr>
<td>Mz - Moment</td>
</tr>
<tr>
<td>Pick</td>
</tr>
<tr>
<td>Selected</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.33  Force Moment Submenu

Where:

- Fx - Force = force in the x-direction
- Fy - Force = force in the y-direction
- Fz - Force = force in the z-direction
- Mx - Moment = moment about the x-axis
- My - Moment = moment about the y-axis
- Mz - Moment = moment about the z-axis

Temperature Submenu

This menu is used to fix temperatures at nodes in the model. It can be used for both thermal and structural analyses. Its use is the same as that described for displacements. Temperatures are applied directly to nodes or through node association with curves and regions.
Nodal Heat Submenu

This submenu deals with defining, plotting, listing, and deleting of concentrated heat generation at nodes. Select the operation type, entity type, **Pick** or **Selected** and **Accept**. Choosing **Quit** quits the submenu. The entity type may be **Nodes, Curves, or Regions**. The specified value will be used for each node. If **Curves** or **Regions** are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used by HSTAR, the heat transfer analysis module.
Element Heat Submenu

This submenu deals with defining, plotting, listing, and deleting of element heat generation. Select the operation type, entity type, Pick or Selected and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value is the rate of heat generation per unit volume. If Curves or Regions are selected, the specified value will be applied to every element associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used by HSTAR, the heat transfer analysis module.

![Element Heat Submenu](image)

Pressure Submenu

This menu is used in relation to distributed pressures in the model for a structural analysis. Pressure is specified for element faces or edges associated with the selected curves or regions and is normal to the edge or face. Note that pressures in relation to curves cannot be used with Tetrahedral elements. The use of this submenu is the same as described for displacements.
Convection Submenu

This menu is used in relation to the modeling of convection boundary conditions for a thermal analysis. To define convection the user will be asked to input a value for the convection coefficient and the ambient temperature. Note that convection in relation to a curve cannot be used with Tetrahedral elements. The use of this menu is the same as described for displacements.
Heat Flux Submenu

Applies a value of heat flux to the element faces for a subsequent thermal analysis. The use of this menu is the same as that described for displacements. Note that heat flux in relation to curves cannot be used for Tetrahedral elements.

<table>
<thead>
<tr>
<th>HEAT FLUX</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete Select</td>
</tr>
<tr>
<td>Delete All</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Curves</th>
</tr>
</thead>
<tbody>
<tr>
<td>Regions</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pick</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Accept</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.39  Heat Flux Submenu

Radiation Submenu

Applies radiation boundary conditions to element faces in the model for a subsequent thermal analysis. The use of this menu is the same as described for displacements. To define radiation, the user will be asked to input values for emissivity, view factor and ambient temperature. Note that radiation in relation to curves cannot be used with Tetrahedral elements.

<table>
<thead>
<tr>
<th>RADIATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete Select</td>
</tr>
<tr>
<td>Delete All</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Curves</th>
</tr>
</thead>
<tbody>
<tr>
<td>Regions</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Pick</th>
</tr>
</thead>
<tbody>
<tr>
<td>Selected</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Accept</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.40  Radiation Submenu
Fluid Flow Submenu

This submenu contains the commands to Define, Plot, List, and Delete fluid flow loadings and boundary conditions.

<table>
<thead>
<tr>
<th>FLUID FLOW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>TK Energy</td>
</tr>
<tr>
<td>Dissp Rate</td>
</tr>
<tr>
<td>Node Pressure</td>
</tr>
<tr>
<td>Density</td>
</tr>
<tr>
<td>Boundary EL</td>
</tr>
</tbody>
</table>

Figure 5.41 Fluid Flow Submenu

Velocity Submenu

This submenu deals with defining, plotting, listing, and deleting velocities at nodes. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value will be used for each node. If Curves or Regions are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used in fluid flow analysis.

<table>
<thead>
<tr>
<th>VELOCITY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete Select</td>
</tr>
<tr>
<td>Delete All</td>
</tr>
<tr>
<td>Nodes</td>
</tr>
<tr>
<td>Curves</td>
</tr>
<tr>
<td>Regions</td>
</tr>
<tr>
<td>VX</td>
</tr>
<tr>
<td>VY</td>
</tr>
<tr>
<td>VZ</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Pick</td>
</tr>
<tr>
<td>Selected</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.42 Velocity Submenu
TK Energy Submenu

This submenu contains the commands to Define, Plot, List, and Delete turbulence kinetic energy values at nodes. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value will be used for each node. If Curves or Regions are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used in fluid flow analysis.

Dissp Rate Submenu

This submenu contains the commands to Define, Plot, List, and Delete a dissipation rate at nodes. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value will be used for each node. If Curves or Regions are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used in fluid flow analysis.
Figure 5.44  Dissp Rate Submenu

Node Pressure Submenu

This submenu contains the commands to Define, Plot, List, and Delete pressure values at nodes. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value will be used for each node. If Curves or Regions are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used in fluid flow analysis.
Density Submenu

This submenu contains the commands to Define, Plot, List, and Delete density values at nodes. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Nodes, Curves, or Regions. The specified value will be used for each node. If Curves or Regions are selected, the specified value will be applied at every node associated with any of the specified curves or regions. The user will be prompted according to the selections made. This submenu is only used in fluid flow analysis.

**Figure 5.46  Density Submenu**

Boundary EL Submenu

This submenu contains the commands to Define, Plot, List, and Delete zero normal velocity on faces of elements. Select the operation type, entity type, Pick or Selected, and Accept. Choosing Quit quits the submenu. The entity type may be Curves, Regions, or Elements. Zero normal velocity will be applied on the specified face of specified elements or elements associated with any of the specified curves or regions. This submenu is only used in fluid flow analysis.
Load Options Submenu

This item gives access to the submenu containing additional loading commands.

Acceleration

Defines components of gravitational acceleration to be used in a static analysis of the model. The gravity option in the Static command in Analysis menu must be turned on to activate gravity loading. Material density and/or mass elements should also be defined.

Input Description:
- X-component of acceleration.
  (default is 0)
- Y-component of acceleration.  
  (default is 0)

- Z-component of acceleration.  
  (default is 0)

**Angular Velocity**

Defines angular velocities for use in a static analysis of the model. The centrifugal option in the **Static** command in the Analysis menu must be turned on to activate centrifugal loading. Material density and/or mass elements should also be defined.

**Angular Velocity**

**Input Description:**
- X-component of angular velocity.  
  (default is 0)

- Y-component of angular velocity.  
  (default is 0)

- Z-component of angular velocity.  
  (default is 0)

**Times**

Defines the start time, end times, and time step size for transient thermal analysis.

**Times**

**Input Description:**
- Starting time for the analysis.  
  (default is 0)

- Ending time for the analysis.  
  (default is starting time)

- Time increment.

**Reference Temp**

Defines the reference temperature for the model when performing a thermal stress analysis.
Input Description:
- Value of reference temperature.
  (default is 0)

Uniform Temp
Defines a uniform temperature to all nodes in the model as a starting condition for thermal analyses.

Input Description:
- Uniform temperature value.
  (default is 0)

Read Temp
Reads temperatures from a specified time step of a transient thermal analysis for use as thermal loads in a subsequent static analysis. To read temperatures from steady-state thermal analysis use the default time step number. The thermal loading flag in the Run command from the Analysis-Static menu must be turned on and the coefficient(s) of thermal expansion must be defined.

Input Description:
- Time step number.
  (default is 1)

Initial Conditions Submenu
Defines initial conditions at nodes for transient analyses. Choose the type of initial condition, the operation type, and Accept to execute the command, or Quit to abandon it. The valid types of initial conditions and related analysis types are given below.
Initial Conditions

Option & Description:

*Temperature*

Define initial temperatures for use with transient thermal analysis using HSTAR and FFE Thermal.

*Displacement*

Define initial displacements for use in ASTAR and NSTAR.

*Velocity*

Define initial velocities for use in ASTAR and NSTAR Dynamic.

*Acceleration*

Define initial acceleration for use in ASTAR and NSTAR.

Input Description:

- Beginning node in the pattern.
  (default is 1)

- Ending node in the pattern.
  (default is highest node defined)

- Increment between nodes in the pattern.
  (default is 1)

- Initial value in the X-direction or the temperature value.
  (default is 0.0)

*The next two arguments are not prompted if Temperature is selected.*

- Initial value in the Y-direction.
  (default is 0.0)

- Initial value in the Z-direction.
  (default is 0.0)
**Status**

This option opens the status table which shows the currently active sets and the plotting status of various geometric entities. The user can toggle the flags to change the settings and keep them for this model by closing the table using the Save box in the upper right-hand corner. If
the changes are not to be kept the user can exit by selecting the Quit box in the upper left hand corner. Note that some entities and some flags are not explicitly used in COSMOS/M CAD Interface, but are useful when working in the GEOSTAR mode.

The active sets are shown in the top bar of the table. A guide to the symbols follows.

```
<table>
<thead>
<tr>
<th>Status Table 1 Active Element: TETRA4R</th>
</tr>
</thead>
<tbody>
<tr>
<td>EG</td>
</tr>
<tr>
<td>1</td>
</tr>
</tbody>
</table>
```

**Figure 5.51 Status Table**

### Status

**Sets & Description:**

- **EG** - Element group
- **MP** - Material set
- **RC** - Real constant set
- **CS** - Coordinate system
- **LC** - Load case
- **TC** - Time curve
- **TP** - Temperature curve
- **MC** - Magnetic curve

*The entities considered in the table are:*

**Entities & Description:**

- **PT** - Keypoints
- **CR** - Curves
- **SF** - Surfaces
- **VL** - Volumes
- **CT** - Contours
- **RG** - Regions
The status information that can be controlled for each of the entities listed above is as follows:

**Option & Description:**

**PLOT**
Flag ON/OFF, if set ON, the entity will be plotted if a higher entity is requested for plotting. The entity will be colored according to the PCLR.

**LABL**
Flag ON/OFF, if set ON, will plot entity labels using the color set in LCLR.

**MAXM**
Maximum entity label in the model.

**KEEP**
Flag ON/OFF, used to specify whether lower order entities should be kept when higher entities are deleted.

**PSEL**
Flag to activate or deactivate a selection set.

**MARK**
Flag ON/OFF. When ON a marker is placed on the entity to show its orientation.

**DMSH**
Flag ON/OFF to activate or deactivate default mesh generation for the entity. (Not used in COSMOS/M CAD Interface.)

### Fill Color Submenu

Sets the fill color to be used for the elements when plotted with hidden lines removed.

<table>
<thead>
<tr>
<th>FILL COLOR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Bright</td>
</tr>
<tr>
<td>Dark</td>
</tr>
<tr>
<td>Blue</td>
</tr>
<tr>
<td>Green</td>
</tr>
<tr>
<td>Cyan</td>
</tr>
<tr>
<td>Red</td>
</tr>
<tr>
<td>Magenta</td>
</tr>
<tr>
<td>Yellow</td>
</tr>
<tr>
<td>Grey</td>
</tr>
</tbody>
</table>

Accept

Quit

*Figure 5.52  Fill Color Submenu*
Fill Color

Option & Description:

*Bright*
  Use bright color.

*Dark*
  Use dark color.

*Blue, Green, Cyan, Red, Magenta, Yellow, Grey*
  Choose a color.

*Accept*
  Accepts command options.

*Quit*
  Abandons the command and returns to the Viewing menu.

Boundary Opts Submenu

Opens a submenu of options for controlling element boundary plotting and evaluation.

<table>
<thead>
<tr>
<th>BOUNDARY OPTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>No Boundary</td>
</tr>
<tr>
<td>Mesh Boundary</td>
</tr>
<tr>
<td>Model Boundary</td>
</tr>
<tr>
<td>No Evaluation</td>
</tr>
<tr>
<td>Evaluate Face</td>
</tr>
<tr>
<td>Evaluate Edge</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.53  Boundary Opts Submenu

Boundary Opts

Option & Description:

*No Boundary*
  A subsequent element or results plot will be shown with no element boundaries.

*Mesh Boundary*
  A subsequent element or results plot will be shown with element boundaries.

*Model Boundary*
  A subsequent plot will only show boundaries of the model.
No Evaluation
   No evaluation is performed.

Evaluate Face
   Identifies exterior faces of the model and uses them in subsequent plots.

Evaluate Edge
   Identifies edges of the model.

Accept
   Activates the selected options.

Quit
   Returns to the Viewing menu without changing the previous options.

Hide Opts Submenu

Opens a submenu to provide options for hidden line removal.

<table>
<thead>
<tr>
<th>HIDE OPTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Depth Sort</td>
</tr>
<tr>
<td>Z-Buffer</td>
</tr>
<tr>
<td>Positive Normal</td>
</tr>
<tr>
<td>Negative Normal</td>
</tr>
<tr>
<td>Both</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.54  Hide Opts Submenu

Hide Opts

Option & Description:

Depth Sort
   This is the default and fast technique but it may have problems when angles between element faces are small.

Z-Buffer
   This method takes longer to calculate and display the image but does not suffer from the limitations of the depth sort technique.

Positive Normal
   Shows elements only on the positive side of the region.

Negative Normal
   Shows elements only on the negative side of the region.
Both
   Shows elements on both sides of the region.

Accept
   Activates the selected options.

Quit
   Returns to the Viewing menu without changing the previous options.

Hidden Submenu

This menu enables or disables hidden line removal for subsequent element plots.

<table>
<thead>
<tr>
<th>HIDDEN</th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>On</td>
<td></td>
</tr>
<tr>
<td>Off</td>
<td></td>
</tr>
<tr>
<td>Quit</td>
<td></td>
</tr>
</tbody>
</table>

Figure 5.55  Hidden Submenu

Hidden

Option & Description:

On
   Activates hidden line removal and redraws the screen using the algorithm set by the Hide
   Opt command.

Off
   De-activates hidden line removal and repaints the window.

Quit
   Closes submenu and returns to the Viewing menu.

Shade Submenu

Opens an option menu to control the production of shaded images.
### Shade

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Flat</td>
<td>Use the flat shading algorithm.</td>
</tr>
<tr>
<td>Gouraud</td>
<td>Use the Gouraud shading algorithm. This option requires that the Z-Buffer option in the Hide Opts command to be active.</td>
</tr>
<tr>
<td>No Shade</td>
<td>Turn off shading.</td>
</tr>
<tr>
<td>Blue, Green, Cyan, Red, Magenta, Yellow, Grey</td>
<td>Choose a color for shading.</td>
</tr>
<tr>
<td>Accept</td>
<td>Accepts command options and produces a shaded image.</td>
</tr>
<tr>
<td>Quit</td>
<td>Abandons shading and returns to the Viewing menu.</td>
</tr>
</tbody>
</table>

#### Shell Shade

The **Shell Shade** command activates shading of all shell elements for subsequent plots such that the top face will be colored red and the bottom face will be colored green. You need to repaint the window or plot the elements to see the effect. The shell shade may be turned off using the **Shade** command. The command is particularly useful in checking that the top of a shell element is connected to the top (not the bottom) of adjacent shell elements so that meaningful results are obtained when nodal stresses are averaged. Top and bottom faces are determined by the nodal connectivity of shell elements (refer to Chapter 4). If the user is
interested in stress results, shell elements must be oriented properly. If the user is only interested in displacements, then orientation of shell elements is not important.

---

**Shell Shade**

**View Submenu**

Sets the viewing point for the model.

<table>
<thead>
<tr>
<th>VIEW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Isometric</td>
</tr>
<tr>
<td>X-view</td>
</tr>
<tr>
<td>Y-view</td>
</tr>
<tr>
<td>Z-view</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

*Figure 5.57  View Submenu*

**View**

**Option & Description:**

*Isometric*

Plots the isometric view.

*X-View*

Plots the model viewed from the positive X-direction.

*Y-View*

Plots the model viewed from the positive Y-direction.

*Z-View*

Plots the model viewed from the positive Z-direction.

*Quit*

Abandons changes to the view and returns to the Viewing menu.

**Pscale**

Redraws the current image by automatically scaling it to fit the window.
Rotate
Rotates the object about the X-, Y- and Z-axes using the specified angles and redraws the window.

Input Description:
- X-rotation in degrees.
  (default is 0)
- Y-rotation in degrees.
  (default is 0)
- Z-rotation in degrees.
  (default is 0)
- Coordinate system to use.
  (default is -1)
- Rescale image.
  (default is 0)

Scale
Replots the object after scaling it by the specified factor. The default value of 0 automatically scales the object to fit the window.

Input Description:
- Scaling factor.
  (default is 0)

Note:
1. This command considers all of the object, even if it is not currently plotted. The Pscale command works with only the plotted entities.

Zoomin
This command needs two opposite corners of a viewing window to be specified and then attempts to fill the screen with the magnified image. The command can be repeated to achieve greater magnification.

Input Description:
- Select the first corner of window by using the mouse.
- Select the opposite corner of window by using the mouse.

**Zoomout**

This command recovers the magnified view previous to the last zoomin command. Up to 10 images may be recovered.

**Dynamic View Submenu**

This submenu provides for the interactive rotation, scaling and translation of models in 3D space. The command can operate in demo or non-demo modes. If the demo mode is selected, the user has no control over the dynamic viewing. If the non-demo mode is selected, the user may use the trackball mode or the coordinate axis mode.

<table>
<thead>
<tr>
<th>DYNAMIC VIEW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Blue Shade</td>
</tr>
<tr>
<td>Green Shade</td>
</tr>
<tr>
<td>Cyan Shade</td>
</tr>
<tr>
<td>Red Shade</td>
</tr>
<tr>
<td>Magenta Shade</td>
</tr>
<tr>
<td>Yellow Shade</td>
</tr>
<tr>
<td>Grey Shade</td>
</tr>
<tr>
<td>Black Background</td>
</tr>
<tr>
<td>Blue Background</td>
</tr>
<tr>
<td>Green Background</td>
</tr>
<tr>
<td>Cyan Background</td>
</tr>
<tr>
<td>Red Background</td>
</tr>
<tr>
<td>Magenta Background</td>
</tr>
<tr>
<td>Yellow Background</td>
</tr>
<tr>
<td>Grey Background</td>
</tr>
<tr>
<td>Demo off</td>
</tr>
<tr>
<td>Demo on</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

*Figure 5.58  Dynamic View Submenu*

**Dynamic View**

**Option & Description:**

*Blue, Green, Cyan, Red, Magenta, Yellow, Grey Shade*

Set shade color.
Blue, Green, Cyan, Red, Magenta, Yellow, Grey Background
Set background color.

Demo off
Go to user mode.

Demo on.
Show demo.

Note:
1. This command is currently only supported on SGI Indigo, Personal Iris (both require 24-bit plane and Z-buffer) and HP700 series with CRX graphics board.

Config Dview
Sets the graphical attributes and some functional parameters for Dynamic View. The defaults shown are those when the command is first used. The defaults used for subsequent use of the command will be the values entered by the user.

<table>
<thead>
<tr>
<th>Config Dview</th>
</tr>
</thead>
</table>

**Input Description:**
- Trackball mode.
  - = 0 Off, use Coordinate Axis Mode.
  - = 1 On, use Trackball Mode.
    (default is 1)

- Mouse movement sensitivity (1-100).
  (default is 10)

- Flip surface normals.
  - = 0 No
  - = 1 Yes
    (default is 0)

- Plot iso or section planes directly in 3D. This flag starts Dynamic View whenever Section or Iso Plots are started.
  - = 0 No
  - = 1 Yes
    (default is 0)

- Plot a coordinate system in the center of the model.
  - = 0 No
  - = 1 Yes
    (default is 0)

- Plot a bounding box around the model.
  - = 0 No
  - = 1 Yes
    (default is 0)
- Backface elimination.
  = 0    Off
  = 1    On
  (default is 0)

**Trackball mode**

The left mouse button controls the motion of the virtual trackball in which the model is embedded. The middle mouse button zooms out of the model and the right mouse button zooms into the model. Translations are not possible in this mode.

**Coordinate Axis Mode**

The left, middle and right mouse buttons control the rotation about the X-, Y- and Z-axes respectively. A single press of one of these buttons causes a 10 degree rotation of the model about the corresponding axis. By pressing a button and moving the mouse, continuous rotation will occur with a magnitude to the distance between the two mouse move events (projected onto a horizontal direction). Moving in different directions will cause clockwise or counterclockwise rotation.

While holding the shift key, a single press of the left mouse button will move the model one step further away on the screen. Continuous zooming is possible by holding down the mouse button and translating the mouse horizontally on the screen. The use of the middle mouse button with the shift key depressed works in a similar way but moves the object closer. Holding the shift key and pressing the left mouse button permits the user to move the model on the screen. The direction of motion is in the same direction as the mouse motion.

There are some special keys to be used with dynamic viewing. Some work as toggle keys setting their corresponding functions on or off.

**Windows Submenu**

A multiple window option is available. It is possible to open up to 4 windows. Only one window is active at one time. The window can be made active, i.e., commands will be sent to that window by clicking the right hand mouse button in the top white banner. The window can be seen as active by a blue square in the top left corner. The window can be manipulated by using the left mouse button in the speckled border to move the window and the middle button to resize the window. The window can be closed by clicking on the blue square and reopened by selecting its icon with the right mouse button.

`| WINDOWS |
<table>
<thead>
<tr>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Window Create</td>
</tr>
<tr>
<td>Window Delete</td>
</tr>
</tbody>
</table>

**Figure 5.59  Windows Submenu**
Window Create

Creates a new window.

Window Delete

Deletes the active window.

Clear

Clears the window.

Repaint

Redraws the current window.
Figure 5.60 Analysis Menu
Figure 5.60  Analysis Menu  (Continued)
Figure 5.60  Analysis Menu  (Continued)

Static Submenu

Opens the options menu to control and submit a model for static analysis. The options are:
**In-Plane Stiffness**

- **No**: Do not include in-plane effects.

---

**Static Submenu**

**Option & Description:**

- **Thermal Loading**: Include temperature loadings.

- **Gravity Loading**: Include acceleration loadings.

- **Centrifugal Loading**: Include angular velocity loadings.

- **All**: Include all of the above types of loading.

- **None**: Include none of these special loads.

- **In-plane Stiffness**: Include in-plane effects in the calculation. For example, a plate in tension will deform less under lateral pressure than if no tension is present when this flag is active.

- **No**: Do not include in-plane effects.
Soft Spring
Add a small term to the diagonal terms of the global stiffness matrix to overcome instability. A value of 1E-6 is used.

No
Do not add this value to the diagonals.

Stress
Perform a stress calculation immediately after the displacement solution.

No
Return to interface after displacement calculation.

Direct Solver
Use the Direct Solver.

1st order FFE Solver
Use the 1st order FFE Solver.

2nd order FFE Solver
Use the 2nd order FFE Solver.

Run
Executes the static analysis with the above options.

Quit
Abort the changes and returns to the Analysis menu.

Frequency Submenu
Opens the options menu to select a type of solution for frequency analysis. The options are:

<table>
<thead>
<tr>
<th>FREQUENCY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Direct Solver</td>
</tr>
<tr>
<td>FFE Solver</td>
</tr>
</tbody>
</table>

Figure 5.62 Frequency Submenu

Direct Solver Submenu
Use the Subspace or Lanczos methods to perform frequency analysis.
Figure 5.63 Direct Solver Submenu (for frequency analysis)

Direct Solver (for frequency analysis)

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Subspace</td>
<td>Use the Subspace iteration method.</td>
</tr>
<tr>
<td>Lanczos</td>
<td>Use the Lanczos method.</td>
</tr>
<tr>
<td>Sturm Sequence</td>
<td>Apply the sturm check for lost eigenvalues.</td>
</tr>
<tr>
<td>No</td>
<td>Do not apply this check.</td>
</tr>
<tr>
<td>Eigenvalue Shift</td>
<td>Applies an Eigenvalue shift, useful when calculating natural frequencies of structures with rigid body modes.</td>
</tr>
<tr>
<td>No</td>
<td>Do not apply this shift.</td>
</tr>
<tr>
<td>In-plane Stiffness</td>
<td>Include in-plane stiffening effects.</td>
</tr>
<tr>
<td>No</td>
<td>Do not include this effect.</td>
</tr>
</tbody>
</table>

**Option & Description:**

*Subspace*
- Use the Subspace iteration method.

*Lanczos*
- Use the Lanczos method.

*Sturm Sequence*
- Apply the sturm check for lost eigenvalues.

*No*
- Do not apply this check.

*Eigenvalue Shift*
- Applies an Eigenvalue shift, useful when calculating natural frequencies of structures with rigid body modes.

*No*
- Do not apply this shift.

*In-plane Stiffness*
- Include in-plane stiffening effects.

*No*
- Do not include this effect.
Soft Spring
Adds a small value to the diagonal terms of the global stiffness matrix to improve stability. A value of 1e-6 is used.

No
Do not add this value.

Lumped
Uses the lumped mass matrix.

Consistent
Uses the consistent mass matrix.

Run
Executes the frequency analysis with the options chosen above.

Quit
Aborts the changes and returns to the Frequency submenu.

Before the analysis is run, the user must input values in the message window as follows:

Input Description:
- Number of frequencies to calculate.
  (default is 1)

- Include modal acceleration.
  = 0  Omit
  = 1  Include
  (default is 0)

- Mass Participation Factor.
  = 0  No
  = 1  Yes
  (default is 0)

- Save Stiffness Matrix flag.
  = 0  No
  = 1  Yes
  (default is 0)

- Frequency for nonlinear analysis flag.
  = 0  No
  = 1  Yes
  (default is 0)

- Form Stiffness Matrix flag.
  = 0  No
  = 1  Yes
  (default is 0)

FFE Solver Submenu
Use the FFE Solver submenu to perform frequency analysis.
Figure 5.64  FFE Solver Submenu (for frequency analysis)

FFE Solver (for frequency analysis)

Option & Description:

1st Ord FFE Solver
Uses 1st order FFE Solver.

2nd Ord FFE Solver
Uses 2nd order FFE Solver.

Done
Executes the frequency analysis with the options chosen above.

Quit
Aborts the changes and returns to the Frequency submenu.

Before the analysis is run, the user must input values in the message window as follows:

Input Description:
- Number of frequencies to calculate. (default is 1)
- Lower bound of frequencies to calculate. (default is 0)
- Upper bound of frequencies to calculate. Enter 0 if the number of frequencies is specified.

Notes:
1. Either the number of frequencies or the upper bound must be 0.
2. The actual number of frequencies calculated will be the number specified + 1 if the specified number is not zero. If the number of frequencies is set to zero, all frequencies in specified range + 1 frequency (outside range) will be calculated.

Buckling Submenu
Opens the options menu to control and submit a model for linear buckling analysis. The options are:
**Figure 5.65  Buckling Submenu**

**Buckling**

**Option & Description:**

*Inverse Power*
- Uses the inverse iteration method to extract one eigenvalue.

*Subspace*
- Uses subspace iteration to extract multiple eigenvalues.

*Eigenvalue Shift*
- Applies an Eigenvalue shift, useful when calculating natural frequencies of structures with rigid body modes.

*No*
- Do not apply this shift.

*Soft Spring*
- Adds a small value to the diagonal terms of the global stiffness matrix to improve stability. A value of 1e-6 is used.

*No*
- Do not add this value.

*Sturm Sequence*
- Applies the check for lowest eigenvalues.

*No*
- Do not apply this check.

*Run*
- Executes the buckling analysis with the options chosen above.
Quit
Aborts the changes and returns to the Analysis menu.

If you selected Subspace, you will have to specify the number of buckling modes desired. Inverse Power evaluates the lowest buckling mode only.

Input Description:
- Number of buckling load factors.
  (default is 1)
- Form Stiffness Matrix flag.
  = 0 Form.
  = 1 Do not form.
  (default is 0)

Stress Submenu
Opens the options menu to control and submit a model for linear stress analysis. The options are:

<table>
<thead>
<tr>
<th>STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Principal Stress</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Run</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.66  Stress Submenu

Stress

Option & Description:

Principal Stress
Starts the stress calculation with the option to print principal stress values.

No
Starts the stress calculation with the option not to print the principal stress values.

Run
Executes stress analysis.

Quit
Aborts the changes and returns to the Analysis menu.
Thermal Submenu

A menu related to specifying options related to performing thermal analysis.

<table>
<thead>
<tr>
<th>THERMAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Load Curves</td>
</tr>
<tr>
<td>Run Thermal</td>
</tr>
</tbody>
</table>

Figure 5.67 Thermal Submenu

Thermal

Option & Description:

Load Curves
Submenu related to management of time and temperature curves.

Run Thermal
Submenu to choose thermal analysis options.

Thermal Load Curves Submenu

Menu to Define, Activate, Plot, List, List, and Delete time and temperature curves used in thermal analysis. Choose the operation type, curve type, and Accept to perform the operation, or Quit to abandon it.

Time curves are used in transient analyses to specify a load or boundary condition history by associating it with a time curve. The procedure to associate a load or boundary condition value with a time curve is:

a. Define and activate a time curve.
b. Define a value for the load or boundary condition.
c. The actual value at a given time will be calculated by multiplying the specified value (of load or boundary condition) by the value of the curve at that time.

Temperature curves are used in a similar manner to define temperature-dependent material properties as follows:

a. Define and activate a temperature curve.
b. Define a value for the material property.
c. The temperature of the element is calculated by averaging the temperatures at the associated nodes.
d. The property value is calculated by multiplying the specified value (of the property) by the curve value at that temperature. Use a property value of "1.0" if the curve represents the actual values of the property at different temperatures.
Note that temperature curves are used in both transient and steady state analyses as well as linear static, frequency, buckling, and nonlinear analyses.

Refer to the Properties and Load/BConds menus for details on association of material properties, loads and boundary conditions with time and temperature curves.

<table>
<thead>
<tr>
<th>LOAD CURVES</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>Activate</td>
</tr>
<tr>
<td>Plot</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Delete</td>
</tr>
<tr>
<td>Time</td>
</tr>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

**Figure 5.68  Thermal Load Curves Submenu**

**Define Curve**

Defines a time or a temperature curve. To define a curve highlight the **Define** option, and choose one option from each group. The options are as follows.

**Option & Description:**

**Time**
- Specify time as the type of the curve.

**Temperature**
- Specify temperature as the type of the curve.

**Accept**
- Accept command options.

**Quit**
- Quit the command.

*Curve data may be read from an external file. The independent variable values must be input in ascending order.*

**Input Description:**

- **Type of input.**
  - 1  Point by point entry.
= 0 Read from an external file.
(default is 1)

- Curve label.
(default is 1)

- Start point (i). (Prompted if type of input is set to 1.)

- Time (temperature) value at point (i). (Prompted if type of input is set to 1.)

- Curve value at point (i). (Prompted if type of input is set to 1.)

- Name of file containing curve data. (Prompted if type of input is set to 0.)

Notes:

1. The file must be an ASCII file formatted in one of two ways:
   a. The first row lists the number of data points to be read, 0 (for the format type), and the increment value of the independent variable. The dependent variable values are listed one value per row from second row onwards. The independent variable will start from zero.
   b. The first row lists the number of data points to be read and 1 (for the format type). The independent and dependent variables values are then listed one pair per row from the second column onwards.

2. A maximum of 5000 points can be defined for a time curve and 200 points for a temperature curve.

3. Curve points must be specified in ascending order of the independent variable.

4. The most recently defined curve becomes the active and hence the default curve. For association with other curves, the user should activate the desired curve first through the 
   **Activate** command.

5. Gravity, centrifugal and temperature loadings as well as forces, pressure, prescribed displacements and base excitations can be independently associated with time curves.

6. The command terminates if two equal consecutive time or temperature values are specified, a blank is entered for the time or temperature value, or the maximum number of points is reached.

**Activate Curve**

Activates a particular time curve (to be associated with loads or boundary conditions) or a temperature curve (to be associated with a material property).

<table>
<thead>
<tr>
<th>Activate Curve</th>
</tr>
</thead>
</table>

**Input Description:**
- Curve label.

**Plot Curve**

Plots a predefined time or temperature curve.
Plot Curve

**Input Description:**
- Graph number.
- Curve number.
- Graph color.
- Graph line style.
- Graph symbol.

List Curve

Lists predefined time or temperature curves.

Delete Curve

Deletes the specified pattern of predefined time or temperature curves.

Run Thermal Submenu

Submenu to choose thermal analysis options. Choose Steady State or Transient, Direct Solver, or FFE Solver, and Run to invoke thermal analysis. or Quit to abort and return to the Analysis menu.

<table>
<thead>
<tr>
<th>RUN THERMAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Steady State</td>
</tr>
<tr>
<td>Transient</td>
</tr>
<tr>
<td>Direct Solver</td>
</tr>
<tr>
<td>1st Ord FFE Solver</td>
</tr>
<tr>
<td>2nd Ord FFE Solver</td>
</tr>
<tr>
<td>Run</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.69  Run Thermal Submenu
Run Thermal

Option & Description:

*Steady State*
- Steady state thermal analysis.

*Transient*
- Transient thermal analysis. Results will be available for all time steps.

*Direct Solver*
- Direct solver using Gauss Elimination to solve the equations resulting from the FE Model.

*1st Ord FFE Solver*
- Use the COSMOS/M FFE with low_order elements. The element groups must be defined as low_order. Middle nodes are defined internally.

*2nd Ord FFE Solver*
- Use the COSMOS/M FFE with high_order elements. The element groups must be defined as low_order. Middle nodes are defined internally.

Note:
COSMOS/M FFE modules
COSMOS/M FFE modules use a superior special iterative solver developed by Structural Research and may be, in many cases, more than 100 times faster in comparison with classical direct solvers. This solver is currently available for the solution of thermal and linear static problems only.

Post Dynamic Submenu

This submenu is related to pre- and postprocessing of various types of post dynamic analyses.

<table>
<thead>
<tr>
<th>POST DYNAMIC</th>
</tr>
</thead>
<tbody>
<tr>
<td>P_dyn Analysis Type</td>
</tr>
<tr>
<td>P_dyn Analysis Curve</td>
</tr>
<tr>
<td>P_dyn Output Opts</td>
</tr>
<tr>
<td>P_dyn Damping</td>
</tr>
<tr>
<td>P_dyn Gap</td>
</tr>
<tr>
<td>Base Excitation</td>
</tr>
<tr>
<td>Run Dynamic</td>
</tr>
<tr>
<td>P_dyn Prepare</td>
</tr>
</tbody>
</table>

Figure 5.70  Post Dynamic Submenu

ASTAR uses mode superposition and hence the natural frequencies and corresponding mode shapes must be available in the problem database prior to running ASTAR.
P_dyn Analysis Type Submenu

A menu to Define and List options for various types of analyses.

<table>
<thead>
<tr>
<th>P_DYN ANALYSIS TYPE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define</td>
</tr>
<tr>
<td>List</td>
</tr>
<tr>
<td>Resp Spectra</td>
</tr>
<tr>
<td>Time History</td>
</tr>
<tr>
<td>Spectra Gen</td>
</tr>
<tr>
<td>Random Vibr</td>
</tr>
<tr>
<td>Harmonic</td>
</tr>
<tr>
<td>Static Stress</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.71  P_dyn Analysis Type Submenu

Define Response Spectra Analysis

Specify input related to response spectra analysis.

**Define Response Spectra Analysis**

**Input Description:**
- Number of frequencies to be used in the analysis.
  (default is 1)
- Mode combination method.
  = 0  SRSS (Square Root Sum of the Square method)
  = 1  CQC (Complete Quadratic method)
  = 2  NRL (Naval Research Laboratory method)
    (default is 0)
- Cluster factor used for the SRSS method only.
  (default is 0)
- Flag defining units of the exciting frequency and Ws & We.
  = 0  Radians/second
  = 1  Cycles/second (Hz)
    (default is 0)
- Lower limit of the exciting frequency to be considered in the analysis (Ws).
  (default is 0.10E-10)
Chapter 5  Command Reference

- Upper limit of the exciting frequency to be considered in the analysis (We). Must be greater than (Ws).
  (default is .0E10)

- Flag for printout of mode displacements.
  = 1  Print mode displacements.
  = 0  Suppress printing of mode displacements.
  (default is 0)

- Flag defining type of response printout.
  = 0  Print response due to each base spectrum separately.
  = 1  Print the combined response (R.M.S) for all base response spectra.
  = 2  Print both of the above options. In addition, write the R.M.S. of response to the plot file.
  (default is 0)

Notes:

1. clusf  < [T(i-1)-T(i)]/T(i-1) -> add mode response using the SRSS method
   clusf  > [T(i-1)-T(i)]/T(i-1) -> add mode response by absolute values
   where:
   T(i-1) & T(i) represent two consecutive natural periods of structure.

2. For the NRL method, maximum response due to a base excitation is evaluated from:
   \[ ABS \{ R \} + SQRT ( \{ SS \} - \{ R**2 \} ) \]
   where:
   \{ SS \} = vector of sum of the squares of mode responses.
   \{ R \} = maximum response among all mode responses for each node and direction.

   The r.m.s. response due to all excitations is evaluated by combining responses (due to different excitations) by the SRSS method.

Define Modal Time History Analysis

Specify input related to modal time history analysis.

Define Modal Time History Analysis

Input Description:
- Number of frequencies to be used in the analysis.
  (default is 1)
- Total number of solution steps.
  (default is 10)
- Starting time for solution.
  (default is 0.0)
- Time step size.
- Time integration method.
  = 0    Newmark method
  = 1    Wilson-theta method
         (default is 0.0)

- First integration parameter.
  Newmark method:  [\delta]  
         (default is 0.50)
  Wilson-theta method: [\theta]  
         (default is 1.40)

- Second integration parameter.
  Newmark method:  [\alpha]  
         (default is 0.25)
  Wilson-theta method:  not used

- Type of response printout.
  = 0    Print relative displ. and relative vel.
  = 1    Print relative displ. but absolute vel.
  = 2    Print absolute displ. and absolute vel.
         (default is 0)

Note:
1. Accelerations are always printed in absolute values.

Define Response Spectra Generation

Specify input related to response spectra generation.

<table>
<thead>
<tr>
<th>Define Response Spectra Generation</th>
</tr>
</thead>
</table>

Input Description:
- Number of frequencies to be used in the analysis.
  (default is 1)
- Starting frequency for a response spectrum generation in rad/sec.
  (default is 0)
- Ending frequency for a response spectrum generation in rad/sec.
  (default is 0)
- Flag for response spectrum abscissa scale.
  = 0    Logarithmic scale
  = 1    Linear scale
         (default is 0)
- Number of points to define the response spectrum curve.
  (default is 1)
- Label of the node at which response spectrum is to be generated.
  (default is 1)
- Damping ratio associated with this node.
  (default is 0.0)

- Flag for generation of response spectrum for translation in the X-direction.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

- Flag for generation of response spectrum for translation in the Y-direction.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

- Flag for generation of response spectrum for translation in the Z-direction.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

- Flag for generation of response spectrum for rotation about the X-axis.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

- Flag for generation of response spectrum for rotation about the Y-axis.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

- Flag for generation of response spectrum for rotation about the Z-axis.
  = 0 Do not generate response spectrum.
  = 1 Generate response spectrum.
  (default is 0)

**Define Random Vibration Analysis**

Specify input related to random vibration analysis.

<table>
<thead>
<tr>
<th>Define Random Vibration Analysis</th>
</tr>
</thead>
</table>

**Input Description:**

- Number of frequencies to be used in the analysis.
  (default is 1)

- Flag defining units of the exciting frequency and Ws & We.
  = 0 Radians/second
  = 1 Cycles/second (Hz)
  (default is 0)

- Lower limit of exciting frequency to be considered in the analysis (Ws).
  (default is 0.1E-10)
- Upper limit of exciting frequency to be considered in the analysis (We). Must be greater than Ws.
  (default is 1.0E10)

- Correlation flag.
  = 0  Fully correlated.
  = 1  Fully uncorrelated.
  (default is 0)

- Method of analysis.
  = 0  Standard method.
  = 1  Approximate method.
  (default is 0)

- Number of frequency points to be selected between any two adjacent natural frequencies.
  (See notes below)
  (default is 1)

- Gauss Integration order to be used in integration of response power spectral densities.
  = 2  Two point Gaussian integration.
  = 3  Three point Gaussian integration.
  (default is 2)

- Biasing parameter used to define the location of frequency points to be selected. (See notes below)
  (default is 0)

- Cross-mode cut-off ratio defines a limit on the ratio of any two mode frequencies (Wi/Wj, i>j). If Wi/Wj is greater than this ratio, the cross-mode effects between the two modes will not be considered.
  (default is 1.0E10)

- PSD stress computation flag.
  = 0  Do not compute PSD of stresses.
  = 1  Compute PSD of stresses for each solution step.

Notes:

1. The random excitations must be stationary, Gaussian, with a mean value of zero, defined by one-sided power spectral density curves.
2. The correlation flag is considered when base excitations in different directions or forces in more than one node are prescribed. If the excitations are considered to be correlated then the cross-spectral density terms will also be included in the analysis.
3. The standard method performs a classical random vibration analysis. The approximate method uses the assumption that power spectral densities around each mode are flat and cross-mode effects can be neglected.
4. The last four fields are only used if the standard method is selected.
5. Number of frequencies points and bias parameter are used to select frequency points at which P.S.D. of response is evaluated and serve as steps for a numerical integration in the frequency domain. These parameters must be selected to minimize integration error and computational effort as well.

The default values for number of frequency points and bias parameter are dependent on the modal critical damping ratio. (See table below).
6. If the PSD stress computation flag is activated, RMS stresses will be stored in step "n+1" location, where "n" is the number of solution steps. The PSD stresses are stored in the corresponding location for each solution step.

<table>
<thead>
<tr>
<th>Modal Damping Ratio $\zeta$</th>
<th>Default for Number of Freq Points</th>
<th>Default for Bias Parameter</th>
</tr>
</thead>
<tbody>
<tr>
<td>$\zeta &lt; 0.01$</td>
<td>21</td>
<td>11.</td>
</tr>
<tr>
<td>$0.01 &lt; \zeta &lt; 0.1$</td>
<td>$21 - 4.34 \ln(\zeta/0.01)$</td>
<td>$11. - 3.47 \ln(\zeta/0.01)$</td>
</tr>
<tr>
<td>$\zeta &gt; 0.1$</td>
<td>1</td>
<td>3.</td>
</tr>
</tbody>
</table>

**Define Harmonic Analysis**

Specify input related to harmonic analysis.

**Define Harmonic Analysis**

**Input Description:**

- Number of frequencies to be used in the analysis.
  (default is 1)

- Flag defining units of the exciting frequency and Ws & We.
  
  = 0  Radians/second
  = 1  Cycles/second (Hz)
  (default is 0)

- Lower limit of exciting frequency to be considered in the analysis (Ws).
  (default is 0.10E-10)

- Upper limit of exciting frequency to be considered in the analysis (We). Must be greater than (Ws).

- Number of output frequencies to be selected in the range of exciting frequencies to be considered.
  (default is 1)

- Flag for locating frequency points.
  
  = 0  Logarithmic interpolation.
  = 1  Linear interpolation.
  (default is 0)

- Type of response printout.
  
  = 0  Write relative displ. and relative vel. in both the output and plot files.
  = 1  Write relative displ. but absolute vel. in both the output and plot files.
  = 2  Write absolute displ. and absolute vel. in both the output and plot files.
  (default is 0)

**Note:**

1. Accelerations are always printed in absolute values.
Define Static Stress Analysis

Specifies that subsequent stress run will be based on results from linear static analysis (not from dynamic analysis).

List P_dyn Analyses

Lists the active options for the chosen analysis type.

P_dyn Analysis Curve Submenu

Menu to Define, Activate, Plot, List, List, and Delete time, frequency, and harmonic curves used in postdynamic analysis. Choose the operation type, curve type, and Accept to perform the operation, or Quit to abandon it.

Define P_dyn Analysis Curve

Defines a curve to be used in postdynamic analysis. The syntax and prompts depend on the type of the curve to be defined. The curve data may be input through the keyboard, or read from an external file. The independent variable values (time/frequency) must be given in ascending order.
Input Description:
- Curve label (limit is 100).
  (default is highest defined + 1)

- Excitation type.
  = 0  Force or pressure.
  = 1  Uniform base excitation.
  = 2  Multi-base excitation.
  (default is last defined value or 0)

- Data entry flag.
  = 0  Read from a file.
  = 1  Input using keyboard.
  (default is 1)

- Type of the curve.
  = 0  Time-dependent curve defined by points on the curve.
  = 1  Frequency-dependent curve defined by points on the curve.
  = 2  Time-dependent curve defined coefficients and frequencies.
  (default is last type used or 0)

The syntax after this entry depends on the type of the curve as shown below.

Syntax for Time- or Frequency-dependent curve defined by points

Define Time- or Frequency-dependent Curve

Input Description:
- Starting location index to define data (not prompted if reading from a file).
  (default is highest defined + 1)

- Name of file containing data. The file must be an ASCII file, in which the first row has
  the number of data points, and data points follow in the second column onwards, where
  each row lists a time/frequency value, and the corresponding curve value for a data point.
  Prompted only if reading from a file.
  (default is pname.xcr)

The following entries are not prompted if data is read from an external file.

- Time at the ith point for time-dependent curves or frequency at the ith point for frequency-
  dependent curves.

- Value of the curve at the ith point (pairs are prompted for 10 points at a time).

Notes:
  1. The starting point argument allows modifying existing curve points. Use default values to
     define additional points to an existing curve.
  2. Up to 5000 points can be defined for a curve.
  3. Up to 10 pairs can be defined each time the command is issued. The command terminates
     if two equal consecutive time values are specified or a blank is entered for the time or
     frequency value.
Syntax for defining Time-dependent Harmonic curve by coefficients and frequencies

Define Time-dependent Harmonic Curve

Input Description:
- Curve label.
  (default is highest defined + 1)
- Starting time for harmonic curve.
  (default is 0.0)
- Ending time for harmonic curve.
- Amplitude of sine function.
  (default is 0.0)
- Frequency of sine function in radians/second.
  (default is 0.0)
- Phase angle of sine function in radians.
  (default is 0.0)
- Amplitude of cosine function.
  (default is 0.0)
- Frequency of cosine function in radians/second.
  (default is 0.0)
- Phase angle of cosine function in radians.
- Constant of exponential function.
  (default is 0.0)

Delete P_dyn Analysis Curves

Deletes a pattern of post dynamic curves.

Delete P_dyn Analysis Curves

Input Description:
- Beginning post dynamic curve in the pattern.
- Ending post dynamic curve in the pattern.
  (default is beginning curve)
- Increment between post dynamic curves in the pattern.
  (default is 1)
List P_dyn Analysis Curves

Lists data points of post dynamic curves. For a harmonic curve, the command lists the constants defining the harmonic function.

**List P_dyn Analysis Curves**

**Input Description:**

- Curve label.
  (default is highest defined + 1)

- First point on the curve to be listed (not prompted for a harmonic curve).
  (default is 1)

- Last point on the curve to be listed (not prompted for a harmonic curve).
  (default is highest point defined)

**P_dyn Output Opts Submenu**

Submenu to control the information that will be printed in the output file (pname.OUT) as well as information to be stored for plotting and listing in the postprocessing stage.

**P_dyn Print**

Controls the information that will be printed in the output file (pname.OUT).

**Input Description:**

- Flag for displacement printout.
  
  = 1  Print displacements.
  
  = 0  Do not print displacements.
  
  (default is 0)
- Flag for velocity printout.
  = 1 Print velocities.
  = 0 Do not print velocities.
    (default is 0)

- Flag for acceleration printout.
  = 1 Print accelerations.
  = 0 Do not print accelerations.
    (default is 0)

- Phase angle printout.
  = 0 Suppress phase angle printout.
  = 1 Prepare phase angle printout used for harmonic (frequency response) analysis only.
    (default is 0)

- Flag for missing mass correction factor printout.
  = 0 Do not print mass correction factors.
  = 1 Print correction factors (see note below).
    (default is 0)

The next three arguments are prompted for time history, harmonic and random vibrations analyses only.

- Beginning solution step for stress printout.
  (default is 1)

- Ending solution step for stress printout.
  (default is 1)

- Increment between solution steps.
  (default is 1)

Notes:

1. For a random vibration analysis these flags control printouts of response (as well as stress) power spectral densities. The R.M.S. displacements, velocities, and accelerations are printed by default.

2. Response and stress printout options are only applicable to modal time history and harmonic analyses. For random and response spectra analyses, the R.M.S. stresses are available through the Run Stress command.

3. The missing mass correction factor accounts for the contribution of higher frequency modes not considered in the analysis. This factor is printed in the output file, and should be used to correct the acceleration and stress responses by multiplying the corresponding values by this factor.

**P_dyn Plot**

A preprocessing command that must be issued before running ASTAR to select the solution steps at which the nodal response is to be stored in the database. The nodal response includes displacements, velocities, and accelerations. If stresses are calculated, nodal results will also be available at the specified steps. Five patterns of solution steps may be specified. Note that
Nodal Resp may be used to select the nodes for which the response is desired at all solution steps.

**P_dyn Plot**

**Input Description:**
- Starting solution step for the ith pattern.
- Ending solution step for the ith pattern.  
  (default is starting step)
- Increment between solution steps for the ith pattern.  
  (default is 1)

**Plot Interval**

Specifies the pattern of solution steps for the xy-plotting of results requested by **Nodal Response** and **P_dyn Stress Opts**. Note that if this command is not issued, then the xy-plot files will include information for all solution steps. Use this command to save disk space and time by reducing the resolution of the requested xy-plots.

**Nodal Response**

A preprocessing command that must be issued before running ASTAR to request storing all response information for the selected nodes. The response at the selected nodes is stored at all steps in the database for later xy-plotting. A maximum of 50 nodes can be selected. The command continues to prompt for more nodes until location 50 is filled, or a null entry for the node number is entered.

**Input Description:**
- Starting location.  
  (default is highest defined + 1)
- Node number at the Starting location.
- Node number at the Starting location + 1.

Relative Response

Requests the calculation of relative response (displacement, velocity, and acceleration) at a specified set of nodes relative to a corresponding set of reference nodes. The command continues to prompt for pairs of nodes until limit is reached or entering "0" at the Node number prompt.

**Relative Response**

**Input Description:**
- Node number at which response is desired.
- Corresponding reference node. If 0 is entered, response will be evaluated relative to the uniform base motion (if an), or to the ground.

**Notes:**
1. Up to 10 sets can be specified by this command.
2. Repeating this command overwrites all previous assignments.
3. A value of zero for the reference node evaluates the response relative to the uniform base motion (if any) or to the ground.
4. Use **P_dyn Prepare** to view the relative response results after a successful run.

Peak Response

A postprocessing command that searches for nodes with highest response values among a set of nodes and in a given time (or frequency) range, and lists the peak response at these nodes. A maximum of 10 nodes from a set of 300 nodes can be listed each time.

Note that by limiting the number of nodes in the set to the number of nodes to be listed, the maximum response attained by each node with respect to time (frequency) and its time (freq.) of occurrence will be listed (1 to 10 nodes at a time). On the other hand, by limiting the time (freq.) range, nodal response at a particular step of solution can be obtained. **P_dyn Prepare** must be issued to prepare the files for listing the max/min values, and **PD Extremes** from the Postprocessing-Lists-Extremes submenu should be used to list the results.

**Peak Response**

**Input Description:**
- Flag defining type of peak response desired.
  - = 1 Displacement.
  - = 2 Velocity.
  - = 3 Acceleration.
    (default is 1)
- Flag defining direction of maximums.
  = 1  X-translation.
  = 2  Y-translation.
  = 3  Z-translation.
  = 4  X-rotation.
  = 5  Y-rotation.
  = 6  Z-rotation.
  (default is 1)

- Total number of nodes with maximum response values to be listed (limit is 10 nodes).
  (default is 10)

- Peaks will be searched among nodes starting from node_1 and ending at node_2 [ (node_2
  -node_1) < 300 ].

- Maximums will be selected in the time (frequency) range stating from time_1 (freq_1) and
  ending at time_2 (freq_2).
  (default is total range of analysis)

**Note:**
- Time-history analysis ; Response based on time.
- Random Vibration ; Power spectral densities based on the exciting frequency.
- Frequency Response ; Amplitude of harmonic motion based on the exciting frequency.

In the case of random vibration or frequency response, the frequency range limits are
assumed to have the same units as the excitation curves.

### P_dyn Stress Opts

Requests the ASTAR module to generate a file that records a stress component for a specified
element versus time or frequency depending on the analysis type. Forces and moments can
also be requested for 1-D elements such as beams and pipes. The command must be issued
prior to performing stress analysis using Run Stress.

#### Input Description:
- Plot number (limit is 10).
  (default is highest defined + 1)
- Element label.
- Stress, force or moment component (valid entries are 1 through 6).
- Location of force or moment defined by local node number (refer to notes below). (Used
  for 1-D elements only.)

#### Notes:
1. Direction is an integer between 1 & 6 defining the direction of a stress, force or moment
   component. Stresses are considered at the center of elements. Values from 1 to 3 refer to
   normal stresses in x, y, and z directions, and from 4 to 6 refer to shear stresses tau(xy),
tau(xz), and tau(yz), respectively. For 1-D elements, forces or moments can be requested. Components are specified by the their direction as follows:

1. \( N \) = Normal force
2. \( Vs \) = Shear in the s-direction
3. \( Vt \) = Shear in the t-direction (for beam, pipe, and elbow elements)
4. \( Tr \) = Torsion
5. \( Ms \) = Moment in the s-direction
6. \( Mt \) = Moment in the t-direction

The location of forces or moments for 1-D elements is specified by the local node number (i.e. the order of the node in the element definition as in obtained by listing elements).

\[
\begin{align*}
1 & \text{ Element end i} \\
2 & \text{ Element end j} \\
3 & \text{ Element center (for elbow elements only)}
\end{align*}
\]

For each stress run, for one element and one direction, plot can be requested at one location only.

2. Plots of different components of stress can be requested for the same element. For stress plots at different elements, the element labels must be input in ascending order.

3. Forces or moments can only be requested at one node for a specified element.

**List P\_dyn Stress**

Lists all plot information available in the database related to post dynamic analysis.

**P\_dyn Damping Submenu**

This menu deals with defining, listing, and deleting damping. A concentrated damper may be created between two nodes.
Rayleigh Damp

Defines the parameters for Rayleigh (proportional) damping. The damping matrix \([C]\) is then calculated as:

\[
[C] = \alpha \times [M] + \beta \times [K]
\]

where:

- \(\alpha\) = Coefficient of mass matrix
- \(\beta\) = Coefficient of stiffness matrix
- \([M]\) = Mass matrix
- \([K]\) = Stiffness matrix

**Rayleigh Damp**

**Input Description:**
- Coefficient of Mass matrix (\(\alpha\)).
- Coefficient of Stiffness matrix (\(\beta\)).

Modal Damp

Defines modal damping (ratio of damping to the critical damping). Different damping values can be assigned to different sets of modes. Each set includes modes with a common modal damping. Up to 8 sets of modes can be defined.

**Modal Damp**

**Input Description:**
- Set number for modes with common modal damping ratio.
- First mode in the set.
- Last mode in the set.
  (default is first mode in set)
- Critical damping ratio for the set.

Read Damp

Activates damping ratios calculated based on specified Composite Material Damping using the Material Damping property. Information given by the **Modal Damping** command will be overwritten.

**Read Damp**

**Input Description:**
- Set number for modes with common modal damping ratio.
- First mode in the set.

- Last mode in the set.
  (default is first mode in set)

- Critical damping ratio for the set.

**Note:**

1. The following steps must be followed for proper use of this command:
   a. Define the Composite Material Damping (use command **Additional Prop** in Properties/Material menu prior to running Frequency (DSTAR)).
   b. Issue this command after running DSTAR.
   c. Use **List Conc-damper** to list calculated damping ratios.
   d. Use **Modal Damp** to modify the calculated damping ratios if desired.
   e. Run ASTAR.

**List Damp**

Lists available Rayleigh and Modal damping information.

**Define Conc-damper**

Defines a concentrated damper to be used in modal time-history analysis.

**Input Description:**

- Label of this concentrated damper.
  (default is highest defined + 1)

- Node label at one end of damper.

- Node label at the other end of damper. If = 0, damper will be connected to ground.

- Value of concentrated damper in X-direction.
  (default is 0.0)

- Value of concentrated damper in Y-direction.
  (default is 0.0)

- Value of concentrated damper in Z-direction.
  (default is 0.0)
Delete Conc-damper

Deletes a pattern of concentrated dampers.

**Delete Conc-damper**

**Input Description:**
- Beginning damper in the pattern.
- Ending damper in the pattern.
  (default is beginning damper)
- Increment between dampers in the pattern.
  (default is 1)

List Conc-damper

Lists a pattern of concentrated dampers.

**List Conc-damper**

**Input Description:**
- Beginning damper in the pattern.
- Ending damper in the pattern.
  (default is beginning damper)
- Increment between dampers in the pattern.
  (default is 1)

P_dyn Gap Submenu

Submenu to Define, Delete, and List gap elements for use with modal time-history analysis.

**Figure 5.75  P_dyn Gap Submenu**

Define Gap

Defines gap elements for modal time-history analysis.
Define Gap

**Input Description:**
- Label assigned to this gap element.
  (default is highest defined + 1)

- Node label defining one end of the gap.

- Node label defining the other end of the gap.

- Allowable relative displacement between the two nodes with no gap resistance.
  > 0.0  Gap resists compression.
  = 0.0  Gap is originally closed & resists compression.
  < 0.0  Gap resists tension.

- Gap-stiffness used to evaluate gap resistant force.
  (default is 1.E7)

- Coefficient of friction used to define the magnitude of friction force to be applied in the direction normal to the gap resisting force.

- Defines type of gap element. Prompted only if coefficient of friction > 0.0.
  = 0  Regular gap element.
  = 1  Once the gap is closed, the evaluated gap force remains constant during the rest of the analysis. This option is useful when friction is caused by a constant normal force.

### Demonstration of a Dynamic Gap-friction Element

![Diagram](image)

where:

- $F_s$ = friction force in the s-direction $\leq G_{fric} \times F_n$
- $F_n$ = compressive gap force in the n-direction and proportional to $G_{stiff}$
- $V_{rel}$ = relative velocity in the s-direction

### List Gap

Lists a pattern of gap elements defined for post dynamic analysis.
**List Gap**

**Input Description:**
- Beginning gap element in the pattern.
- Ending gap element in the pattern. (default is beginning gap)
- Increment between gap elements in the pattern. (default is 1)

**Delete Gap**

Deletes a pattern of gap elements defined for post dynamic analysis.

**Delete Gap**

**Input Description:**
- Beginning gap element in the pattern.
- Ending gap element in the pattern. (default is beginning gap)
- Increment between gap elements in the pattern. (default is 1)

**Base Excitation Submenu**

Submenu to Define, List, and Delete base excitation.

<table>
<thead>
<tr>
<th>BASE EXCITATION</th>
</tr>
</thead>
<tbody>
<tr>
<td>Define Base Excitation</td>
</tr>
<tr>
<td>List Base Excitation</td>
</tr>
<tr>
<td>Excite Node</td>
</tr>
<tr>
<td>Delete Node Excite</td>
</tr>
<tr>
<td>List Motion</td>
</tr>
</tbody>
</table>

**Figure 5.76  Base Excitation Submenu**

**Define Base Excitation**

Defines data for base excitation.
Define Base Excitation

Input Description:
- Type of uniform or multi-base excitation.
  = 0 Velocity.
  = 1 Acceleration.
  = 2 Displacement.
  (default is last defined value or 0)

The following three arguments are only valid for uniform base excitation.

- Base curve multiplier for translation in the X-direction.
  (default is 0.0)

- Base curve multiplier for translation in the Y-direction.
  (default is 0.0)

- Base curve multiplier for translation in the Z-direction.
  (default is 0.0)

- Phase angle of the base excitation in degrees (used in harmonic analysis only for all types of base motion).
  (default is 0.0)

Note:
1. The form of the harmonic excitation is:
   Sine/Cosine (frequency*time + phase *2*pi/360.)
   where * indicates multiplication.

List Base Excitation

Lists uniform base excitations.

Excite Node

Applies a base excitation to a pattern of nodes. The excitation in the specified direction is defined by the active post dynamic curve and a multiplier. The command can be repeatedly used to define multiple base motions for other patterns of nodes. The type of base excitation (displacement, velocity, or acceleration) is specified by Define Base Excitation.

Excite Node

Input Description:
- Support level, a label assigned to the motion to be defined. Maximum number allowed is 100.
  (default is highest defined + 1)
- Beginning node in the pattern.

- Input component of excitation.
  = 1 Global X direction.
  = 2 Global Y direction.
  = 3 Global Z direction.
  = 4 Rotation about global X direction.
  = 5 Rotation about global Y direction.
  = 6 Rotation about global Z direction.

- Multiplier to the active curve to scale the magnitude of displacement.

- Ending node in the pattern.
  (default is beginning node)

- Increment between nodes in the pattern.
  (default is 1)

Notes:

1. Base motion assignments and the type of dynamic analysis to be performed must be specified prior to calculating the natural frequencies and mode shapes.
2. A number of Support Levels may be associated with one curve.
3. Only one base excitation is considered for a given node and input component. Uniform base excitation however may be added on top of the defined motion using Define Base Excitation.
4. For Response Spectra Analysis, all base nodes associated with a multi-base motion curve, should be excited in the same direction.
5. For Random Vibration Analysis, all the base nodes associated with a multi-base motion curve, must have the same curve multiplier.
6. All the input for the support levels must be given prior to running DSTAR, any modifications after that are ignored by ASTAR, unless DSTAR is run again.

Delete Node Excite

Deletes a pattern of support levels (base motions previously defined by Excite Node).

Delete Node Excite

Input Description:
- Beginning support level in the pattern.
- Ending support level in the pattern.
  (default is beginning level)
- Increment between support levels.
  (default is 1)
**List Motion**

Lists a pattern of support levels (base motions previously defined by **Excite Node**).

**List Motion**

**Input Description:**
- Beginning support level in the pattern.
- Ending support level in the pattern.
  (default is highest defined)
- Increment between support levels.
  (default is 1)

**Run Dynamic**

Run the type of dynamic analysis specified. Use **List** in the P_dyn Analysis Type submenu to list the active options. The natural frequencies and mode shapes must be available when this command is issued.

**Run Dynamic**

**P_dyn Prepare**

Prepares files for listing of the extreme response and relative response values after a successful post dynamic run.

**P_dyn Prepare**

**Input Description:**
- Flag to indicate the type of files to be prepared.
  = 1 Prepares a file for listing the extreme response values specified by **Peak Response**.
  = 2 Evaluates the relative response for the nodes previously defined by **Relative Response**. The generated relative response is written to the output file.
- Time or frequency interval for relative response calculations. Prompted only for modal time history and harmonic analyses.

**Nonlinear Submenu**

Opens the nonlinear menu to provide the means to solve nonlinear structural problems. The nonlinear submenu contains the following submenus.
Load Curves Submenu

This submenu contains the commands to Define, Activate, Plot, List, and Delete time and/or temperature curves. Time curves are used in association with loads and boundary conditions. Temperature curves are used in association with various material properties. Refer to the Additional Prop command for details on which material properties may be associated with temperature curves.

Define Curve

Defines a time or a temperature curve. To define a curve, highlight the Define option and choose one option from each group. The options are as follows:

- Time: Specify time as the type of the curve.
- Temperature: Specify temperature as the type of the curve.
- Accept: Execute the highlighted options.
- Quit: Quit the command.
Curve data may be read from an external file. The independent variable values must be input in ascending order.

**Define Curve**

**Input Description:**
- Type of input.
  - = 1  Point by point entry.
  - = 0  Read from an external file.
    (default is 1)

- Curve label.
  (default is 1)

- Start point (i). (Prompted if type of input is set to 1.)

- Time (temperature) value for point (i). (Prompted if type of input is set to 1.)

- Function value for point (i). (Prompted if type of input is set to 1.)

- Name of file containing curve data. (Prompted if type of input is set to 0.)

**Notes:**
1. The file must be an ASCII file formatted in one of two ways:
   a. The first row lists the number of data points to be read, 0 (for the format type), and the increment value of the independent variable. The dependent variable values are listed one value per row from second row onwards. The independent variable will start from zero.
   b. The first row lists the number of data points to be read and 1 (for the format type). The independent and dependent variables values are then listed one pair per row from the second column onwards.
2. A maximum of 5000 points can be defined for a time curve and 200 points for a temperature curve.
3. Curve points must be specified in ascending order of the independent variable.
4. The most recently defined curve becomes the active and hence the default curve. For association with other curves, the user should activate the desired curve first through the **Activate** command.
5. Gravity, centrifugal and temperature loadings as well as forces, pressure, prescribed displacements and base excitations can be independently associated with time curves.

**Activate Curve**

Activates a particular time curve (to be associated with loads or boundary conditions) or a temperature curve (to be associated with a material property).
Input Description:
- Curve label.

**Plot Curve**
Plots a predefined time or temperature curve.

**Plot Curve**

Input Description:
- Graph number.
- Curve number.
- Graph color.
- Graph line style.
- Graph symbol.

**List Curve**
Lists predefined time or temperature curves.

**List Curve**

**Delete Curve**
Deletes the specified pattern of predefined time or temperature curves.

**Delete Curve**

Input Description:
- Curve type.
- The beginning curve in the pattern.
- The ending curve in the pattern.
- The increment between the curves in the pattern.

**Analysis Opts Submenu**
This menu contains commands related to options available in running nonlinear analysis.
The **NL Analysis Option** command specifies details for the nonlinear analysis to be performed by the NSTAR module through the **Run Nonlinear** command.

### NL Analysis Option

**Input Description:**
- Nonlinear analysis option.
  - = S  Static analysis.
  - = D  Dynamic analysis.
- Number of solution steps between reforming the stiffness matrix.
- Number of solution steps between equilibrium iterations.
- Maximum number of equilibrium iterations at each step.
- Convergence tolerance used for equilibrium iterations.
- Not used. Restart information is always saved.
- Special loading flag. Any one character can be assigned. Two or three characters can be assigned in any combination of C, G and T.
  - = N  Do not include special loading.
  - = C  Include centrifugal loading.
  - = G  Include gravity loading.
  - = T  Include thermal loading.
- Flag for displacement-dependent loading.
  - = 0  Direction and area considered for pressure loading are constant.
  - = 1  Direction and area may change based on the deformed element configuration.
- Flag for end moment calculation for shell elements under pressure loading.
  - = 0  Do not apply fixed end moments.
  - = 1  Apply fixed end moments.
- Local constraint penalty stiffness value to be used in the calculation of the penalty matrix.
- Constraint tolerance for local prescribed displacements.

- Creep/Plasticity strain increment tolerance.

- Geometry update flag.
  
  = 0  Do not update geometry.
  
  = 1  Update geometry.

Notes:

1. The specifications of this command are ignored by the NSTAR module if no nonlinearities are specified in the Define Element Group command.

2. Special loadings are considered in addition to the regular forces and pressure loadings.

3. The geometry update flag should only be used when it is desired to use the deformed shape of the structure at the final solution step as the original shape (with no strains or stresses) for subsequent analyses.

Example:  

NL Analysis Option, S, 1, 5, 25, , , CG, 0, 0,,,,,  

This command specifies nonlinear static analysis with stiffness reformation at every time step, and equilibrium iterations at every fifth time step. The maximum number of equilibrium iterations is set to 25. Default displacement tolerance is used. Gravity and centrifugal loadings are included. Fixed end moments for shell elements under pressure are not considered.

NL Control

The NL Control command specifies the numerical procedure to be used in nonlinear analysis. The command specifies the control technique (Force, Displacement, or Arc-Length) and the iterative method (Modified Newton-Raphson (MNR), Newton-Raphson (NR), or Broyden-Fletcher-Goldfarb-Shanno (BFGS) to be used in the analysis.

Force control  
The applied loads (prescribed displacements, thermal loads, base motion, gravity loading, ...) are incremented according to associated time curves.

Displacement control  
The pattern of the applied loads is proportionally incremented to achieve equilibrium under the control of the specified degree of freedom. The controlled DOF is incremented through the use of a time curve. It can be used for nonlinear static analysis with structural loads. This option cannot be used for the dynamic analysis.

Arc-length control  
The pattern of the applied loads is proportionally incremented to achieve equilibrium under the control of the length of the equilibrium path. No time curve is needed. It can be used for static analysis with structural loads.

MNR iteration  
The stiffness matrix is formed and decomposed at the user-specified reformation interval. The computed stiffness is then used throughout that interval during the iteration process. Refer to the NL Analysis Option command.

NR iteration  
The stiffness matrix is formed and decomposed at each iteration within a particular step.
BFGS iteration

A BFGS Quasi-Newton update is used during iterations within a particular step (used only with Force Control for static analysis).

**NL Control**

**Input Description:**

- Control technique flag.
  - = 0 Force control (static and dynamic analyses).
  - = 1 Displacement control (static analysis).
  - = 2 Riks Arc-length control (static analysis).

- Iterative method flag.
  - = 0 MNR iteration (static and dynamic analyses).
  - = 1 NR iterations (static and dynamic analyses).
  - = 2 BFGS iterations (static analysis with Force control only).

- Line search flag, used in the BFGS method only.
  - = 0 Off, do not use line search.
  - = 1 On, use line search.

- Search tolerance, used in the BFGS method only.

- The node number for which the degree of freedom is controlled during the solution process. (For Displacement control only.)

- Displacement component. Indicates the direction of the controlled DOF. (For Displacement control only.) 
  - UX Displacement in the Global X-direction.
  - UY Displacement in the Global Y-direction.
  - UZ Displacement in the Global Z-direction.
  - RX Rotation about the Global X-direction.
  - RY Rotation about the Global Y-direction.
  - RZ Rotation about the Global Z-direction.

- The maximum load-pattern multiplier at which the analysis is to be terminated if exceeded (approximate value). (For Riks Arc-Length control only.)

- The maximum value of any DOF at which the analysis is to be terminated if exceeded (approximate value). (For Riks Arc-Length control only.)

- The maximum number of arc steps at which analysis is to be terminated if exceeded. (For Riks Arc-Length control only.)

- The desired average number of iterations for each arc step used in adjusting the length of the arc steps. (For Riks Arc-Length control only.)

- The initial trial load multiplier used to calculate the first arc step. (For Riks Arc-Length control only.)

- The flag used to select the criterion for determining the state of unloading during the solution process. (For Riks Arc-Length control only.)
  - = 0 Use the sign of the determinant of the system stiffness matrix (negative sign indicates unloading).
= 1  Use the sign of the incremental work (negative sign indicates unloading).
= 2  Use the sign of the determinant and the incremental work (a negative sign for any of them indicates unloading).
= 3  Use the sign of the determinant and the incremental work (negative signs for both indicates unloading).

- The coefficient used in adjusting the automatic calculation of the arc-step length increment during the analysis. (For Riks Arc-Length control only.)

Notes:

1. The NR, MNR, and BFGS iterative methods are effective only if equilibrium iterations are performed.
2. The user must define either (rlmax) or (rumax) for the Arc-Length technique. Defaults should not be used for both of them. Otherwise, an error message is issued by the program 'Max load parameter or displ/rotation must be defined'.

Example: Let nonlinear static analysis be active.
NL Control, 0, 2, 1, 0.5,
This command activates the Force control technique with BFGS iterations including line search option with search tolerance 0.5.

Example: NL Control, 1, 1, 5, UY,
This command activates the Displacement control technique with NR iterations. The controlled node is node 5 and the displacement component is UY.

Example: NL Control, 2, 0, 100, 30, 50, 5, 10, 0, 0.5
This command activates the Riks Arc-Length control technique with MNR iterations. The Arc-Length control information are:
Max. load parameter = 100.0
Max. displacement/rotation = 30.0
Max. number of arc steps = 50
Desired average number of iterations/step = 5
Initial load parameter = 10.0
Initial load parameter = 10.0
Unloading check flag = 0
Arc-Length step adjustment coefficient = 0.5

**NL R-damping**

The **NL R-damping** command is used to activate the proportional Rayleigh damping option to be used in nonlinear dynamic analysis only. The actual damping matrix is calculated by multiplying the mass and stiffness matrices by the specified coefficients and adding the resulting matrices.

**Input Description:**
- Rayleigh damping flag.
  = 0  Off, no proportional damping.
On, consider proportional damping.
- Damping coefficient associated with the stiffness matrix.
- Damping coefficient associated with the mass matrix.

Example: NL R-damping, 1, 0.01, 0.1.
This command activates the Rayleigh damping option during nonlinear dynamic analysis and defines a value of 0.01 to be associated with \[K\] and a value of 0.1 to be associated with \[M\] to define \[C\].

**NL Base**

The **NL Base** command is used to activate the uniform base motion acceleration option to be used in nonlinear dynamic structural analysis (NSTAR). Actual values of acceleration are specified by a multiplier and the associated time curve.

**Input Description:**
- Base motion acceleration flag.
  - 0 Off, base motion not considered.
  - 1 On, base motion considered.
- Base acceleration multiplier in the global X-direction.
- Base acceleration multiplier in the global Y-direction.
- Base acceleration multiplier in the global Z-direction.
- Time curve associated with acceleration in the global X-direction.
- Time curve associated with acceleration in the global Y-direction.
- Time curve associated with acceleration in the global Z-direction.

Example: NL Base, 1, 1, -2, 3, 1, 5, 8,
This command activates the base motion option during nonlinear dynamic analysis. The value of the base acceleration at any time is the value read or interpolated from the associated curve multiplied by the corresponding multiplier. In this example, the time-history of acceleration in the X-direction is 1.0 multiplied by time curve 1, acceleration in the Y-direction is 2.0 multiplied by time curve 5, and acceleration in the Z-direction is 3.0 multiplied by time curve 8.

**NL Integration**

The **NL Integration** command specifies direct time integration parameters for use in nonlinear dynamic analysis.
Input Description:
- Direct time integration method.
  = 0  Newmark-Beta Method.
  = 1  Wilson-Theta Method.
  (default is 0)

- First integration parameter.
  Wilson-theta method: \( \theta \) 
  (default is 1.4)
  Newmark method: \( \gamma \) 
  (default is 0.5)

- Second integration parameter. Beta used in Newmark's Method only.
  (default is 0.25)

Example: NL Integration, 0, 0.5, 0.26
This command specifies the Newmark-Beta method for direct time integration with Beta as 0.26, and Gamma as 0.5.

NL Auto Step

The **NL Auto Step** command is used to activate the automatic stepping option to be used in nonlinear structural analysis (NSTAR). Both static and dynamic options and all control techniques are supported by this command.

<table>
<thead>
<tr>
<th>NL Auto Step</th>
</tr>
</thead>
</table>

Input Description:
- Automatic stepping flag.
  = 0  Off, do not use automatic stepping.
  = 1  On, use automatic stepping.

- Minimum step increment.
- Maximum step increment.
  (Default is the final time defined in the **Times** command for the Force and Displacement Control methods. For the Arc-Length control method, default is equal to the maximum displacement or rotation defined in the Arc-length input, in the **NL Control** command.)

Note:
1. The program stops the analysis if:
   a. The number of step-increment adjustments for any step exceeded 5 trials.
   b. The step increment required for convergence becomes smaller than \( dt_{\text{min}} \). In any case, the user may make changes (tolerance, step size, iteration method, etc.), and use the **Restart** option to continue.
Example: NL Auto Step, 1, 0.1, 0.5
This command activates the automatic stepping option in nonlinear structural analysis and defines a minimum step increment of 0.1, and a maximum step increment of 0.5.

NL Output Opts Submenu
This menu contains commands to control the information in the output and plot files.

<table>
<thead>
<tr>
<th>NL OUTPUT OPTS</th>
</tr>
</thead>
<tbody>
<tr>
<td>NL Nodal Resp</td>
</tr>
<tr>
<td>NL Plot</td>
</tr>
<tr>
<td>NL Print</td>
</tr>
<tr>
<td>Print Node Set</td>
</tr>
<tr>
<td>Print Elem Set</td>
</tr>
<tr>
<td>Strain Out</td>
</tr>
</tbody>
</table>

Figure 5.80  NL Output Opts Submenu

NL Nodal Resp
The NL Nodal Resp command is a preprocessing command that must be issued before running NSTAR to request storing response information at the specified nodes at all solution steps for subsequent xy-plotting. The response information include displacements, velocities, accelerations, reaction forces, and load factors. Load factors are only available for the displacement and arc-length control methods. A maximum of 50 nodes can be selected. The command continues to prompt for more nodes until location 50 is filled, or a null entry for the node number is entered.

Input Description:
- Starting location.
- Node number at the "start" location.
- Node number at the "start+1" location.

Note:
1. The command may be used to change the node number stored in a given location.
**NL Plot**

The **NL Plot** command is a preprocessing command that must be issued before running NSTAR to select the solution steps at which the response at all nodes is to be stored in the database. The response includes deformation and stress in the case of static analysis, and deformation, stress, velocity and acceleration for the dynamic analysis. Five patterns of solution steps may be specified. Note that the **NL Nodal Resp** command may be used to specify nodes for which the response is desired at all solution steps for subsequent xy-plotting. A maximum of 1000 solution steps can be stored.

**Input Description:**
- Starting solution step for the ith pattern.
- Ending solution step for the ith pattern.
- Increment between solution steps for the ith pattern.

**NL Print**

The **NL Print** command controls the output information for the nonlinear structural analysis module (NSTAR).

**Input Description:**
- Flag for displacement printout.
  - $= 0$ No printout.
  - $= N$ Print at every $N$ time step.
- Flag for velocity printout.
  - $= 0$ No printout.
  - $= 1$ Print at solution steps specified for displacements.
- Flag for acceleration printout.
  - $= 0$ No printout.
  - $= 1$ Print at solution steps specified for displacements.
- Flag for stiffness matrix and equation number printout.
  - $= 0$ No printout.
  - $= 1$ Print stiffness matrix in the output file.
- Flag for detailed input printout.
  - $= 0$ No printout.
  - $= 1$ Print nodal coordinates, element connectivity, material and real constant sets, prescribed displacements and applied loads including the effect of surface pressure.
- Overwrite/append flag for analysis output.
  = 0       Overwrite previous output unless restart option is active.
  = 1       Append to previous output.

**Example:**

NL Print, 2, 0, 1;
This command instructs the program to print displacements and accelerations every other solution step, velocities are not to be printed. All other defaults are accepted.

## Print Node Set

The **Print Node Set** command defines groups of nodes for which displacements, velocities, accelerations and temperatures will be written in the output file. Up to 10 groups can be specified.

### **Print Node Set**

**Input Description:**
- Number of groups to be specified.
- Beginning node of group i.
- Ending node of group i. (i=1, 2, ...., 10)

**Example:**

Print Node Set, 3, 1, 35, 40, 45, 140, 150
This command instructs the program to print displacements and accelerations every other solution step, velocities are not to be printed. All other defaults are accepted.

## Print Elem Set

The **Print Elem Set** command defines groups of elements for which stresses will be written in the output file. Up to 10 groups can be specified.

### **Print Elem Set**

**Input Description:**
- Number of groups to be specified.
- Beginning element of group i.
- Ending element of group i. (i=1, 2, ...., 10)

**Example:**

Print Elem Set, 3, 1, 35, 40, 45, 140, 150
This command instructs the program to print stresses for elements 1 through 35, 40 through 45 and 140 through 150 in the output file.
### Strain Out

The **Strain Out** command controls writing of strain values in the output for nonlinear analysis.

<table>
<thead>
<tr>
<th><strong>Input Description:</strong></th>
</tr>
</thead>
<tbody>
<tr>
<td>- Main flag for strain output.</td>
</tr>
<tr>
<td>= 0  Do not write strains. (No more prompts are issued.)</td>
</tr>
<tr>
<td>= 1  Write strain components as specified next.</td>
</tr>
<tr>
<td>- Flag for total strain output.</td>
</tr>
<tr>
<td>= 0  No total strain output.</td>
</tr>
<tr>
<td>= 1  Include total strain output.</td>
</tr>
<tr>
<td>- Flag for thermal strain output.</td>
</tr>
<tr>
<td>= 0  No thermal strain output.</td>
</tr>
<tr>
<td>= 1  Include thermal strain output.</td>
</tr>
<tr>
<td>- Flag for creep strain output.</td>
</tr>
<tr>
<td>= 0  No creep strain output.</td>
</tr>
<tr>
<td>= 1  Include creep strain output.</td>
</tr>
<tr>
<td>- Flag for plastic strain output.</td>
</tr>
<tr>
<td>= 0  No plastic strain output.</td>
</tr>
<tr>
<td>= 1  Include plastic strain output.</td>
</tr>
<tr>
<td>- Flag for principal stretch ratios.</td>
</tr>
<tr>
<td>= 0  No principal strains output.</td>
</tr>
<tr>
<td>= 1  Include principal strains in the output file.</td>
</tr>
</tbody>
</table>

**Note:**

1. At least one type of strain output should be requested if the main strain flag is not set to 0.

**Example:**

```
Strain Out, 1, 1, , ,1
```

This command requests total and plastic strains output for the nonlinear analysis to be performed.

### Run Nonlinear

The **Run Nonlinear** command performs nonlinear structural analysis using the NSTAR module.
**Run Fluid**

The **Run Fluid** command performs fluid flow analysis.

**Print Options Submenu**

Sets the options for information to be printed in the OUT file by the analysis modules. The options are as follows.

<table>
<thead>
<tr>
<th>PRINT OPTIONS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Displacement</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Velocity</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Acceleration</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Mode Shape</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Detailed</td>
</tr>
<tr>
<td>No</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

*Figure 5.81  Print Options Submenu*

**Print Options**

**Option & Description:**

*Displacement*

- Print displacements in the output file.

*No*

- Do not print displacements in the output file.

*Velocity*

- Print velocities in the output file.

*No*

- Do not print velocities in the output file.

*Acceleration*

- Print accelerations in the output file.
No
  Do not print accelerations in the output file.

Mode Shape
  Print mode shapes in the output file.

No
  Do not print mode shapes in the output file.

Detailed
  Print detailed input model information concerning nodes, elements, and loads and boundary conditions.

No
  Suppress the printing of detailed input information.

Accept
  Accept command options.

Quit
  Abandon the changes.

**P-Method Submenu**

Sets the analysis to work with the P-method. Available for 10-noded tetrahedral, 6-node triangles, and 8-node plane elements. The P-Method is only available for linear static analysis.

<table>
<thead>
<tr>
<th>Option</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Activate</td>
<td>Activates P-Method for subsequent linear static analysis.</td>
</tr>
<tr>
<td>Deactivate</td>
<td>Deactivates P-Method analysis for subsequent linear static analysis.</td>
</tr>
<tr>
<td>Quit</td>
<td></td>
</tr>
</tbody>
</table>

**Figure 5.82  P-Method Submenu**
Reaction Force Submenu

Activates or deactivates the calculation of reaction force.

<table>
<thead>
<tr>
<th>REACTION FORCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Calculate</td>
</tr>
<tr>
<td>Skip</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.83  Reaction Force Submenu

Reaction Force

Option & Description:

*Calculate*

Activate reaction force calculation.

*Skip*

Deactivate reaction force calculation.

Restart

Activates or deactivates the continuation of the solution from the last successful time step for nonlinear structural and transient thermal analyses.

Input Description:

- Flag to activate or deactivate the restart option.
  
  = 1  Activate restart option.
  
  = 0  Deactivate restart option.

Example: Suppose you have an NSTAR problem which diverged during solution step number 5, you may change the solution time increment, or other parameters to improve the chances of convergence. If the restart flag is off and the nonlinear is run, then the solution will start all over again from the starting time ignoring the previous results. If the restart flag is on, the solution will continue from the last successful step.
POSTPROCESSING Menu

Figure 5.84  Postprocessing Menu
Figure 5.84 Postprocessing Menu (Continued)
Figure 5.84  Postprocessing Menu  (Continued)
Figure 5.84  Postprocessing Menu  (Continued)

This menu is related to postprocessing of the results for various types of analyses in text and graphic formats.

Analysis Type Submenu

A submenu to activate an analysis type for postprocessing. The last analysis type performed is active by default. You will be prompted according to the active analysis type.
Display Plots Submenu

This menu contains commands related to displaying the corresponding activated result quantity. The quantities available for plotting depend on the active type of analysis which may be changed using the Analysis Type submenu.

Plot Displacement Submenu

Produces a filled, contour, or vector plot for a displacement component. A filled plot fills areas of approximately equal values with the same color. A contour plot connects points of equal values using different colors for different levels. A vector plot consists of vectors whose direction and magnitude represent that of the displacement component. Use Set Plot
to change color set or range of plot. Choose **Filled Contour**, **Line Contour**, or **Vector**, **Deformed** or **Undeformed**, and **Accept** to plot, or **Quit** to return to the Display Plots menu.

![Plot Displacement Submenu](image)

**Figure 5.87 Plot Displacement Submenu**

<table>
<thead>
<tr>
<th>Option &amp; Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Plot Displacement</strong></td>
</tr>
</tbody>
</table>

**Displacement**
- Translation in the X, Y, or Z direction.

**Rotation**
- Rotation about the X, Y, or Z axis.

The following components are available for ASTAR and NSTAR only:

**Velocity**
- Linear velocity in the X, Y, or Z direction.

**Acceleration**
- Linear acceleration in the X, Y, or Z direction.

**Angular Vel**
- Angular velocity about the X, Y, or Z axis.

**Angular Accel**
- Angular acceleration about the X, Y, or Z axis.
Reaction Force
   Reaction force in the X, Y, or Z direction.

Reaction Moment
   Reaction moment about the X, Y, or Z axis.

X direction
   Activate the X component of the response quantity selected.

Y direction
   Activate the Y component of the response quantity selected.

Z direction
   Activate the Z component of the response quantity selected.

Resultant
   Resultant.

Filled Contour
   Generate a color-filled plot.

Line Contour
   Generate a contour plot.

Vector
   Generate a vector plot.

Deformed
   Use deformed shape for plot.

Undeformed
   Use undeformed shape for plot.

Once you select Accept, you will be prompted to input the load case, mode shape, or solution step.

Input Description:
- Load case, mode shape, or solution step number.
- Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor [shown between brackets], scales the maximum resultant deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 results in using the same scale for both model and deformation assuming there is no isolated remote nodes or keypoints in the model. Prompted only if Deformed is selected.

Notes:
1. Only one plot can be stored in the plot buffer. The most recently loaded component replaces the other component in the buffer, if any, and becomes the active one.
2. The active type of analysis can be changed using Analysis Type.
Plot Stress Submenu

Produces a filled, contour, or vector plot for a stress component. A filled plot fills areas of approximately equal values with the same color. A contour plot connects points of equal values using different colors for different levels. A vector plot consists of vectors whose direction and magnitude represent that of the stress component. Use Set Plot to change color set or range of plot. Choose Filled Contour, Line Contour, or Vector, Deformed or Undeformed, and Accept to plot, or Quit to return to the Display Plots submenu.

<table>
<thead>
<tr>
<th>PLOT STRESS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Stress X</td>
</tr>
<tr>
<td>Stress Y</td>
</tr>
<tr>
<td>Stress Z</td>
</tr>
<tr>
<td>Shear Stress XY</td>
</tr>
<tr>
<td>Shear Stress XZ</td>
</tr>
<tr>
<td>Shear Stress YZ</td>
</tr>
<tr>
<td>Principal 1</td>
</tr>
<tr>
<td>Principal 2</td>
</tr>
<tr>
<td>Principal 3</td>
</tr>
<tr>
<td>Von Mises</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Nodal Element</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Face</td>
</tr>
<tr>
<td>Bottom Face</td>
</tr>
<tr>
<td>Membrane Stresses</td>
</tr>
<tr>
<td>Bending Stresses</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Filled Contour Line Contour Vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>Deformed</td>
</tr>
<tr>
<td>Undeformed</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Accept</th>
</tr>
</thead>
<tbody>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.88  Plot Stress Submenu

Plot Stress

Option & Description:

Stress X
Normal stress in the X direction.

Stress Y
Normal stress in the Y direction.

Stress Z
Normal stress in the Z direction.
Shear Stress XY
Shear stress in the Y direction, in the plane normal to the X axis.

Shear Stress XZ
Shear stress in the Z direction, in the plane normal to the X axis.

Shear Stress YZ
Shear stress in the Z direction, in the plane normal to the Y axis.

Principal 1
Maximum principal stress.

Principal 2
Intermediate principal stress.

Principal 3
Minimum principal stress.

Von Mises
von Mises stress.

Nodal
Stresses at nodes.

Element
Stresses at centers of elements.

The following options are only available for shell elements. Refer to the Element Library chapter for definition of top and bottom faces for shell elements (top: face 5, bottom: face 6).

Top Face
Membrane and bending stresses at top face.

Bottom Face
Membrane and bending stresses at bottom face.

Membrane Stresses
Membrane stresses only.

Bending Stresses
Bending stresses only.

Filled Contour
Generate a color-filled plot.

Line Contour
Generate a contour plot.

Vector
Generate a vector plot.
Deformed
Use deformed shape for plot.

Undeformed
Use undeformed shape for plot.

Once you select Accept, you will be prompted to input the load case or solution step.

Input Description:
- Load case or solution step number.
- Scale factor for deformation plotting. Refer to Plot Displacement for details.

Notes:
1. Only one plot can be stored in the plot buffer. The most recently loaded component replaces the other component in the buffer, if any, and becomes the active one.
2. The von Mises stress is computed from the basic stress components as follows:
   \[ VON = \left(\frac{1}{2}\right)\left\{ \left(\frac{1}{2}\right) \left[ (SX - SY)^2 + (SX - SZ)^2 + (SY - SZ)^2 \right] + 3(TXY^2 + TXZ^2 + TYZ^2) \right\}^{1/2} \]
or equivalently, from the principal stresses as follows:
   \[ VON = \left(\frac{1}{2}\right)\left\{ \left(\frac{1}{2}\right) \left[ (P1 - P2)^2 + (P1 - P3)^2 + (P2 - P3)^2 \right] \right\}^{1/2} \]
where:
   - SX  = stress x
   - SY  = stress y
   - SZ  = stress z
   - TXY = shear stress xy
   - TXZ = shear stress xz
   - TYZ = shear stress yz

Plot Thermal Submenu

Produces a filled, contour, or vector plot for a thermal component. A filled plot fills areas of approximately equal values with the same color. A contour plot connects points of equal values using different colors for different levels. A vector plot consists of vectors whose direction and magnitude represent that of the thermal component. Use Set Plot to change color set or range of plot. Choose Filled Contour, Line Contour, or Vector and Accept to plot, or Quit to return to the Display Plots submenu.
Figure 5.89  Plot Thermal Submenu

<table>
<thead>
<tr>
<th>PLOT THERMAL</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Grad X</td>
</tr>
<tr>
<td>Grad Y</td>
</tr>
<tr>
<td>Grad Z</td>
</tr>
<tr>
<td>Grad N</td>
</tr>
<tr>
<td>Hflux X</td>
</tr>
<tr>
<td>Hflux Y</td>
</tr>
<tr>
<td>Hflux Z</td>
</tr>
<tr>
<td>Hflux N</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Filled Contour Line Contour Vector</th>
</tr>
</thead>
<tbody>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

**Plot Thermal**

**Option & Description:**

*Temperature*

Nodal temperature.

*Grad X*

Temperature gradient in the X-direction.

*Grad Y*

Temperature gradient in the Y-direction.

*Grad Z*

Temperature gradient in the Z-direction.

*Grad N*

Resultant temperature gradient.

*Hflux X*

Heat flux in the X-direction.

*Hflux Y*

Heat flux in the Y-direction.

*Hflux Z*

Heat flux in the Z-direction.

*Hflux N*

Resultant heat flux.
Filled Contour
Generate a color-filled plot.

Line Contour
Generate a contour plot.

Vector
Generate a vector plot.

Once you select Accept, you will be prompted to input the load case or solution step.

Input Description:
- Load case or solution step number.

Note:
1. Only one plot can be stored in the buffer. The most recently loaded component replaces the existing component in the buffer, if any, and becomes the active one.

Plot Fluid Submenu

Produces a filled, contour, or vector plot for a fluid component. Choose Filled Contour, Line Contour, or Vector and Accept to plot, or Quit to return to the Display Plots submenu.

<table>
<thead>
<tr>
<th>PLOT FLUID</th>
</tr>
</thead>
<tbody>
<tr>
<td>VRES</td>
</tr>
<tr>
<td>VX</td>
</tr>
<tr>
<td>VY</td>
</tr>
<tr>
<td>VZ</td>
</tr>
<tr>
<td>Stream</td>
</tr>
<tr>
<td>Pressure</td>
</tr>
<tr>
<td>Tau XY</td>
</tr>
<tr>
<td>Tau YZ</td>
</tr>
<tr>
<td>Tau ZX</td>
</tr>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Grad X</td>
</tr>
<tr>
<td>Grad Y</td>
</tr>
<tr>
<td>Grad Z</td>
</tr>
<tr>
<td>Grad N</td>
</tr>
<tr>
<td>TKE</td>
</tr>
<tr>
<td>EPS</td>
</tr>
<tr>
<td>Density</td>
</tr>
<tr>
<td>Mach Number</td>
</tr>
<tr>
<td>Filled Contour</td>
</tr>
<tr>
<td>Line Contour</td>
</tr>
<tr>
<td>Vector</td>
</tr>
</tbody>
</table>

Accept
Quit

Figure 5.90  Plot Fluid Submenu
Plot Fluid

Option & Description:

- **VRES**
  - Resultant velocity.

- **VX**
  - Velocity in X-direction.

- **VY**
  - Velocity in Y-direction.

- **VZ**
  - Velocity in Z-direction.

- **Stream**
  - Stream function.

- **Pressure**
  - Pressure

- **Tau XY**
  - Shear stress in the X-direction in XZ plane.

- **Tau YZ**
  - Shear stress in the Y-direction in YX plane.

- **Tau ZX**
  - Shear stress in the Z-direction in ZY plane.

- **Grad X**
  - Temperature gradient in X-direction.

- **Grad Y**
  - Temperature gradient in Y-direction.

- **Grad Z**
  - Temperature gradient in Z-direction.

- **Grad N**
  - Resultant temperature gradient.

- **TKE**
  - Turbulence kinetic energy.

- **EPS**
  - Dissipation rate.

- **Density**
  - Fluid density.
Mach Number
   Mach number.

Filled Contour
   Generate a color filled plot.

Line Contour
   Generate a contour plot.

Vector
   Generate a vector plot.

Once you select Accept, you will be prompted to input the time step number.

Input Description:
- Time step number.

Plot Strain Submenu

Produces a filled contour plot or vector plot for a strain component. A filled plot fills areas of approximately equal values with the same color. A contour plot connects points of equal values using different colors for different levels. A vector plot consists of vectors whose direction and magnitude represent that of the strain component. Use Set Plot to change color set or range of plot. Choose Filled Contour, Line Contour, or Vector, Deformed or Undeformed, and Accept to plot, or Quit to return to the Display Plots submenu.
Plot Strain

Option & Description:

**Strain X**
Normal element strain in the X-direction.

**Strain Y**
Normal element strain in the Y-direction.

**Strain Z**
Normal element strain in the Z-direction.

**Shear Strain XY**
Change in angle between lines initially parallel to X and Y axes.

**Shear Strain XZ**
Change in angle between lines initially parallel to X and Z axes.

**Shear Strain YZ**
Change in angle between lines initially parallel to Y and Z axes.

**Strain type (for nonlinear analysis only).** For linear static analysis total strain is always activated.

**Total**
Total strain.

**Thermal**
Thermal strain.

**Creep**
Creep strain.

**Plastic**
Plastic strain.

**Top Face**
Membrane and bending strain at top face.

**Bottom Face**
Membrane and bending strain at bottom face.

**Membrane Strain**
Membrane strain only.

**Bending Strain**
Bending strain at top face.

**Filled Contour**
Generate a color-filled plot.

**Line Contour**
Generate a contour plot.
Vector
Generate a vector plot.

Deformed
Use deformed shape for plot.

Undeformed
Use undeformed shape for plot.

Once you select Accept, you will be prompted to input the load case or solution step.

Input Description:
- Load case or solution step number.
- Scale factor for deformation plotting. Refer to Plot Displacement for details.

Notes:
1. Only one plot can be stored in the plot buffer. The most recently loaded component replaces the other component in the buffer, if any, and becomes the active one.
2. The equivalent strain (ESTRN) is calculated from:
   \[
   ESTRN = 2\left[\frac{\varepsilon_1 + \varepsilon_2}{3}\right]^{1/2}
   \]
   where:
   \[
   \varepsilon_1 = 0.5[(EPSX - \varepsilon^*)^2 + (EPSY - \varepsilon^*)^2 + (EPSZ - \varepsilon^*)^2]
   \]
   \[
   \varepsilon_2 = \left[\frac{(GMXY)^2 + (GMXZ)^2 + (GMYZ)^2}{4}\right]
   \]
   \[
   \varepsilon^* = \frac{(EPSX + EPSY + EPSZ)}{3}
   \]
   where:
   EPSX = strain x
   EPSY = strain y
   EPSZ = strain z
   GMXY = shear strain xy
   GMXZ = shear strain xz
   GMYZ = shear strain yz

Deformation
Plots the specified deformed shape corresponding to a load case, time step or mode shape, depending on the active type of analysis.

Input Description:
- Load case, time step, or mode shape number.
- Scale factor for deformation plotting. Refer to Plot Displacement for details.
Plot Sections
Generates a section plot for the active quantity in the plot buffer for 3-D models. The section is defined by defining a cutting plane.

### Plot Sections

**Input Description:**
- **Cutting plane.**
  - = 0  Plane normal to the global X-axis.
  - = 1  Plane normal to the global Y-axis.
  - = 2  Plane normal to the global Z-axis.
  - = 3  Plane defined by 3 nodes.

- Number of sections to be plotted. Prompted only if a global plane is used.

- Locations of the section. Not prompted if plane is defined by nodes.
  - = 1  Specify the locations of sections.
  - = 2  Use default locations at equal intervals within model extents.
    (default is 2)

- Specify locations of sections if the cutting plane is normal to X, Y, or Z. If cutting plane is normal to, for example, enter X values at desired locations. Not prompted if plane is defined by 3 nodes.

- Shape of model to be used for plotting.
  - = 0  Undeformed model shape.
  - = 1  Deformed model shape.
    (default is 0)

- Scale factor for deformation plotting. The given factor is a multiplier of actual deformation. The default factor [shown between brackets], scales the maximum resultant deformation to 10% of the model extents (largest dimension in the model). A scale factor of 1.0 results in using the same scale for both model and deformation assuming there is no isolated remote nodes or keypoints in the model.

Plot Iso Surfaces
Plots isosurfaces for the active quantity in the plot buffer for 3-D models. A similar isolines plot may be obtained for 2-D models using the **Contour** option in **Plot Displacement**, **Plot Stress**, etc.

### Plot Iso Surfaces

**Input Description:**
- Number of isoplanes to be plotted.

- Flag for the choice of isoplanes.
  - = 0  Take default values at equal intervals of data values.
  - = 1  Specify the values for isosurfaces.
    (default is 0)
- Specify values for isosurfaces. Not prompted if default above is used.

- Shape of model to be used for plotting.
  - 0 Undeformed model shape.
  - 1 Deformed model shape.
  (default is 0)

- Scale factor for deformation plotting. Refer to Plot Displacement for details.

Animate

The Animate command results in the animation of the current plot in the active window. If no plot exists on the screen, the deformed shape will be animated for STAR, ASTAR, and NSTAR, and mode shape in the case of DSTAR (buckling or frequency). For other types of analyses, one of the following plots must exist in the active window for the Animate command to work.

1. A contour plot for any quantity
2. A section plot
3. An isosurface plot

The animation represents snap shots of plotted data versus time or frequency. If results are only available for one step (linear static analysis, and steady state analyses), results are linearly interpolated for several frames. Animation should be performed in the following sequence for deformed shapes:

a. If no post-processing plot is in the window, you may directly give the Animate command for deformed/mode shape animation.

b. You may plot some other quantity on the deformed shape (a stress component for example).

c. You may use the Animate command to animate the deformed shape. If a quantity is plotted on top of the deformed shape, then the quantity itself as well as the deformed shape are linearly interpolated for the load case/mode shape specified.

For animation of other data, use the following sequence:

a. Activate the component of your choice for the active analysis type using commands like Plot Stress.

b. You may use Plot Setting to change default setting.

c. Plot the activated data using the corresponding command like Plot Stress. For NSTAR and ASTAR, stress plots may be generated on top of the corresponding deformed shape.

d. You may use Plot Sections to generate a section plot for 3-D models.

e. Finally, use Animate to animate the plot on the screen. Animation is memory-demanding, use a small window for animation to save memory.
In cases where results are available for several time or frequency steps, the user will be prompted for a pattern of time steps to be used as animation frames. A maximum of 20 frames can be used for animation.

### Animate

**Input Description:**
- Beginning solution step, load case, or mode shape number.

*The following 3 arguments are not prompted if the active analysis type for postprocessing is linear static, frequency, buckling or other steady state analyses.*
- Beginning solution step number.
- Ending solution step number.
- Solution step increment.
- Flag to specify one or two way animation.
  - \( = 2 \) Two-way animation. Frames are animated back and forth.
  - \( = 1 \) One-way animation. Frames are animated in cyclic order (with a jump).
    (default is 1)
- Slow_down_factor to reduce speed of animation.
  - \( = 0 \) Fast animation, no slow down.
  - \( = n \) Where \( n \) is a number greater than 0 to slow down the animation. The higher the number, the slower the animation will be.
- Scale factor for deformation plotting. Refer to **Plot Displacement** for details.

**Notes:**
1. A fault memory message is issued when the memory requirement for the specified window exceeds the available memory. The user may then define a smaller window for animation.
2. In animating single step, steady state, or static analysis data, note the following:
   a. If an Iso Planes plot or planar section plot exists on the undeformed shape, animation displays the iso-surfaces or planar sections one by one.
   b. If a quantity is plotted on the deformed shape, the deformed shape is animated, but the quantity itself will assume original fixed levels.

### Dynamic View Submenu

This submenu provides for the interactive rotation, scaling and translation of models in 3D space. The command can operate in demo or non-demo modes. If the demo mode is selected, the user has no control over the dynamic viewing. If the non-demo mode is selected, the user may use the trackball mode or the coordinate axis mode.
Dynamic View

Option & Description:

Blue, Green, Cyan, Red, Magenta, Yellow, Grey Shade
Set shade color.

Blue, Green, Cyan, Red, Magenta, Yellow, Grey Background
Set background color.

Demo off
Go to user mode.

Demo on
Show demo.

Note:

1. This command is currently only supported on SGI Indigo, Personal Iris (both require 24-bit plane and Z-buffer) and HP700 series with CRX graphics board.

Config Dview

Sets the graphical attributes and some functional parameters for Dynamic View. The defaults shown are those when the command is first used. The defaults used for subsequent use of the command will be the values entered by the user.
**Config Dview**

**Input Description:**
- Trackball mode.
  
  = 0  Off, use Coordinate Axis Mode.
  
  = 1  On, use Trackball Mode.
  (default is 1)

- Mouse movement sensitivity (1-100).
  (default is 10)

- Flip surface normals.
  
  = 0  No
  
  = 1  Yes
  (default is 0)

- Plot iso or section planes directly in 3D. This flag starts Dynamic View whenever Section or Iso Plots are started.
  
  = 0  No
  
  = 1  Yes
  (default is 0)

- Plot a coordinate system in the center of the model.
  
  = 0  No
  
  = 1  Yes
  (default is 0)

- Plot a bounding box around the model.
  
  = 0  No
  
  = 1  Yes
  (default is 0)

- Backface elimination.
  
  = 0  Off
  
  = 1  On
  (default is 0)

**Trackball mode**

The left mouse button controls the motion of the virtual trackball in which the model is embedded. The middle mouse button zooms out of the model and the right mouse button zooms into the model. Translations are not possible in this mode.

**Coordinate Axis Mode**

The left, middle and right mouse buttons control the rotation about the X-, Y- and Z-axes respectively. A single press of one of these buttons causes a 10 degree rotation of the model about the corresponding axis. By pressing a button and moving the mouse, continuous rotation will occur with a magnitude to the distance between the two mouse move events (projected onto a horizontal direction). Moving in different directions will cause clockwise or counterclockwise rotation.
While holding the shift key, a single press of the left mouse button will move the model one step further away on the screen. Continuous zooming is possible by holding down the mouse button and translating the mouse horizontally on the screen. The use of the middle mouse button with the shift key depressed works in a similar way but moves the object closer. Holding the shift key and pressing the left mouse button permits the user to move the model on the screen. The direction of motion is in the same direction as the mouse motion.

There are some special keys to be used with dynamic viewing. Some work as toggle keys setting their corresponding functions on or off.

**Identify Result**

Identifies the value of the plotted quantity near the node or element (depending on the plot) closest to the point picked by the mouse.

*Identify Result  (pick a point on the plot using the mouse)*

**Plot Setting**

A command to specify plot settings to be used in subsequent plots. You may specify the color set, range of interest, and the labeling of line contour plots.

*Plot Setting*

*Input Description:*
- Color set number (1 to 7).
  - = 1  6 levels
  - = 2  8 levels
  - = 3  10 levels
  - = 4  12 levels
  - = 5  40 levels
  - = 6  64 levels
  - = 7  76 levels

- Scale factor to be multiplied by all values of the component to be plotted. (default is 1.0)

- Minimum value to be used in the plot. (default is min. value)

- Maximum value to be used in the plot. (default is max. value)

- Approximate distance along the contour lines between labels. Used only for line contour plots.
  - = 0  Do not write contour labels.
  - = X  Write contour labels at x intervals. (default is 0)
- Draw chart flag.
  \[= 0\] Do not draw chart.
  \[= 1\] Draw chart.
  (default is 1)

- Relative horizontal location of chart in the window should be between 0.0 and 1.0.
  \[= 0.0\] Extreme left edge.
  \[= 1.0\] Extreme right edge.

- Relative vertical location of chart in the window should be between 0.0 and 1.0.
  \[= 0.0\] Extreme lower edge.
  \[= 1.0\] Extreme upper edge.

- Title of the plot (not title of chart) can be changed from default title.
  \[= 0\] Use default or existing title.
  \[= 1\] Use new title.

*If new title is selected, then a new title can be input as a sequence of words up to a total of 80 characters. Null string terminates the input.*

**Notes:**

1. Parameters in this command assume their original default values whenever a new component is activated.

2. The scale factor can be used for conversion of units. As an example, a scale factor of 2.54 may be used to display displacements in millimeters instead of inches.

**Boundary Opts Submenu**

This command specifies how a model's boundary is to be plotted. A submenu will pop up for the user to specify the options. The submenu is organized into three sections. The first section allows the user to specify *No Boundary, Mesh Boundary, or Model Boundary* edge evaluation. The second section specifies how element edges are evaluated based on the selection of the first section. The choices are no evaluation, evaluation of element edges on the model surfaces, or on the model boundary edge.
Boundary Opts

Option & Description:

No Boundary
No boundary evaluation.

Mesh Boundary
Mesh boundary evaluation.

Model Boundary
Model boundary evaluation.

No Evaluation
No evaluation.

Evaluate Face
Evaluate Face.

Evaluate Edge
Evaluate Edge.

Clear
Clears the window.

Clear

Repaint
Repaints the screen.

Repaint

Lists Submenu
A menu to list and find the extreme values of results of various types of analyses.
List Displacement Submenu

Used with STAR, DSTAR, NSTAR, and ASTAR to list displacements, velocities, and accelerations. Choose a set and Accept to execute the command, or Quit to abandon it.

**Option & Description:**

*For linear static analysis (STAR):*
- Set 1 List translations and rotations.

*For nonlinear analysis (NSTAR), and post-dynamic analysis (ASTAR):*
- Set 1 List displacements and rotations.
- Set 2 List linear and angular velocities.
- Set 3 List linear and angular accelerations.

*For frequency and buckling analyses (DSTAR):*
- Set 1 List mode shape.

**Input Description:**
- Load case, mode shape, or solution step number.
  (default is active load case, mode shape, or solution step)
- Coordinate system.
  (default is 0)
Note:
1. The type of analysis can be specified by **Analysis Type**.

### List Stress Submenu

A submenu to list nodal or element stresses. Choose **Nodal**, or **Element**, set number, and **Accept** to execute the command, or **Quit** to abandon it. Then you will be prompted as follows.

<table>
<thead>
<tr>
<th>List Stress</th>
<th>Option &amp; Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Nodal</strong></td>
<td>List nodal stresses.</td>
</tr>
<tr>
<td><strong>Element</strong></td>
<td>List element stresses.</td>
</tr>
<tr>
<td><strong>Set 1</strong></td>
<td>List normal and shear stresses.</td>
</tr>
<tr>
<td><strong>Set 2</strong></td>
<td>List principal stresses, von Mises, stress intensity, and error.</td>
</tr>
<tr>
<td><strong>Top Face</strong></td>
<td>List stresses for top face.</td>
</tr>
<tr>
<td><strong>Bottom Face</strong></td>
<td>List stresses for bottom face.</td>
</tr>
<tr>
<td><strong>Membrane</strong></td>
<td>List membrane stresses.</td>
</tr>
</tbody>
</table>

**Figure 5.96 List Stress Submenu**


**Bending**

List bending stresses.

**Input Description:**
- Load case or solution step number.
  (default is active load case or solution step)
- Coordinate system. Only prompted if the stresses were requested in the global coordinate system for all elements in the preprocessing stage.
  (default is 0)

**Notes:**

1. The type of analysis can be specified by **Analysis Type**.
2. The stresses for 2D and 3D elements are listed either in the Cartesian global coordinate system, or in the ECS (element coordinate system), as specified in the option in the element group definition. Stresses for 1D elements are always listed in the local element system.

**List Strain Submenu**

A submenu to list element strains. Choose one option and **Accept** to execute the command, or **Quit** to abandon it. You will be prompted as follows.

<table>
<thead>
<tr>
<th>LIST STRAIN</th>
</tr>
</thead>
<tbody>
<tr>
<td>Top Face</td>
</tr>
<tr>
<td>Bottom Face</td>
</tr>
<tr>
<td>Membrane Strain</td>
</tr>
<tr>
<td>Bending Strain</td>
</tr>
<tr>
<td>Total</td>
</tr>
<tr>
<td>Thermal</td>
</tr>
<tr>
<td>Creep</td>
</tr>
<tr>
<td>Plastic</td>
</tr>
<tr>
<td>Equivalent</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

---

**List Strain**

**Option & Description:**

*Top Face*

List strains at top face (for shell elements only).
**Bottom Face**
List strains at bottom face (for shell elements only).

**Membrane Strain**
List membrane strain only (for shell elements only).

**Bending Strain**
List bending strain only (for shell elements only).

**Total**
List total strain.

**Thermal**
List thermal strain (NSTAR only).

**Creep**
Plastic strain (NSTAR only).

**Equivalent**
Principal strains for hyperelastic models only.

**Input Description:**
- Load case or solution step number.
  (default is active load case or solution step)
- Layer number (for linear analysis only).
  (default is 1)

**List Freq/Buckling**
Lists natural frequencies or buckling load factors.

**List Thermal**
Lists nodal temperature, temperature gradients, and heat flux results for thermal analysis.

**Input Description:**
- Solution step number.
  (default is solution step)
- Listing set.
  = 1 Lists temperature and gradients.
  = 2 Lists heat flux components and resultant.
  (default is 1)
Extremes Submenu

A submenu to list the extreme values of a specified component of response, stress, strain, or temperature. Values within a specified percentage of the extreme values are also listed. All listing is done for the active type of analysis. Choose an item and respond to prompts.

<table>
<thead>
<tr>
<th>Extremes</th>
</tr>
</thead>
<tbody>
<tr>
<td>Temperature</td>
</tr>
<tr>
<td>Displacement</td>
</tr>
<tr>
<td>Stress</td>
</tr>
<tr>
<td>Strain</td>
</tr>
<tr>
<td>PD Extremes</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

![Extremes Submenu](image)

**Figure 5.98 Extremes Submenu**

**Option & Description:**

*Temperature*  
List extreme results from thermal analysis.

*Displacement*  
List extreme response for structural analyses.

*Stress*  
List extreme stresses for structural analyses.

*Strain*  
List extreme strains for structural analyses.

*PD Extremes*  
List extreme response for post-dynamic analysis.

**Input Description:**

- Load case or solution step number. (Mode shape used with displacement components only.)
  
  (default is active load case, mode shape, or solution step)

  *Enter 0 to check all load cases, mode shapes, or solution steps.*

  The component depends on the selected quantity as follows:

  a. For Thermal results:
     
     = TEMP  
     Nodal temperature.

     = GRADX  
     Temperature gradient in the x-direction.

     = GRADY  
     Temperature gradient in the y-direction.

     = GRADZ  
     Temperature gradient in the z-direction.

     = GRADN  
     Resultant temperature gradient.

     = HFLUXX  
     Heat flux in the x-direction.
= HFLUXY Heat flux in the y-direction.
= HFLUXZ Heat flux in the z-direction.
= HFLUXN Resultant heat flux.
(default is TEMP)

b. For Displacements:
Displacement component to be listed.
= UX Displacement in the x-direction.
= UY Displacement in the y-direction.
= UZ Displacement in the z-direction.
= RX Rotation about the x-direction.
= RY Rotation about the y-direction.
= RZ Rotation about the z-direction.
= URES Resultant displacement.
(default is UX)

The following components are available for ASTAR and NSTAR only:
= VX Velocity in the x-direction.
= VY Velocity in the y-direction.
= VZ Velocity in the z-direction.
= WX Angular velocity about x-direction.
= WY Angular velocity about y-direction.
= WZ Angular velocity about z-direction.
= VRES Resultant velocity.
= AX Acceleration in the x-direction.
= AY Acceleration in the y-direction.
= AZ Acceleration in the z-direction.
= BX Angular acceleration about x-direction.
= BY Angular acceleration about y-direction.
= BZ Angular acceleration about z-direction.
= ARES Resultant acceleration.
(default is VX)

c. For Stress:
= SX Normal stress in the x-direction.
= SY Normal stress in the y-direction.
= SZ Normal stress in the z-direction.
= TXY Shear stress in the y direction in the plane normal to the x axis.
= TXZ Shear stress in the z direction in the plane normal to the x axis.
= TYZ Shear stress in the z direction in the plane normal to the y axis.
= P1 Normal stress in the principal x-axis direction.
= P2 Normal stress in the principal y-axis direction.
= P3 Normal stress in the principal z-axis direction.
= VON von Mises stress.
= INT Stress intensity.
= ERR Stress error (not available for NSTAR).
(default is VON)

d. For Strain:
= EPSX Normal strain in the x-direction.
= EPSY Normal strain in the y-direction.
= EPSZ Normal strain in the z-direction.
= GMXY Shear strain in the x-y plane.
= GMXZ Shear strain in the x-z plane.
= GMYZ Shear strain in the y-z plane.
= ESTRN  Equivalent shear strain. (Refer to the Plot Strain command, note #2)
= SED    Strain energy density.
= ENERGY Total strain energy.  
(default is ESTRN)

- Percentage tolerance in listing. Values within the extreme value are also listed.  
(default is 5)

- Criterion for the listing.  
  = 0    Absolute maximum.  
  = 1    Algebraic maximum.  
  = 2    Algebraic minimum.  
  (default is 0)

- Sorting flag.  
  = 0    Unsorted list (sequential listing).  
  = 1    Sorted list.  
  (default is 1)

**PD Extremes**

Searches ASTAR results for extreme response values for a given set of nodes. The highest 10 values nodes will be listed from a set of 300. This command is a postprocessing command that may only be issued after post dynamic analysis has been performed. The entire range of solution will be searched.

**PD Extremes**

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Flag defining type of maximums.</td>
</tr>
<tr>
<td>= 1    Displacement</td>
</tr>
<tr>
<td>= 2    Velocity</td>
</tr>
<tr>
<td>= 3    Acceleration</td>
</tr>
<tr>
<td>(default is 1)</td>
</tr>
</tbody>
</table>

| - Flag defining direction of extreme values.  |
|  = 1    X-translation  |
|  = 2    Y-translation  |
|  = 3    Z-translation  |
|  = 4    X-rotation  |
|  = 5    Y-rotation  |
|  = 6    Z-rotation  |
|  (default is 1)  |

- Node_1  
  (default is 1)

*Extremes will be searched among nodes starting from node_1 and ending at node_1 + 300.*

**Note:**  
Time-history analysis; Response based on time.
Random Vibration; Power spectral densities based on the exciting frequency.
Frequency Response; Amplitude of harmonic motion based on the exciting frequency.
In the case of random vibration or frequency response, the frequency range limits are assumed to have the same units as the excitation curves.

**Graphs Submenu**

This submenu is related to graphing input curves and analysis results.

<table>
<thead>
<tr>
<th>GRAPHS</th>
</tr>
</thead>
<tbody>
<tr>
<td>Initialize Graph</td>
</tr>
<tr>
<td>Act Pre X-Y</td>
</tr>
<tr>
<td>Act Post X-Y</td>
</tr>
<tr>
<td>Set Graph</td>
</tr>
<tr>
<td>Graph Range</td>
</tr>
<tr>
<td>Reference Line</td>
</tr>
<tr>
<td>Identify</td>
</tr>
<tr>
<td>List Graph</td>
</tr>
<tr>
<td>List Point</td>
</tr>
<tr>
<td>Plot Graph</td>
</tr>
</tbody>
</table>

**Figure 5.99** Graphs Submenu

**Initialize Graph**

Clears all previously activated xy-plots from the xy-plot buffer. The initialization includes graphs activated using **Act Pre X-Y**, and **Act Post X-Y**.

**Act Pre X-Y**

Loads data generated by the preprocessor (like time, temperature, and post dynamic curves) into the X-Y plot buffer. This command will initialize any graphs previously activated by **Act Post X-Y**.

**Input Description:**
- Number of the graph to be activated. Up to 6 graphs can be activated simultaneously.
  (default is highest defined + 1)
- Type of graph to be activated.
  = 0 Time, temperature, or MPC (material property curve).
  = 1 Post-dynamic curves.
  (default is 0)

- Component to be activated. Valid components depend on the type of graph selected above.
  For graphtype 0:
  = TIME Time curve.
  = TEMP Temperature curve.
  = MPC Material property curve.
  For graphtype 1:
  = TIME Postdynamic time curve.
  = FREQ Postdynamic frequency curve.

- Curve label.

- Color to be used for plotting.
  (default is 16)

- Line style to plot graph.
  = 0 Solid line.
  = 1 Dotted line.
  (default is 0)

- Symbol type to be plotted at data points.
  = 0 No symbol
  = 1 Circle
  = 2 Filled circle
  = 3 Square
  = 4 Filled square
  = 5 Upward triangle
  = 6 Filled upward triangle
  = 7 Downward triangle
  = 8 Filled downward triangle
  = 9 Star
  = 10 Filled star
  (default is 1)

**Act Post X-Y**

Loads vectors to be used in time-history like plots from the database to the plot buffer. The graph information is read from an existing file that was prepared by one of the analysis modules. Each graph is assigned a number. Up to 6 graphs can be loaded in one plot. This command will initialize any graphs previously activated by **Act Pre X-Y**.

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>- Graph number (1 to 6).</td>
</tr>
<tr>
<td>(default is highest defined + 1)</td>
</tr>
</tbody>
</table>
- Name of variable in the X-direction.
  = TIME  Time.
  = FREQ  Frequency.
  (default is TIME)
The following entries are valid only for NSTAR for load versus displacement graphs.
  = UX  Displacement in the X-direction.
  = UY  Displacement in the Y-direction.
  = UZ  Displacement in the Z-direction.
  = RX  Rotation about the X-axis.
  = RY  Rotation about the Y-axis.
  = RZ  Rotation about the Z-axis.
  = URES  Resultant displacement.
- Name of variable in the Y-direction. Valid entries depend on the active analysis type (refer to Analysis Type).
  a) For postdynamic analysis (ASTAR):
    = UX  Displacement in the x-direction.
    = UY  Displacement in the y-direction.
    = UZ  Displacement in the z-direction.
    = RX  Rotation about the x-direction.
    = RY  Rotation about the y-direction.
    = RZ  Rotation about the z-direction.
    = URES  Resultant displacement.
    = VX  Velocity in the x-direction.
    = VY  Velocity in the y-direction.
    = VZ  Velocity in the z-direction.
    = WX  Angular velocity about x-direction.
    = WY  Angular velocity about y-direction.
    = WZ  Angular velocity about z-direction.
    = VRES  Resultant velocity.
    = AX  Acceleration in the x-direction.
    = AY  Acceleration in the y-direction.
    = AZ  Acceleration in the z-direction.
    = BX  Angular acceleration about x-direction.
    = BY  Angular acceleration about y-direction.
    = BZ  Angular acceleration about z-direction.
    = ARE S  Resultant acceleration.
    = FORCE  Force.
    = MOMENT  Moment.
    = STRESS  Stress for 2-D and 3-D elements.
    (default is UX)
  b) For nonlinear structural analysis (NSTAR):
    = UX  Displacement in the x-direction.
    = UY  Displacement in the y-direction.
    = UZ  Displacement in the z-direction.
    = RX  Rotation about the x-direction.
    = RY  Rotation about the y-direction.
    = RZ  Rotation about the z-direction.
    = URES  Resultant displacement.
    = VX  Velocity in the x-direction.
    = VY  Velocity in the y-direction.
    = VZ  Velocity in the z-direction.
= WX  Angular velocity about x-direction.
= WY  Angular velocity about y-direction.
= WZ  Angular velocity about z-direction.
= VRES Resultant velocity.
= AX  Acceleration in the x-direction.
= AY  Acceleration in the y-direction.
= AZ  Acceleration in the z-direction.
= BX  Angular acceleration about x-direction.
= BY  Angular acceleration about y-direction.
= BZ  Angular acceleration about z-direction.
= ARES Resultant acceleration.
= LFACT Load factor.
= RFX Reaction force in the x-direction.
= RFY Reaction force in the y-direction.
= RFZ Reaction force in the z-direction.
= RFRES Resultant reaction force.
= RMX Reaction moment about the x-direction.
= RMY Reaction moment about the y-direction.
= RMZ Reaction moment about the z-direction.
= RMRES Resultant reaction moment.
    (default is UX)

c) For thermal analysis (HSTAR):
= TEMP Nodal temperature.
= GRADX Temperature gradient in the x-direction.
= GRADY Temperature gradient in the y-direction.
= GRADZ Temperature gradient in the z-direction.
= GRADN Resultant temperature gradient.
= HFLUXX Heat flux in the x-direction.
= HFLUXY Heat flux in the y-direction.
= HFLUXZ Heat flux in the z-direction.
= HFLUXN Resultant heat flux.
    (default is TEMP)

d) For design optimization, valid components are:
= OBJFUN Objective function.
= BCON Behavior constraints.
= DVAR Design variables.

Note that the active type of analysis for postprocessing may be Linear Static, Frequency (or Buckling), or Thermal.

- Node or element number. For the stress option, element number is prompted. Not prompted for linear static analysis.

- Flag to plot magnitude or absolute phase angle of a displacement component versus frequency. Prompted for postdynamic harmonic analysis only. Phase angles are prompted in harmonic analysis for forces.
  = 0  Magnitude
  = 1  Phase
    (default is 0)

- Component label. (For postdynamic analyses only when the Y variable is FORCE, MOMENT or STRESS)
  = 1  x-translation (or equivalent).
  = 2  y-translation (or equivalent).
= 3  z-translation (or equivalent).
= 4  Rotation about the x-axis (or equivalent).
= 5  Rotation about the y-axis (or equivalent).
= 6  Rotation about the z-axis (or equivalent).

- Behavior constraint, or design variable set number. (prompted for design optimization only)

- Color to be used for plotting (1 to 16).
  (default is 16)

- Line style to plot graph.
  = 0  Solid line.
  = 1  Dotted line.
  (default is 0)

- Symbol type to be plotted at data points.
  = 0  No symbol
  = 1  Circle
  = 2  Filled circle
  = 3  Square
  = 4  Filled square
  = 5  Upward triangle
  = 6  Filled upward triangle
  = 7  Downward triangle
  = 8  Filled downward triangle
  = 9  Star
  = 10 Filled star
  (default is 1)

Note:
1. The specified component at the specified node must have been requested by the user before performing the corresponding analysis (NL Nodal Resp for nonlinear, Nodal Response for post dynamic. At other nodes if NL Plot and P_dyn Plot are used, a coarse graph may be obtained. Thermal analysis does not require pre-specification of nodes since plots for all nodes will be available.

Set Graph

Sets flags and parameters for plots to be generated by the Plot Sections command. You need to issue this command only when the default settings are not desired.

Set Graph

Input Description:
- Logarithmic scale flag in the x-direction.
  = 0  Linear scale.
  = 1  Logarithmic scale.
  (default is 0)

- Logarithmic scale flag in the y-direction.
  = 0  Linear scale.
= 1 Logarithmic scale.
  (default is 0)

- Number of intervals in the x-direction. Prompted only for linear x-scale.
  (default is 10)

- Number of intervals in the y-direction. Prompted only for linear y-scale.
  (default is 10)

- Axis showing flag.
  = 0 Do not show axes.
  = 1 Show the x-axis only.
  = 2 Show the y-axis only.
  = 3 Show both axes.
  (default is 3)

- Location of plotting the x-axis.
  = 0 Plot x-axis at minimum y value.
  = 1 Plot x-axis at y = 0.0.
  = 2 Plot x-axis at maximum y value.
  (default is 1)

- Location of plotting the y-axis.
  = 0 Plot y-axis at minimum x value.
  = 1 Plot y-axis at y = 0.0.
  = 2 Plot y-axis at maximum x value.
  (default is 1)

- Grid type in the x-direction.
  = 0 No grid.
  = 1 Dotted lines.
  = 2 Solid lines.
  (default is 1)

- Grid type in the y-direction.
  = 0 No grid.
  = 1 Dotted lines.
  = 2 Solid lines.
  (default is 1)

- Background fill color for xy-plots.

- Color number for grid lines.

- Relative width of xy-plot with respect to current window. Should be between 0.0 and 1.0.
  (default is 0.5 for section plots, 1.0 otherwise)

- Relative height of xy-plot with respect to current window. Should be between 0.0 and 1.0.
  (default is 0.5 for section plots, 1.0 otherwise)

- Relative x-position of xy-plot. Should be between 0.0 and 1.0.
  (default is 0.0 (left corner))

- Relative y-position of xy-plot. Should be between 0.0 and 1.0.
  (default is 0.0 (lower corner))
Graph Range

Specifies the X-Y graph ranges and scale factors.

Graph Range

Input Description:
- Scale factor to determine the x-range for the graph.
  (default is 1.0 (full range))

- Scale factor to determine the y-range for the graph.
  (default is 1.0 (full range))

- Minimum x value in the graph.
  (default is x-min times x-scale)

- Maximum x value in the graph.
  (default is x-max times x-scale)

- Minimum y value in the graph.
  (default is y-min times y-scale)

- Maximum y value in the graph.
  (default is y-max times y-scale)

Reference Line

Plots a reference line parallel to the x- or y-axis at a given value.

Reference Line

Input Description:
- Axis parallel to the reference line to be plotted.
  = 0 Y-axis.
  = 1 X-axis.
  (default is 0)

- Distance from the origin to the reference line. (X or Y coordinate of the reference line)

Identify

Identifies the coordinates of the pixel closest to the tip point of the mouse arrow. The point can be anywhere in the graph range.

Identify (move the mouse to the desired point and click left button)
**List Graph**

Lists the available information to produce xy-plots from program-generated files. Title, node, name, and direction (if applicable) are listed for the analysis type active for postprocessing (refer to **Analysis Type**).

**List Point**

Lists numeric values of a pattern of points for all the activated xy-graphs. Graph number, title and the corresponding values of the dependent and independent variables are listed.

**Input Description:**
- Beginning point to be listed.
  (default is 1)
- Ending point in the pattern.
  (default is last point)
- Increment between points in the pattern.
  (default is 1)

**Plot Graph**

Generates a time-history-like plot for the vectors stored in the specified graph. The plot data must have been activated by **Act Pre X-Y**, or **Act Post X-Y**.

**Input Description:**
- Number of the graph to be plotted. Up to 6 graphs can be activated and plotted simultaneously.
SELECTION Menu

A selection list is a filter that can be conveniently used in many operations. When no selection list is defined for a particular entity type, then all members of that entity are equally accessed by the interface for all operations, but when a selection list is available, an option is available to only operate on the members in the selection list. Selection lists are particularly useful in the pre- and postprocessing of large models and are available for Points, Curves, Contours, Regions, Nodes, and Elements. Several procedures are provided for creating new selection lists or adding to existing ones. As an example, suppose that you are working on a model that has many regions, and you want to fix all nodes associated with these regions in the x-direction. This task may be done in two ways; you may apply the required constraint region by region, or you may create a selection list which contains these regions and then apply the desired constraint to all selected regions at once. The Selected option in the Displacement submenu of the Loads/BConds menu and similar submenus is provided for this purpose. If geometry is created in a CAD program, then a selection list for curves is automatically created. The figure below shows an outline for the commands available in the Selection menu.

Figure 5.100 Selection Menu
Select Window Submenu

The Select Window command opens the following option menu:

<table>
<thead>
<tr>
<th>SELECT WINDOW</th>
</tr>
</thead>
<tbody>
<tr>
<td>Box, Circle, Polygon, Single Pick</td>
</tr>
<tr>
<td>Keypoints, Curves, Regions, Contours, Nodes, Elements</td>
</tr>
<tr>
<td>Select, Unselect</td>
</tr>
<tr>
<td>Accept, Quit</td>
</tr>
</tbody>
</table>

![Figure 5.101 Select Window Submenu]

The command initiates, adds to or takes away members from a selection list of a specified entity by forming a window surrounding the region of interest. Rectangular, circular or polygon windows can be specified by using the mouse. All plotted members of the specified entity whose labels are within the window are added to the selection list. Select the window and entity types and Accept. A rectangular window is specified by two opposite corners using the mouse. A circular window is specified by the center and a point on the circumference picked by mouse. A polygon window is drawn using the mouse in a rubber band fashion, type "c" to close the polygon, "d" to delete the last segment, the Escape key and the middle button of the mouse abort the command. The Single Pick option allows picking of individual members of the entity using the mouse. Choose the window type, entity type, Select or Unselect and Accept to add to or take away from a selection list respectively.

It is important to note that only plotted members of the entity may be selected.

<table>
<thead>
<tr>
<th>Select Window</th>
</tr>
</thead>
</table>

**Input Description:**
- Window type, select Box, Circular, Polygon, or Single Pick.
- Entity type can be Keypoints, Curves, Regions, Contours, Nodes, or Elements.
- Select Accept to execute the command or Quit to quit it and return to the Selection menu.

*After selecting the window type and entity type, use Quit to quit the command or Accept to execute it with the highlighted options. For example, if Box and Keypoints are highlighted and Accept is*
selected, you can then use the mouse to create the box and all keypoints inside it will be added to the selection list.

Select Ref Submenu

The Select Ref command initiates, adds to or takes away members from a selection list of a particular entity. You need to specify the entity type you want to select (target entity) and the type of the reference entity. The members of the reference entity type should be put in a selection list before executing this command. When the command is executed, all members of the target entity that are associated with any of the reference entities in the selection list are selected. Choose the target entity type, the reference entity type, and Select or Unselect to add to or take away members of the target entities associated with the reference entities respectively.

<table>
<thead>
<tr>
<th>SELECT REF</th>
</tr>
</thead>
<tbody>
<tr>
<td>Keypoints</td>
</tr>
<tr>
<td>Curves</td>
</tr>
<tr>
<td>Regions</td>
</tr>
<tr>
<td>Contours</td>
</tr>
<tr>
<td>Nodes</td>
</tr>
<tr>
<td>Elements</td>
</tr>
<tr>
<td>Ref Keypoints</td>
</tr>
<tr>
<td>Ref Curves</td>
</tr>
<tr>
<td>Ref Regions</td>
</tr>
<tr>
<td>Ref Contours</td>
</tr>
<tr>
<td>Ref Nodes</td>
</tr>
<tr>
<td>Ref Elements</td>
</tr>
<tr>
<td>Select</td>
</tr>
<tr>
<td>Unselect</td>
</tr>
<tr>
<td>Single Pattern</td>
</tr>
<tr>
<td>All</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.102  Select Ref Submenu

Select Ref

Input Description:
- Type of entity to be selected. Highlight one of the following: Keypoints, Curves, Regions, Contours, Nodes, or Elements.

- Type of reference entity. Highlight Keypoints, Curves, Regions, Contours, Nodes, or Elements.

- Select Accept to execute the command or Quit to quit it and return to the Selection menu.
If **Accept** is selected and a selection list for the reference entity exists the user is prompted for the boundary flag as follows:

- **Selection Flag** (enter by keyboard).
  - \( = 0 \)  Consider all members of the reference entity.
  - \( = 1 \)  Consider members of the reference entity that are in the selection list.
    (default is 1)

- **Selection of boundary target entities** (boundary element flag) (enter by keyboard).
  - \( = 0 \)  To select target entities associated with reference entities in the selection list only.
  - \( = 1 \)  To include in addition, all members that have common boundaries with the selected entities.
    (default is 0)

### Sel Elem By Prop Submenu

This command initiates or adds elements to an element selection list. All elements associated with a pattern of element groups, material property sets, or real constant sets are added to the element selection list.

![Sel Elem By Prop Submenu](image)

**Figure 5.103 Sel Elem By Prop Submenu**

**Sel Elem By Prop**

**Input Description:**
- Property type, select **Elem Group**, **Material Prop**, or **Real Constants**.
- Select **Accept** to execute the command or **Quit** to quit it and return to the Selection menu.

### Init Select Submenu

This submenu may be used to initialize the selection list for a particular entity. Highlight options from the following menu:
Init Select

Input Description:
- Type of entity for which the selection list is to be initialized. Select from Keypoints, Curves, Regions, Contours, Nodes, or Elements.

- Select Accept to execute the command or Quit to quit it and return to the Selection menu.

If Accept is selected, the user is prompted for the selection set number as follows:

- Selection set number.
  - = 0     All
  - = n     Set number.

Example: If Nodes and Accept are selected, then the keypoint selection list will be initialized. Quit quits the command and returns to the Selection menu.

Complement Submenu

This submenu may be used to replace an existing selection list for a particular entity type by its complement. Highlight options from the following menu:
Every member in the selection list will be taken out and every member that was not in the selection list will be a member. This command is useful when it is needed to divide the model into two distinct groups of members for a particular operation.

### Complement

**Input Description:**
- Type of entity for which the selection list is to be complemented. Select from **Keypoints**, **Curves**, **Regions**, **Contours**, **Nodes**, or **Elements**.
- Select **Accept** to execute the command or **Quit** to quit it and return to the Selection menu.

*If Accept is selected, the user is prompted for the selection set number as follows:*

- Selection set number.
  - $= 0$ All
  - $= n$ Set number.

### Init All

This command initializes all selection lists for all entity types.

### Select Plot Submenu

This command may be used for plotting all members of the selection list for a given entity. Highlight the entity type from the following menu:

<table>
<thead>
<tr>
<th>SELECT PLOT</th>
</tr>
</thead>
<tbody>
<tr>
<td>Keypoints</td>
</tr>
<tr>
<td>Curves</td>
</tr>
<tr>
<td>Regions</td>
</tr>
<tr>
<td>Contours</td>
</tr>
<tr>
<td>Nodes</td>
</tr>
<tr>
<td>Elements</td>
</tr>
</tbody>
</table>

- **Accept**
- **Quit**

*Figure 5.106  Select Plot Submenu*
**Input Description:**
- Type of entity to be plotted. Select from **Keypoints, Curves, Regions, Contours, Nodes**, or **Elements**.
- Select **Accept** to execute the command or **Quit** to quit it and return to the Selection menu.

*If Accept is selected, the user is prompted for the selection set number as follows:*

- Selection set number.
  - = 0  All
  - = n  Set number.

**Selection Status**

This command displays and controls multiple selection lists.

<table>
<thead>
<tr>
<th>Selection Status</th>
</tr>
</thead>
</table>

**Clear**

This command clears the window and is repeated in several windows for convenience.

<table>
<thead>
<tr>
<th>Clear</th>
</tr>
</thead>
</table>
Read Input File

This loads an existing meshed model or geometry model from a CAD program to the COSMOS/M CAD Interface.

Input Description:
- Model name without extension.
**COSMOS/M Dbase**

Loads an existing COSMOS/M database for continued work.

**COSMOS/M Dbase**

**Input Description:**
- Database name.

**System**

Opens a shell window. Type exit to return to the interface.

**Save**

Saves the current database to a new name or just saves the current session file. The session files has the extension SES and is an ASCII file that has a record of commands. The SES file can be used to reconstruct the model using the **Pro Model** command.

**Save**

**Input Description:**
- Save option.
  - = 0  Save session file only.
  - = 1  Save entire database.
    (default is 0)

- Input new database name for saving entire database.

**Status**

This option opens the status table which shows the currently active sets and the plotting status of various geometric entities. The user can toggle the flags to change the settings and keep them for this model by closing the table using the Save box in the upper right-hand corner. If the changes are not to be kept the user can exit by selecting the Quit box in the upper left hand corner. Note that some entities and some flags are not explicitly used in COSMOS/M CAD Interface, but are useful when working in the GEOSTAR mode.

The active sets are shown in the top bar of the table. A guide to the symbols follows.
### Status

#### Sets & Description:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EG</td>
<td>Element group</td>
</tr>
<tr>
<td>MP</td>
<td>Material set</td>
</tr>
<tr>
<td>RC</td>
<td>Real constant set</td>
</tr>
<tr>
<td>CS</td>
<td>Coordinate system</td>
</tr>
<tr>
<td>LC</td>
<td>Load case</td>
</tr>
<tr>
<td>TC</td>
<td>Time curve</td>
</tr>
<tr>
<td>TP</td>
<td>Temperature curve</td>
</tr>
<tr>
<td>MC</td>
<td>Magnetic curve</td>
</tr>
</tbody>
</table>

The entities considered in the table are:

#### Entities & Description:

<table>
<thead>
<tr>
<th>Symbol</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PT</td>
<td>Keypoints</td>
</tr>
<tr>
<td>CR</td>
<td>Curves</td>
</tr>
<tr>
<td>SF</td>
<td>Surfaces</td>
</tr>
<tr>
<td>VL</td>
<td>Volumes</td>
</tr>
<tr>
<td>CT</td>
<td>Contours</td>
</tr>
<tr>
<td>RG</td>
<td>Regions</td>
</tr>
<tr>
<td>ND</td>
<td>Nodes</td>
</tr>
<tr>
<td>EL</td>
<td>Elements</td>
</tr>
<tr>
<td>CS</td>
<td>Coordinate systems</td>
</tr>
</tbody>
</table>

The status information that can be controlled for each of the entities listed above is as follows:

---

```plaintext
<table>
<thead>
<tr>
<th>PRIM</th>
<th>PLOT</th>
<th>PCLR</th>
<th>LABL</th>
<th>LCLR</th>
<th>MAXM</th>
<th>KEEP</th>
<th>MARK</th>
<th>DMSH</th>
</tr>
</thead>
<tbody>
<tr>
<td>PT</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>87</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
</tr>
<tr>
<td>CR</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>179</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
</tr>
<tr>
<td>SF</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>58</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
</tr>
<tr>
<td>VL</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>0</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
</tr>
<tr>
<td>CT</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>35</td>
<td>OFF</td>
<td>ON</td>
<td>N/A</td>
</tr>
<tr>
<td>RG</td>
<td>OFF</td>
<td>ON</td>
<td>OFF</td>
<td>ON</td>
<td>29</td>
<td>OFF</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>PH</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>1</td>
<td>OFF</td>
<td>N/A</td>
<td>ON</td>
</tr>
<tr>
<td>PA</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>1</td>
<td>OFF</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>ND</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>0</td>
<td>OFF</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>EL</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>OFF</td>
<td>0</td>
<td>OFF</td>
<td>N/A</td>
<td>N/A</td>
</tr>
<tr>
<td>CS</td>
<td>OFF</td>
<td>N/A</td>
<td>---</td>
<td>N/A</td>
<td>---</td>
<td>N/A</td>
<td>N/A</td>
<td>N/A</td>
</tr>
</tbody>
</table>
```

**Figure 5.108 Status Table**
**Option & Description:**

**PLOT**
Flag ON/OFF, if set ON, the entity will be plotted if a higher entity is requested for plotting. The entity will be colored according to the PCLR.

**LABL**
Flag ON/OFF, if set ON, will plot entity labels using the color set in LCLR.

**MAXM**
Maximum entity label in the model.

**KEEP**
Flag ON/OFF, used to specify whether lower order entities should be kept when higher entities are deleted.

**PSEL**
Flag to activate or deactivate a selection set.

**MARK**
Flag ON/OFF. When ON a marker is placed on the entity to show its orientation.

**DMSH**
Flag ON/OFF to activate or deactivate default mesh generation for the entity. (Not used in COSMOS/M CAD Interface.)

**Log Listing Submenu**

Opens or closes a script file to record response of the interface to listing and help commands. If a file is open, all list and help screens will be written to the file.

<table>
<thead>
<tr>
<th>LOG LISTING</th>
</tr>
</thead>
<tbody>
<tr>
<td>On</td>
</tr>
<tr>
<td>Off</td>
</tr>
</tbody>
</table>

Figure 5.109  Log Listing Submenu

**Log Listing**

**Option & Description:**

**On**
Open a log file.

**Off**
Close a log file.

**Input Description:**
The next two commands are prompted and used only if a file is to be opened.
- Name of file.  
  (default is prob-name.LIS)

- Display flag.  
  = 0  Write listings and help to the open file but do not display on the screen.  
  = 1  Display listing and help on the screen as well as writing them to the file.  
  (default is 1)

Notes:  
1. If the command is used to open a file while another file is already open, the command closes the existing file and opens the new file.  This allows the user to open several files in a session to record relevant information in separate files.  However only one file can be active at any time.
2. If an existing file is opened, the information will be appended to it.
3. The listing file is closed whenever control is transferred to another module.

Output Model

Generates an ASCII format input file for COSMOS/M.

### Output Model

<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
</table>
| - File name with extension.  
  (default is <probname>.GFM) |

<table>
<thead>
<tr>
<th>- Output flag.</th>
</tr>
</thead>
<tbody>
<tr>
<td>= 0  Geometry only.</td>
</tr>
<tr>
<td>= 1  FE Model only.</td>
</tr>
</tbody>
</table>
| = 2  Both.  
  (default is 2) |

<table>
<thead>
<tr>
<th>- Append to existing file.</th>
</tr>
</thead>
<tbody>
<tr>
<td>= 0  No</td>
</tr>
</tbody>
</table>
| = 1  Yes  
  (default is 1) |

| - Entity types, PT, CR, SF, CT, RG, ALL. Only needed if output = 0 or 2.  
  (default is ALL) |

<table>
<thead>
<tr>
<th>- Association flag. Only with output = 2.</th>
</tr>
</thead>
<tbody>
<tr>
<td>= 0  No association.</td>
</tr>
</tbody>
</table>
| = 1  Keep association between geometry and mesh.  
  (default is 1) |

| - Node number offset. Only if output = 1 or 2.  
  (default is 0) |
- Element number offset. For output = 1 or 2.
  (default is 0)

- Offset for coordinate system numbering.
  (default is 0)

**Element Plot Ops**

Sets the element plot options.

**Element Plot Ops**

**Input Description:**
- Element plot order.
  = 0   Linear
  = 1   Bilinear
  (default is 0)

- Plot beam orientation.
  = 0   No
  = 1   Yes
  (default is 0)

- Special element flag.
  = 0   No
  = 1   Yes
  (default is 0)

**Hard Copy Submenu**

This submenu is only available in the Unix version. It opens a menu of commands to generate output plots or files. In the Windows version, the user has only the option to print to the system default printer.

<table>
<thead>
<tr>
<th>HARD COPY</th>
</tr>
</thead>
<tbody>
<tr>
<td>Paint Jet</td>
</tr>
<tr>
<td>Laser Jet</td>
</tr>
<tr>
<td>Desk Jet</td>
</tr>
<tr>
<td>HP7475</td>
</tr>
<tr>
<td>Postscript</td>
</tr>
</tbody>
</table>

**Figure 5.110   Hard Copy Submenu**
Paint Jet
Generates output for a Paintjet printer.

**Paint Jet**

**Input Description:**
- Route option.
  - = 0  Plot now.
  - = 1  Save to file.
  - = 2  Do both.
    (default is 0)

- Input file name for saving.
  (default is hcopy.out)

- Printer device name.

- Printer resolution.
  (default is 90)

Laser Jet
Generates output for an HP Laserjet or compatible printer.

**Laser Jet**

**Input Description:**
- Route option.
  - = 0  Plot now.
  - = 1  Save to file.
  - = 2  Do both.
    (default is 0)

- Input file name for saving.
  (default is hcopy.out)

- Printer device name.

- Printer resolution.
  (default is 90)

Desk Jet
Generates output for an HP_DESKJET 500 C printers.

**Desk Jet**
**Input Description:**
- Deskjet type.
  - = 1 Color
  - = 2 B/W

- Resolution

- Communication mode.
  - = 0 Send now.
  - = 1 Save file.
  - = 2 Do both.

- Hardcopy output file name.

### HP7475
Generates output for HP7475 or compatible plotters.

```
<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Type of plotter.</td>
</tr>
<tr>
<td>= 1 HP7475A</td>
</tr>
<tr>
<td>= 2 ColorPro</td>
</tr>
<tr>
<td>= 3 HP7450</td>
</tr>
<tr>
<td>= 4 DraftMaster</td>
</tr>
</tbody>
</table>
  (default is 2)

- Route option.
  - = 0 Plot now.
  - = 1 Save to file.
  - = 2 Do both.

- Input file name for saving.
  (default is hcoup.out)

- Plotter device name.

- Resolution.
  (default is 72)
```

### Postscript
Saves the image in the active window in postscript format.

```
<table>
<thead>
<tr>
<th>Input Description:</th>
</tr>
</thead>
<tbody>
<tr>
<td>Input file name for saving.</td>
</tr>
</tbody>
</table>
```
Measure Submenu
A menu for geometry and mesh measurement.

<table>
<thead>
<tr>
<th>MEASURE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Distance</td>
</tr>
<tr>
<td>Length</td>
</tr>
<tr>
<td>Area</td>
</tr>
<tr>
<td>Angle</td>
</tr>
</tbody>
</table>

Figure 5.111  Measure Submenu

Distance Submenu
This command calculates and lists the distance between two keypoints, two nodes, or a keypoint and a node.

<table>
<thead>
<tr>
<th>DISTANCE</th>
</tr>
</thead>
<tbody>
<tr>
<td>Two Points</td>
</tr>
<tr>
<td>Point Node</td>
</tr>
<tr>
<td>Two Nodes</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.112  Distance Submenu

Length Submenu
This command calculates and lists the length of a curve or contour.

<table>
<thead>
<tr>
<th>LENGTH</th>
</tr>
</thead>
<tbody>
<tr>
<td>Curve</td>
</tr>
<tr>
<td>Contour</td>
</tr>
<tr>
<td>Accept</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.113  Length Submenu
Area Submenu

This command calculates and lists the area defined by four keypoints or nodes.

```
AREA
Four Points
Four Nodes
Accept
Quit
```

Figure 5.114 Area Submenu

Angle Submenu

This command calculates and lists the angle between three keypoints or three nodes. The angle measured is at the second keypoint or node.

```
ANGLE
Three Points
Three Nodes
Accept
Quit
```

Figure 5.115 Angle Submenu

Productinfo

Displays product information.

Flow Chart

**Flow Chart** is a command available from the Main menu that activates the flowchart utility to help you proceed with your analysis. Flowcharts are diagrams that graphically display the systematic procedure required to perform various types of analyses. The **Flow Chart** utility in COSMOS/M CAD Interface is designed to guide the user throughout the steps for preprocessing, analysis, and postprocessing of results for all types of analyses. Flowcharts may be used to provide information regarding the status of the various steps in the database by listing the relevant information. Flowcharts are linked with a navigation system that
automates the choice of menus and commands needed to assign data towards the completion of a particular step. By pressing the right button of the mouse, you will be automatically taken to the specific menu and command where you may provide the desired information. Colors are used in the flowcharts to quickly identify the status of the various steps as follows:

- A red color for a particular step indicates that the step is not completed and input must be given before you can continue.

  For example, if geometry exists and no mesh has been generated, then the Mesh Model step as well as subsequent steps like applying loads and boundary conditions and postprocessing will show in red. Options related to subsequent steps in the menus will not be available until a mesh has been generated. Pressing the right button will tell you that the geometry and element attributes have been defined and that you may now mesh the model. Pressing the left button will take you to the meshing menu where you can mesh the model.

  Find the first red-colored step in the flowchart, get help using the right button, and execute the required command(s) to give the desired input. Choose Flow Chart again and repeat the process until all results become available for postprocessing.

- A green color indicates that some information related to that step has been provided. It does not tell however whether the existing information is sufficient or not.

  As an example, if one keypoint exists in the model, then geometry is considered to be defined and the corresponding step in the flowchart will show in green. Similarly if for a structural problem one degree of freedom is prescribed, then boundary conditions will be considered defined and the corresponding step will show in green even though the analysis might stop because of insufficient boundary conditions.

- A blue color indicates that the step is optional. You may use the right button of the mouse to get help on the available options.

  As an example, chart number 2 contains the menu for the various types of analyses that may be performed in COSMOS/M CAD Interface and all options at the bottom of the tree will be shown in blue so that you may choose the desired analysis. Note that the upper level (like Nonlinear, Frequency, and Advanced Dynamics) are colored in Gray.

- A gray color is used whenever the block is used for classification purposes as in the case of analysis types in flowchart number 2 illustrated by the example above, or to indicate that the development work for that option has not been completed yet. The color is used for illustration purposes when the step is irrelevant or need not be executed as for meshing when the mesh has been generated in a CAD program.

There are 3 levels in the Flow Chart hierarchy. In the first level (stack 0), choose whether the geometry or mesh is exported from a CAD program to the interface. In the second level (stack 1), choose the type of analysis. The third level will display the flowchart for a particular analysis.

**Masking**

The masking utility helps you avoid inconsistent or irrelevant options throughout preprocessing, analysis, and postprocessing by blocking access to irrelevant menus and options. Masking may be toggled on or off using the F1 key.
As an example if no element group and associated material properties and real constant sets are defined in the model, then the Meshing menu will be masked and assuming that some geometry exists in the model the mask will disappear once such information has been defined. Depending on the element group defined, only meshing of compatible geometric entities will be allowed. For example if the active element group is SHELL3, then you may mesh regions and polyhedrons but not curves or parts.

**Using the Mouse with Flow Chart**

The functions of the right and left buttons of the mouse will be described in every flowchart. Pressing the right button of the mouse will give you help for performing that particular step. Pressing the left button on a step will either take you to the next flowchart or give you the status and take you to the menu related to completing that step.

<table>
<thead>
<tr>
<th>Flow Chart</th>
</tr>
</thead>
<tbody>
<tr>
<td>The top level <strong>Flow Chart</strong> will be displayed. Use the mouse to choose the desired options. Refer to Chapter 2 for examples.</td>
</tr>
</tbody>
</table>

The flowcharts for linear static, frequency, buckling, and steady-state heat transfer analyses are similar except for the name of the type of analysis to run. The first and second level flowcharts as shown in Figures 5-116 and 5-117 respectively. The flowcharts for running linear static analysis in the cases of importing the geometry and the mesh from a CAD program are shown in Figures 5-118 and 5-119 respectively. The flowchart for running transient heat transfer analysis is shown in Figure 5-120.

Flowcharts for nonlinear structural and post-dynamic analyses are available in the COSMOS/M CAD Interface Advanced Modules Manual.
Figure 5.116  First Level Flowchart (stack 0)
Figure 5.117  Second Level Flowchart (stack 1)
Figure 5.118  Flowchart for Running Linear Static Analysis (Importing Geometry from a CAD program)
Figure 5.119 Flowchart for Running Linear Static Analysis (Importing Mesh from a CAD program)
Figure 5.120 Flowchart for Running Transient Thermal Analysis (Importing Geometry from a CAD program)
What Next?

The What Next? command, available from the Main menu, displays a screen that tells you the status of the various steps in the database and what you should do next to proceed with the selected analysis. The command currently supports linear static (STAR and FFE Static), frequency and buckling (DSTAR), and heat transfer (HSTAR and FFE Thermal) analyses only.

<table>
<thead>
<tr>
<th>WHAT NEXT?</th>
</tr>
</thead>
<tbody>
<tr>
<td>Static</td>
</tr>
<tr>
<td>Frequency</td>
</tr>
<tr>
<td>Buckling</td>
</tr>
<tr>
<td>Thermal</td>
</tr>
<tr>
<td>Quit</td>
</tr>
</tbody>
</table>

Figure 5.121  What Next? Submenu

What Next?

Option & Description:

Static
- Linear static analysis.

Frequency
- Modal analysis to calculate natural frequencies and corresponding mode shapes.

Buckling
- Linearized buckling analysis to calculate buckling load factors and corresponding mode shapes.

Thermal
- Heat transfer analysis.
(This page is intentionally left blank)
Introduction

This chapter is intended to familiarize the user with the interface. Several examples are used to demonstrate the general procedures for pre- and postprocessing. The user is encouraged to first follow the instructions, and then examine and try other options. Please note that the following examples are based on the Unix version of the software.

Example 1: Analysis of a Dashboard

Features:
- Geometry: Pro/ENGINEER
- Meshing: Pro/MESH
- Elements: SHELL3 or SHELL3T depending on Pro/ENGINEER release
- Analysis: Linear Static Analysis

The dashboard in this example is included in the Pro/ENGINEER example library under the name `newdash.prt` and represents an automobile dashboard. The user needs to have access to Pro/MESH and Pro/INTERFACE in Pro/ENGINEER to execute this example. The geometry and mesh information is stored in the file `ex3.prt` and the material information is in the file `ex3.mat`.

Preprocessing in Pro/ENGINEER

1. Copy files `ex3.prt`, and `ex3.mat` to your work subdirectory as follows:
   
   ```
   CP $COSMOSM/PROBS/ex3.prt .
   CP $COSMOSM/PROBS/ex3.mat .
   ```

2. Type `COSPRO` to start COSMOS/M CAD Interface.

3. Select CAD Program from the top level menu to invoke Pro/ENGINEER.

4. Select Mode from the Pro/ENGINEER Main menu, Part from Mode, and Search/Retrieve from EnterPart.

5. Select `ex3.prt` to retrieve this example. The model will be displayed on the screen. Material properties were defined in Pro/ENGINEER and will be read from the `ex3.mat` file.

6. Select FEM from Part, Define Model from FEM, and Loads/BC from Define Model. Then select Modify from Constr Case and LC1 from Concase Name. Select Structural from Loads/BC, All and Show from Structural and Select All from FEM Select. All structural loads and boundary conditions will now be displayed. Select Done/Return from Loads/BC to return to the FEM menu.

7. To generate the shell mesh, select Make Model from FEM, Shell Mesh from FEM Mesh, Mesh Pairs from Shell Mesh, Triangles from Shell Type and wait for the mesh to be generated.
8. Select **Output Model** from FEM and **COSMOS/M, Linear, Structural**, and **Accept** from Output Mesh. Pro/MESH should output the linear shell elements in the COSMOS/M format, hit **Enter** to accept the default file name. The preprocessing phase is completed.

9. Select **Exit** from the Main menu to exit Pro/ENGINEER and return to COSMOS/M CAD Interface. The model will be automatically loaded under the same name to COSMOS/M CAD Interface. Figure 6.1 below shows the meshed model.

![Dashboard Model Displayed in COSMOS/M CAD Interface](image)

**Figure 6.1** Dashboard Model Displayed in COSMOS/M CAD Interface

**Inside the Pro/ENGINEER Interface**

In general, once inside COSMOS/M CAD Interface, it is good practice to check the information in the database like element group information, elements, material properties, real constant sets, and loads and boundary conditions.

1. Select **Properties** from the Main menu, **Element Groups** from Properties, and **List Groups** from Element Groups. Verify that SHELL3T is the active element group. The element group for elements generated by the Shell Mesh command in Pro/MESH will be SHELL3 or SHELL3T depending on the Pro/ENGINEER release in use.

2. Select **Properties** from the Main menu, **Material** from Properties, and **List Sets** from Material. In general, verify that the listed material properties are translated correctly from Pro/ENGINEER. Material properties can be modified in the interface by redefining them by the **Define Set** command in the Material menu.

3. Select **Properties** from the Main menu, **Real Constants** from Properties, and **List RC Sets** from Real Constants. Note the listed thickness. In general, real constant sets should be verified against the information specified in Pro/ENGINEER. The thickness of the shell may be modified, if desired, by the **Define RC Set** command in the Real Constants menu.
4. To view the model, select **Viewing** from the Main menu, and **Shell Shade** and **Clear** from Viewing.

Then select **Meshing** from the Main menu, **Element** from Meshing, and **Plot, All**, and **Accept** from Element. Since the shell shade is active, a shaded plot as shown in Figure 6.2 will be generated. Shell Shade gives different colors to top and bottom faces. It is important for stress results to make sure that top and bottom faces of adjacent shell elements are matching. Otherwise the averaging process will mix up bottom and top face stresses. If the mesh is generated in Pro/MESH, then the orientation is such that the top face is at the outer side of the polyhedra, and the bottom is at the inner side.

In cases where geometry is available in the interface, you may use the **Reorient** command which can be called from the Region/Geometry menu to reorient the elements associated with a region if adjacent elements are not correctly oriented. The orientation is not important for displacement, frequency, and critical buckling load calculations.

![Shaded Element Plot](image)

**Figure 6.2 Shaded Element Plot**

5. Select **Meshing** from the Main menu, **Element** from Meshing, and **List, All**, and **Accept** from Element. Note the four element attributes (EG, MP, RC, and EC), and that 3 nodes are listed for each element.

6. Select **Utilities** from the Main menu and **Status** from Utilities. Examine the status of different flags and variables. Note the number of nodes and elements in the database under the MAXM column. To continue, use the mouse to click on **Quit** in the upper left corner in the Status menu.
Running the Analysis

Select **Analysis** from the Main menu, **Static**, from Analysis, and **Run** from Static.

Control is now given to **STAR**, the static analysis module of COSMOS/M to calculate displacements. The COSMOS/M CAD Interface screen disappears, and **STAR** starts to display information related to the problem and the progress of the analysis. **STAR** calls a renumbering routine before it starts to solve the problem. The renumbering routine renumbers the nodes in order to minimize the bandwidth of the stiffness matrix and hence the solution time. Once the deformations are calculated, control is given to the **STRESS** module to calculate strains and stresses. After calculating the stresses, control is given back to COSMOS/M CAD Interface. If the analysis was not completed for one reason or another, a message will be displayed on the screen. Refer to the end of the output file (extension OUT) for explanation.

Postprocessing

1. Select **Postprocessing** from Main, **Display Plots** from Postprocessing, and **Plot Displacement** from Display Plots. Note that you may plot the resultant displacements or individual components as shown in the menu.

2. Choose **Displacement, Resultant, Filled Contour, Undeformed**, and **Accept** from Plot Displacement. Accept the default load case prompted in the dialog area. Figure 6.3 will be generated as shown below. Note that the colored displacement contours may be plotted on the deformed or undeformed shape as desired.

![Figure 6.3 Displacement Contour Plot](image)
Dynamic Viewing

Dynamic viewing is only available for the HP and SGI platforms, if you are using any other platform, please skip this step.

3. Select Dynamic View from Display Plots. Select Demo on and Accept from Dynamic View. Type 1; and hit Enter. Dynamic viewing may be stopped by the Esc key.

4. To plot stresses, select Display Plots from Postprocessing and Plot Stress from Display Plots. Note the various stress components that you may activate.

5. Choose Von Mises, Nodal, Top Face, Filled Contour, Undefomed, and Accept from Plot Stress. Press Enter to accept the default load case number. The generated plot is shown in Figure 6.4. Note again that deformed or undeformed shapes can be used for stress plots.

6. Select Display Plots from Postprocessing and Animate from Display Plots. Accept default values in the message window. Von Mises stresses will be animated on the undeformed shape. The animation process may be interrupted by pressing the Esc key.
Example 2: Analysis of a Front Wheel Assembly

Features:

- **Geometry:** Pro/ENGINEER
- **Meshing:** COSMOS/M CAD Interface
- **Elements:** TETRA4R
- **Analysis:** Linear Static Analysis

This example is an automobile front wheel assembly. The geometry of this model is included in the Pro/ENGINEER library of parts under the name `fskneckle.prt`, and in the COSMOS/M directory under the file `ex4.prt`. The mesh and element attributes will all be generated in the interface. The example will also illustrate the global and local mesh control features in addition to mesh refinement. Note that material properties will not be transferred to the interface in cases where meshing is not done by Pro/MESH.

**Preprocessing**

1. Copy file `ex4.prt` to your work subdirectory as follows:
   
   ```bash
   CP $COSMOSM/PROBS/ex4.prt .
   ```

2. Type `cospro` to start COSMOS/M CAD Interface.

3. Select **CAD Program** to start Pro/ENGINEER.

4. Select **Mode** from the Pro/ENGINEER Main menu, **Part** from Mode, and **Search/Retrieve** from EnterPart.

5. Select `ex4.prt` to retrieve this example. The model will be displayed on the screen.

6. Select **COSMOS Geom** from Part and **IGES Format** from COSMOS Geom. Accept default IGES file name.

7. Select **Exit** from the Main menu to exit Pro/ENGINEER and return to COSMOS/M CAD Interface. The model geometry will be automatically loaded to the interface under the same name (`ex4`). Material properties will have to be defined in the interface even if they were defined in Pro/ENGINEER.

**Inside the Pro/ENGINEER Interface**

1. Once geometry is transferred to the interface, the model will be displayed as shown in Figure 6.5.
2. Select **Geometry** from the Main menu, **Polyhedron** from Geometry, and **List, All**, and **Accept** from Polyhedron. The list screen will list the regions used in the polyhedron, the average global element size for the polyhedra and the tolerance. This element size will be used when the polyhedron or the corresponding part is meshed. The average element size is automatically calculated by the interface upon geometry transfer from Pro/ENGINEER. The listed average element size in this case is **0.6185**. We will show how to change the global average element size after defining element attributes.

3. Select **Geometry** from the Main menu, **Part** from Geometry, and **List, All, and Accept** from Part. One part made up of one polyhedron will be listed.

**Defining Element Attributes**

The element group may be defined as TETRA4 or TETRA4R. TETRA4R has 6 degrees of freedom per node (3 translations and 3 rotations), while TETRA4 has only 3 degrees of freedom per node (3 translations). TETRA4R will be used for this model.

1. Select **Properties** from the Main menu, **Element Groups** from Properties, and **Define Group** from Element Groups. Choose **TETRA4R** and **Accept** from the Define menu. Accept the default label of **1** for this element group. You will be prompted to specify element group options, accept defaults for OP1 through OP8. (Refer to Element Library for details.)

2. Select **Properties** from the Main menu, **Material** from Properties, and **Pick Material** from Material. Press **Enter** for the material property set label [1], type **BRASS** and press **Enter** for the material name and accept the default **FPS** system of units.
3. To verify your input for the element group, select Properties from the Main menu, Element Groups from Properties, and List Groups from Element Groups. Verify that TETRA4R is the active element group. Note that element group options are also listed.

4. To verify your input for material properties, select Properties from the Main menu, Material from Properties, and List Sets from Material. Material properties read from the library for Brass will be listed.

5. Note that no real constant set is needed for TETRA4R elements.

Global Mesh Control

By global mesh control, we mean the ability to change the average element size for polyhedra. Changing the element size for a polyhedron changes the average element size for all associated regions, contours, and curves.

1. Select Meshing from the Main menu, Mesh Density from Meshing, and Polyhedron, All, and Accept from Mesh Density. You will be asked to input the new desired element size. Enter 0.5 for the average element size and accept the default value for the tolerance.

   The specified element size should be reasonable. Otherwise problems may arise in subsequent meshing. A reasonable element size should not exceed twice to three times the length of the smallest curve unless local mesh control is used. The specified element size should not be smaller than \( \frac{1}{10} \)th of the tolerance used to create the polyhedron. The tolerance will be shown by listing the polyhedra.

2. List the polyhedron again to verify the change in the element size.

Local Mesh Control

By local mesh control, we mean the ability to change the average element size for curves and regions. For parts made up of more than one polyhedron, changing the mesh size for polyhedrons may be considered as local meshing. We will first plot regions so that we can pick them from the screen.

1. Select Utilities from the Main menu and Status from Utilities. Turn on the plotting of labels for regions, by clicking on the flag in the intersection of the RG row and LABL column. Click on Save to activate the change. The labels of regions will show up in subsequent region plots.

2. Select Geometry from the Main menu, Region from Geometry, and Plot, All, and Accept from Region. The plotted regions are shown in Figure 6.6. Regions 21, 54, and 79 are shaded in the figure below for convenience.
3. Select **Meshing** from the Main menu, **Mesh Density** from Meshing, and **Region, Pick,** and **Accept** from Mesh Density. Pick region 21 using the mouse, and input an average element size of \(0.25\). Remember to use the right button of the mouse and click twice in order to pick the region. Accept default tolerance.

4. In a general case, the above procedure may be repeated to specify various element sizes for various polyhedra, regions, or curves. It is sufficient in this example to show the effect of local mesh control on region 21. The specified densities should be reasonable and should leave enough room for transition.

5. Select **Viewing** from the Main menu, **Clear** from Viewing, and **Scale** from Viewing and accept default scale factor.

6. Select **Geometry** from the Main menu, **Part** from Geometry, and **Plot, All,** and **Accept** from Part.

### 3-D Automatic Meshing

The element size for the polyhedron (as well as for all regions) is \(0.50\), except region 21 which has an average element size of \(0.25\).

1. You may want to see the number of elements the program will create along each curve. Select **Geometry** from the Main menu, **Contour** from Geometry, and **List, All,** and **Accept** from Contour.

2. Select **Meshing** from the Main menu, **Mesh Part** from Meshing, and **Linear** from Mesh Part. Pick the part using the left button of the mouse and confirm with a second click.
The program will start generating 4-node tetrahedral elements (the Quadratic option generates 10-node elements).

3. Select **Meshing** from the Main menu, **Element** from Meshing, and **List, All, and Accept** from Element. Note that the generated elements are associated with Element Group (EG) number 1, and Material Property set (MP) number 1. The associated Real Constant Set (RC), and the Element Coordinate system are 1 and -1, respectively. The real constant set is not needed for tetrahedral elements. Refer to Chapter 3 for the element coordinate system.

The generated mesh is shown on the left in Figure 6.7. Note that the mesh is finer in region 21 because of the smaller element size specified. The hidden edge removal flag was activated before plotting elements for convenience. (Refer to Hide Opts menu under Viewing.)

4. A rotated view to show the effect of local mesh control is shown on the right-hand side in Figure 6.7.

![Generated Mesh with Local Control]

Figure 6.7 Generated Mesh with Local Control

5. Select **Viewing** from the Main menu and **Shade** from Viewing. Choose **Flat, Red**, and **Accept**. A shaded view will be displayed.

**Loads and Boundary Conditions**

In Pro/ENGINEER you may assign material properties, loads and boundary conditions to geometric entities before meshing. In the interface, loads, and boundary conditions may be specified only after the mesh has been generated. If a node or an element is deleted, the specified loads and boundary conditions for the deleted node or element will also be deleted.
Before applying loads and boundary conditions, it is recommended that you plot all the model regions.

1. Clear the screen by selecting Clear from Viewing. Select Geometry from the Main menu, Region from Geometry, and Plot, All, and Accept from Region.

2. Select Loads/BConds from the Main menu, Displacement from Loads/BConds, and Define from Displacement. Choose Region, Rigid-Joint, Pick, and Accept. You will be prompted to pick the region you want to fix. Pick region 21. Region numbering may be different for different Pro/ENGINEER releases; please refer to Figure 6.6 for corresponding region numbers.

3. Select Loads/BConds from the Main menu, Pressure from Loads/BConds, and Define, Region, Pick, and Accept from Pressure. You will be prompted for the region number, the pressure value, and the pressure direction. Pick region 79 and enter 2000 for the pressure value.

4. Similar to step 3, apply a pressure of 2000 to region 54.

5. Select Loads/BConds from the Main menu, Pressure from Loads/BConds, and List from Pressure. Verify that the pressure value is listed correctly.

**Submitting the Model for Analysis**

Select Analysis from the Main menu, Static from Analysis, and Run from Static.

Control is now given to STAR, the static analysis module of COSMOS/M. The COSMOS/M CAD Interface screen disappears, and STAR starts to display some information about analysis progress. Once the deformations are calculated, control is given to the STRESS module to calculate stresses. After calculating the stresses, control is given back to COSMOS/M CAD Interface and you may proceed to evaluate the results.

**Postprocessing**

1. Select Postprocessing from the Main menu, Display Plots from Postprocessing, and Plot Displacement from Display Plots.

2. Choose Displacement, Resultant, Filled Contour, Deformed, and Accept from Plot Displacement. Accept the default load case. You will be prompted for a scale factor. The default scale factor scales the maximum displacement to 10% of the model size. Type 1.0 and press Enter.

3. Select Viewing from the Main menu and Boundary Opts from Viewing. Choose Model Boundary, Evaluate Edge, and Accept. Type 70 (70 degrees) and press Enter for the "Tolerance angle to ignore curvature [20] >" prompt. Accept default for the Evaluate Criterion option. In general you may need to experiment with the angle to obtain the desired view. Select Repaint from Viewing. The resulting plot is shown in Figure 6.8.
4. Select **Animate** from the Display Plots menu. Accept default values for prompts. Stop the animation when desired using the **Esc** key.

**Sectional Displacement Plot**

5. Select **Display Plots** from Postprocessing and **Plot Section** from Display Plots. Accept the default value for the orientation of section planes. The generated sections will be normal to the x-axis. Type **12** and press **Enter** in response to the "Number of Section Planes" prompt. Accept other defaults. The resulting plot is shown in Figure 6.9.
The deformation plot may be generated on its own by selecting **Deformation** from the Display Plots menu. Accept default for load case 1 and input 30 for scale factor.

**Stress Results**

6. Stresses may be listed by selecting **Lists** from Postprocessing and **List Stress** from the Lists menu. Choose **Nodal, Set1**, and **Accept**. Note that due to the large number of various stress components that may be listed, the information is divided into two sets. Stresses may be listed at nodes or elements. Accept default load case number and coordinate system.

7. Select **Plot Stress** from Display Plots. Choose **Von Mises, Nodal, Top Face, Filled Contour, Deformed**, and **Accept**. Hit **Enter** to accept the default load case number and scale value printed in the message area. The scale is used for plotting the deformed shape. Note that a scale factor of **1.0** results in using the same scale for plotting the model and the deformation. No scale factor is prompted if the undeformed shape is selected.

8. Select **Viewing** from the Main menu and **Clear** from Viewing.

9. Select **Viewing** from the Main menu, **Windows** from Viewing, and **Window Create** from Windows. Click the left button of the mouse inside the newly created window to activate it.

10. Select **Boundary Opts** from Viewing. Choose **Model Boundary, Evaluate Edge**, and **Accept** from Boundary Opts. Type **45**; and press **Enter** in response to the "**Tolerance angle to ignore Curvature [20] >**" prompt. Accept the default value for Evaluation Criterion option.

11. Repeat step 7.

12. Select **Display Plots** from Postprocessing and **Animate** from Display Plots. Accept default values in the message window. Von Mises stresses will be animated on top of the deformed shape. Press the **Esc** key to stop animation.

**Stress Iso Plots**

13. Select **Plot Iso Surfaces** from Display Plots. Type **12**; and press **Enter** in response to the prompt: "**Number of Iso Planes [1] >**". The resulting plot is shown in Figure 6.10. Remember that ";" indicates accepting all remaining default values.
14. Select **Display Plots** from Postprocessing and **Animate** from Display Plots. Accept default values in the message window. Press the **Esc** key to stop animation.

### 3-D Dynamic Viewing

Dynamic viewing is only available for the HP and SGI platforms, if you are using any other platform, please skip this step.

15. Select **Display Plots** from Postprocessing and **Dynamic View** from Display Plots. Select **Demo on** and **Accept** from Dynamic View. The demo mode will start automatic 3-D viewing of the model.

### Sectional Stress Plots

16. Select **Plot Sections** from Display Plots. Type 0 and press **Enter** to define the intersecting plane. Enter 6 for the "Number of Section Planes" prompt and accept all other defaults. The resulting plot is shown in Figure 6.11.

---

**Figure 6.10  von Mises Iso Plots**
The deformation plot may be plotted on its own by selecting **Deformation** from the Display Plots menu.

**Figure 6.11  Section Plot of von Mises Stresses**
Example 3: Creating and Solving a Model in the Interface

Features:
- Geometry: COSMOS/M CAD Interface
- Meshing: COSMOS/M CAD Interface
- Elements: BEAM3D
- Analysis: Buckling and Frequency Analyses

This example illustrates the procedure to build a simple model and solve it inside the interface. Geometry, mesh, element attributes, loads and boundary conditions will be defined in the interface and CAD programs will not be used in this example. We will calculate the critical buckling load of a simply-supported steel beam of one inch by one inch cross section. The problem is sketched in Figure 6.12.

Creating a Scratch Database

1. To create a model from scratch inside the interface, we need to create a file with GEO extension. The file may have any valid GEOSTAR commands in it, or it may be completely empty. Therefore, create an empty file `ex5.GEO` in your current directory by the command "touch ex5.GEO" from the system level.

2. Type `COSPRO` to start COSMOS/M CAD Interface.
3. Select **Read GEO File** to start Pro/ENGINEER.

4. You will be asked to give the name of the problem you want to open. Type `ex5` and press **Enter**. Since the file `ex5.GEO` is empty, the interface will have nothing to process but will initialize the `ex5` database.

5. COSMOS/M CAD Interface expects geometry to come from Pro/ENGINEER and for this reason the Geometry menu will be shaded and hence not available for selection. Press the function key "F1" to turn off the masking utility.

---

**Geometry Creation**

1. The idea is to create two keypoints and connect them to create a straight line. To create a keypoint, select **Geometry** from the Main menu, **Point** from Geometry, and **Define** and **Accept** from Point. Accept the default value of 1 for the label and the default values of 0.0 for the X-, Y-, and Z-coordinates by pressing **Enter** three times, or typing `; ; ;` and **Enter**. A keypoint will be created at the origin. Press "F1" again to turn on the masking utility.

2. Repeat the above process to create keypoint number 2 with X-, Y-, and Z-coordinates as 50.0, 0.0, and 0.0. Note that the default label is 2 since keypoint number 1 already exists in the database.

3. Select **Curve** from Geometry, and **Define** and **Accept** from Curve. Accept label 1 for the new curve. Pick keypoint 1 for the first keypoint prompt and keypoint 2 for the second keypoint to make the curve.

---

**Element Attributes**

1. Select **Properties** from the Main menu, **Element Groups** from Properties, and **Define Group** from Element Groups. Choose **BEAM3D** and **Accept** from the Option menu. Accept the default label of 1 for this element group and accept all default element group options.

2. To verify your input for the element group, select **Properties** from the Main menu, **Element Groups** from Properties, and **List Groups** from Element Groups. Verify that BEAM3D is the active element group.

3. Select **Properties** from the Main menu, **Material** from Properties, and **Pick Material** from Material. Accept the default value of 1 for the material label, and accept A_STEEL as default and press **Enter** for the material name. Accept the default (FPS) for the units type.

4. To verify your input for material properties, select **Properties** from the Main menu, **Material** from Properties, and **List Sets** from Material.

5. Select **Properties** from the Main menu, **Real Constants** from Properties, and **Define RC Set** from Real Constants. You will be prompted to give the following information:

   ```
   Real Constant Set Number [1] >
   (Press Enter)
   ```
Associated Element Group [1] >
(Press Enter)
Start location for the Real Constant [1] >
(Press Enter)
Number of Real Constants to be Entered [10] >
(Press Enter)
RC1: Cross Sectional Area >
(Type 1.0 and press Enter)
RC2: Moment of Inertia about the element Y-axis (Iy) >
(Type 1/12 and press Enter)
RC3: Moment of Inertia about the element Z-axis (Iz) >
(Type 1/12 and press Enter)
RC4: Depth of the beam (y-axis)[0] >
(Type 1 and press Enter)
RC5: Width of the beam (z-axis)[0] >
(Type 1 and press Enter)
RC6: End release code (node 1) [0] >
(Press Enter)
RC7: End release code (node 2) [0] >
(Press Enter)
RC8: Torsion constant (J) [0] >
(Type 0.281667 and press Enter)
RC9: Shear factor in the elem. y-axis >
(Type 5/6 and press Enter)
RC10: Shear factor in the elem. z-axis >
(Type 5/6 and press Enter)

A message saying that you may need 4 more real constants will be printed on your screen.

The Define RC Set option in the Real Constants submenu takes 10 entries at a time. Some element groups may require more than 10 entries. For this problem, no more real constants are needed, but we will demonstrate the general case where more than 10 real constants are needed. Select the Define RC Set option from the Real Constants submenu one more time.

Real Constant Set Number [2] >
(Do not accept default, type 1 and press Enter)
Associated Element Group [1] >
(Type 1 and press Enter)
Start Location for this Entry [11] >
(Note that the default is 11, press Enter)
Number of Real Constants to be Entered [4] >
(Press Enter)
RC11: Temperature difference in the element y-axis [0] >
(Press Enter)
RC12: Temperature difference in the element z-axis [0] >
(Press Enter)
RC13: Orientation angle Theta [0] >
(Press Enter)
RC14: Torsional constant (CTOR) [1] >
(Type 0.234722 and press Enter)
Meshing the Curve

Before we mesh the curve to generate BEAM3D elements, we need to create a third keypoint that will be used to define the element coordinate system. The third point determines the orientation of the cross section in space.

1. Select **Geometry** from the Main menu, **Point** from Geometry, and **Define** and **Accept** from Point. Create keypoint number 3 at coordinates (0, 50, 0).

2. Select **Meshing** from the Main menu and **Mesh Curve** from Meshing. You will be prompted as follows:

   Pick/input the curve >
   (Pick curve 1)

   Number of nodes per element >
   (Type 3 and press Enter)

   Number of elements on curve >
   (Type 20 and press Enter)

   Pick keypoint to define principal axis >
   (Pick keypoint number 3)

The generated elements are shown in Figure 6.13. Node labels may be turned on from the Status Table.

![Figure 6.13  Meshed Model of the Beam](image)
Specifying Loads and Boundary Conditions

We will fix node 1 in all translations and rotations except rotation about the Z-axis. Node 21 will be fixed against translations in the Y- and Z-directions, and rotations about the X- and Y-axes. With the specified boundary conditions, nodes 1 and 21 are free to rotate about the Z-axis, node 21 is free to move along the X-axis (roller). A force in the X-direction will be applied to node 21.

1. Select **Loads/BConds** from the Main menu and **Displacement** from Loads/BConds. Choose **Define, Nodes, Pin-Joint, Pick**, and **Accept**. Pick node 1 to fix it against all translations. Repeat the command and choose **Define, Nodes, RX-restraint, Pick**, and **Accept**. Pick node 1 (at the origin) and specify a value of 0.0 again to fix it against RX rotations. Repeat the same procedure but highlight **RY-restraint** to fix node 1 against the RY rotation. See Figure 6.14.

2. Select **Loads/BConds** from the Main menu and **Displacement** from Loads/BConds. Choose **Define, Nodes, UY-restraint, Pick**, and **Accept**. Pick node 21 and specify a value of 0.0 to fix it against UY translation. Repeat to fix node 21 against UZ, RX, and RY.

3. Select **Loads/BConds** from the Main menu and **Force Moment** from Loads/BConds. Choose **Define, Nodes, FX-Force, Pick**, and **Accept**. Pick node 21 and type -1000 and press **Enter** to specify the value of the force.

```
Figure 6.14  Loads and Boundary Conditions on the Beam
```

Buckling Analysis

1. Run buckling analysis by selecting **Analysis** from the Main menu and **Buckling** from Analysis. Choose **Subspace, No Eigenvalue Shift, No Soft Spring, No Sturm Sequence**, and **Run**. Type 2 and press **Enter** in response to the number of eigenvalues to be calculated. Accept the default value for the Form Stiff matrix flag option. Control will be given to DSTAR to start calculating the buckling load factors. When the analysis is
completed, control is given back to COSMOS/M CAD Interface. The output file will list the buckling load factor as eigenvalues.

2. We may list the two calculated eigenvalues we requested by selecting Lists from Postprocessing and List Freq/Buckling from Lists. The buckling load factors should now be listed on the screen as shown below.

<table>
<thead>
<tr>
<th>Eigenvalue #</th>
<th>Buckling Load Factor</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>9.859889e+00</td>
</tr>
<tr>
<td>2</td>
<td>3.932433e+01</td>
</tr>
</tbody>
</table>

### Buckling Load Calculations

To calculate the buckling load, multiply the buckling load factor by the loads specified.

\[
\text{Buckling load} = (1000.0)(9.85988) = 9859.88 \text{ lbs}
\]

Similarly the critical buckling load for the second mode is

\[
(1000)(39.3243) = 39,324.3 \text{ lbs}
\]

The theoretical lowest buckling load for this beam is equal to

\[
\pi^2EI/(L^2) = (\pi^2)(30E6)(1/12)/(50^2) = 9869.6 \text{ lbs}
\]

You may animate the buckling mode shapes using the Animate command in the Display Plots submenu in the Postprocessing menu. In most cases, only the lowest buckling value is of practical interest.

Note that if we apply a tension force instead of compression, the program will calculate negative eigenvalues, indicating that the load direction must be reversed to realize buckling. Remember that if multiple load values, directions and types exist, all loads must be multiplied by the corresponding eigenvalues to calculate buckling loads.

### Calculating Natural Frequencies and Mode Shapes

1. Select Analysis from the Main menu and Frequency from the Analysis menu. Select Subspace, Sturm sequence, No Eigenvalue Shift, No In-Plane Stiffness, No soft Spring, Lumped Mass, and Run (refer to Chapter 3 for details). The message "Number of Frequencies [1] >" will be prompted, enter 4 and accept default for "Modal Acceleration Flag". Accept default values for the rest of the options. The program will calculate the lowest 4 frequencies and the corresponding mode shapes.

Control is now given to DSTAR, the frequency and buckling module of COSMOS/M. The COSMOS/M CAD Interface screen disappears, and DSTAR starts to display some information about the model and the progress of the analysis. Once the frequency analysis is completed, control is given back to COSMOS/M CAD Interface.
Postprocessing Frequency Analysis

1. We may list the four frequencies we requested by selecting Lists from Postprocessing and List Freq/Buckling from Lists. The frequencies should now be listed on the screen. The resulting list is shown below.

<table>
<thead>
<tr>
<th>Frequency#</th>
<th>Frequency (Rad/sec)</th>
<th>Frequency (Cycles/sec)</th>
<th>Period (Seconds)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>2.30916e+02</td>
<td>3.67514e+01</td>
<td>2.72099e-02</td>
</tr>
<tr>
<td>2</td>
<td>5.22424e+02</td>
<td>8.31463e+01</td>
<td>1.20270e-02</td>
</tr>
<tr>
<td>3</td>
<td>9.22279e+02</td>
<td>1.46785e+02</td>
<td>6.81267e-03</td>
</tr>
<tr>
<td>4</td>
<td>1.43577e+03</td>
<td>2.28509e+02</td>
<td>4.37619e-03</td>
</tr>
</tbody>
</table>

2. To plot a mode shape, select Display Plots from Postprocessing and Deformation from Display Plots. A message asking for the mode shape number will be prompted "Mode Shape Number [1] >", type 1, and hit Enter to plot mode shape number 1. Next the message "Scale Factor [1.35093] >" is displayed, hit Enter to accept the default scale. This scale normalizes the maximum deformation in the model to 10% of the model size. You may repeat the process to plot mode shapes 2, 3, and 4. Figure 6.15 shows the 4 mode shapes.

![Figure 6.15 Mode Shapes](image)

3. Select Display Plots from Postprocessing and Animate from Display Plots. Accept default values in the message window. The first mode shape will be animated. You may continue to animate the other mode shapes as desired.

Note that the results from frequency analysis overwrite the results from buckling analysis.
Example 4: Transient Thermal Analysis of a Mixed Elements Model

Features:
- Geometry: Pro/ENGINEER
- Meshing: Pro/MESH
- Elements: TETRA4 and SHELL3T
- Analysis: Steady and Transient Heat Transfer Analysis

This example illustrates the procedure for applying and running transient thermal analysis. The mesh for this example was generated by Pro/MESH with two types of elements (element groups), TETRA4 and SHELL3T. The geometry and mesh information is stored in the file ex6.prt, and the material information is in the file ex6.mat.

Preprocessing in Pro/ENGINEER

1. Copy the files ex6.prt and ex6.mat to your work subdirectory as follows:
   ```
   CP $COSMOSM/PROBS/ex6.prt .
   CP $COSMOSM/PROBS/ex6.mat .
   ```

2. Type cospro to start COSMOS/M CAD Interface.

3. Select CAD Program to invoke Pro/ENGINEER.

4. Select Mode from the Pro/ENGINEER Main menu, Part from Mode, and Search/Retrieve from EnterPart.

5. Select ex6.prt to retrieve this example. The model will be displayed on the screen. Material properties were defined in Pro/ENGINEER and will be read from the ex6.mat file.

6. Select FEM from Part, Make Model from FEM, Mixed Mesh from FEM Mesh, and Triangles from Shell Type. Confirm the use of default global min mesh control value of 10.0. Wait for Pro/MESH to finish meshing.

7. Select Output Model from FEM. Choose COSMOS/M, Linear, Thermal and Accept from the Output Mesh Option menu. Accept the default file name (ex6.cos) prompted in the dialog window.

8. Select Exit from the Main menu to exit Pro/ENGINEER and return to COSMOS/M CAD Interface. The model will be automatically loaded under the same name.

Inside the Pro/ENGINEER Interface

1. Select Rotate from the Viewing menu, specify x, y, and z angles of -40, 0, and 0. Accept other defaults and clear the screen using Clear from Viewing.
2. Select **Element** from Meshing and **Plot, All**, and **Accept** from Element. The generated plot is shown in Figure 6.16.

![Figure 6.16 Model Displayed in COSMOS/M CAD Interface](image)

3. Select **Properties** from the Main menu, **Element Groups** from Properties, and **List Groups** from Element Groups. Verify that two element groups (SHELL3T and TETRA4) exist in the database.

4. Select **Properties** from the Main menu, **Material** from Properties, and **List Sets** from Material.

5. Select **Properties** from the Main menu, **Real Constants** from Properties, and **List RC Sets** from Real Constants. Note the listed thickness for element group number 2 (SHELL3T), and note that the real constant set number 1 (associated with TETRA4 elements) is an empty set.

6. To view the model, select **Viewing** from the Main menu, and **Shell Shade** and **Clear** from Viewing.

Select **Meshing** from the Main menu, **Element** from Meshing and **Plot, All**, and **Accept** from Element. Shell Shade gives different colors to top and bottom faces. It is important for stress results to make sure that top and bottom faces of adjacent shell elements are matching. Otherwise the averaging process will mix up bottom and top face stresses. If the mesh is generated in Pro/MESH, then the orientation is such that the top face is at the outer side of the polyhedra, and the bottom is at the inner side.

7. Select **Meshing** from the Main menu, **Element** from Meshing, and **List, All**, and **Accept** from Element. Note the four element attributes (EG, MP, RC, and EC), and that 4 nodes are defined for elements associated with EG 1, and that 3 nodes are defined for elements associated with EG 2.
Running the Analysis

1. Before running a transient analysis, we need to specify the duration of interest for the solution. Select Load Options from Loads/BConds and Times from Load Options; respond as follows:

   **Starting time for the analysis** >
   (Type 0.0 and hit Enter)

   **Ending time for the analysis** >
   (Type 40.0 and hit Enter)

   **Time step size [0.4]** >
   (Type 4.0 and hit Enter)

2. Select Analysis from the Main menu, Thermal from Analysis, and Run Thermal from Thermal. Choose Transient, 1st Ord FFE Solver, and Run. The FFE heat transfer module will solve for the 10 specified time steps.

Postprocessing

1. Select Lists from Postprocessing and List Thermal from Lists. Type 10 in response to "Time Step Number", accept the default set number and press Enter to list results for time step number 10.

2. To generate a contour plot for temperatures, select Display Plots from Postprocessing, and Plot Thermal from Display Plots. An option menu will be displayed. Select Temperature, Filled Contour and Accept. Figure 6.17 shows the generated plot.
4. You may generate XY-plots for time versus thermal results using the **Graph** menu from Postprocessing.
Example 5: Static Analysis of a Bracket

Features:
- Geometry: MicroStation Modeler (from Bentley Systems)
- Meshing: COSMOS/M CAD Interface
- Elements: TETRA4
- Analysis: Static Analysis

This example considers the linear static stress analysis of a bracket whose geometry is created in MicroStation Modeler by Bentley Systems. The following figure shows the geometry of the bracket.

![Figure 6.18   Geometry of the Bracket for Static Stress Analysis](image)

Within MicroStation Modeler

1. Create the Bracket model in MicroStation Modeler, as described in the MicroStation Modeler User’s Guide, Version 1, Chapter 4.

2. Choose File, export and IGES to create an IGES file “bracket.igs”. Note that the IGES file can be created in any desired directory using File and Export As in the Export IGES file window and then clicking on the Export button. You may export the IGES file to any work directory.

3. Exit from MicroStation Modeler.

Starting COSMOS/M CAD Interface

Before starting the CAD Interface, make sure the file “bracket.igs” exists in the current work directory.
1. Start the CAD Interface using the command **COSPRO** from the working directory.

2. Choose **Main** and **Read IGES File**. Choose "BSI" from the CAD Program menu and then select "braket" from the available menu.

3. You will now see the CAD Interface screen with the bracket displayed in pink. The geometric entities of the MicroStation Modeler have now been translated into keypoints, curves, contours, regions, a polyhedron, and a part.

   COSMOS/M CAD Interface has a user-friendly flow chart navigator which guides you through the various steps required for the analysis. In case of doubt and “what next” types of questions, you can always refer to this flow chart. The flow chart can only be accessed from the main menu. It displays rectangular and diamond blocks representing different steps of analysis. You can click on any block of the flow chart to access a menu containing the commands applicable to that block.

4. To view the bracket in the correct position, choose **Main, Viewing, and Rotate**. The dialog area will prompt for the following:

   - X-Rotation [0] > -90
   - Y-Rotation [0] > 0
   - Z-Rotation [0] > 90

   Accept the default values for the rest of the options.

---

**Element Attributes**

1. Choose **Main, Properties, Element Groups, and Define Group**. A menu of available types of elements is displayed. We will use tetrahedral element TETRA4. Choose **TETRA4** and **Accept**. The following message appears in the dialog area:

   ```
   Element Group [1] > 1  
   (Type 1 to specify element group label)
   ```

   Press **Enter** to accept all default values in the command.

2. Choose **Main, Properties**, and **Material**. Pick a material from the existing material library by choosing **Pick Material**. The following message appears in the dialog area:

   ```
   Material Set Number [1]>  
   ```

   Press **Enter** to accept all defaults to specify material as Alloy Steel and units in FPS.

3. Choose **Main** and **Flow Chart**. Click on **Import Geometry from CAD System** and then **Linear Static**. Notice that the **Define Element Group** and **Define Material** are shown in green, indicating that they have been already completed. Clicking the left mouse button on these options will list the defined property on the screen. Any step which is yet to be completed is shown in red.
Meshing

1. Choose Main, Meshing, Mesh Density, Polyhedron, and Accept. The following prompts appear in the dialog area:

   Avg. element size > 7
   Tolerance [.0001] > .01

   For most cases, 10% of the element size is recommended for the tolerance value. An element size of 7 in. seems to be reasonable for this model. The specified element size should not be smaller than \( \frac{1}{10} \)th of the tolerance for polyhedron, and it should not exceed the smallest model dimension. You can list the polyhedron to verify the element size.

2. Choose Main, Meshing, Mesh Part, and Linear and select the part to be meshed twice. This leads to the generation of 4-noded tetrahedral elements in accordance with the defined Element Group TETRA4.

3. After the meshing process is complete, select Main, Viewing, and Shade. Choose Flat, Red, and Accept. A shaded view of the model appears on the screen.

Loads and Boundary Conditions


   We want to apply rigid boundary conditions on the inside surfaces of the holes on the two side walls of the bracket. Move the mouse arrow to region 29 and click the left button. The picked region will be highlighted and its label will be displayed in the dialog area. If the picked region is not 29, click the right button of the mouse until region 29 is picked and then confirm by pressing the left mouse button. You can always type-in the region number if known. Repeat this procedure for regions 15, 16, 17, 18, 30, 31, and 32.

2. Choose Main, Loads/Bconds, Pressure, and Define. Select Region, Pick, and Accept. We need to apply a pressure of 1000 psi in the vertical direction on the sloped region of the hole with the cut (on bottom slab). The following message appears in the dialog area.

   Pick/Input Region > 13
   (Using the same procedure as before, pick region 13)
   Pressure Magnitude > 1000
   Pick/Input Ending Region >
   (Enter to accept defaults for next four prompts)

   Repeat this procedure for regions 14 and 33. Pressure is displayed with the arrows pointing in the vertical direction. The following figure shows the applied boundary conditions and loads on the finite element mesh of the bracket.
Performing Analysis

1. Choose Main, Analysis, and Static. When you click on Run, the control is now given to the STAR, the static analysis module of COSMOS/M. The Interface screen is replaced by the analysis screen. Once the static analysis run is completed, control goes back to the CAD Interface.

Postprocessing

Results of the analysis are now available in the database. The file bracket.OUT is an ASCII file that contains the results from the static run.

Displacement Results

1. Displacements are listed by choosing Main, Postprocessing, Lists, and List Displacement. Strains and stresses may be listed by selecting List Strain and List Stress from the List menu, respectively.

2. Choose Main, Viewing, and Boundary Opt. A submenu for the Boundary Opt is displayed, select Model Boundary, Evaluate Edge, and Accept. Press Enter to accept default values for the options appearing in the dialog box. These options instruct the program to evaluate edges of the model and will generate an edge plot after you execute Repaint from Viewing.

3. Next, we will generate a colored contour plot for the resultant displacement. Select Display Plots from Postprocessing. Select Plot Displacement from the Display Plots submenu. Select Displacement, Resultant, Filled Contour, Deformed, and Accept from this submenu. Press Enter to accept the default load case number. Hit Enter to accept the default value for the scale factor “Scale Factor [422.537]>”. A color plot showing the displacement levels on the deformed shape will be displayed.
4. You may reduce the figure on the screen by selecting Main, Viewing, and Scale. The message "Scale Factor \([0]\)" appears, type 0.8 and press Enter to reduce the figure to 80% of its current size. The following figure shows the plot of resultant displacement contours.

![Resultant Displacement Contours](image)

**Figure 6.20 Resultant Displacement Contours**

5. To generate a section plot, select Main, Postprocessing, Display Plots, and Plot Sections. The following messages will appear in the dialog window.

Orientation of Section Planes 0=X, 1=Z, 2=Z, 3=three nodes \([0]\)>
Number of Section Planes to Be Plotted \(1-12\) \([1]\)>
Specify section planes positions \(1=yes, 0=\text{defaults} \([0]\)>
Shape flag \(0=\text{UNDEF, 1=DEF} \([0]\)>

The section plot generated is as shown below.

![Section Plot of the Resultant Displacement Contours](image)

**Figure 6.21 Section Plot of the Resultant Displacement Contours**
Stress Results

1. Stresses may be listed by selecting Main, Postprocessing, Lists, and List Stress.

2. Select Main, Postprocessing, Display Plots, and Plot Stress. Choose Von Mises, Nodal, Filled Contour, Deformed, and Accept. Note that the "Top/Bottom Face" and "Membrane/Bending Stress" flags are relevant to shell elements only. Hit Enter to accept the default load case number and scale values printed in the message area. The scale factor is used for plotting the deformed shape to an exaggerated scale. Note that a scale factor of 1 results in using the same scale for plotting the model and the deformation. The figure below shows the von Mises stress contours.

![Von Mises Stress Contour Plot of the Bracket](image)

Figure 6.22 von Mises Stress Contour Plot of the Bracket

3. Select Main, Postprocessing, Display Plots, and Animate and accept default values in the message window. The von Mises stresses will be animated on top of the deformed shape. Note that animation will be faster in smaller windows. You may stop the animation by pressing the Esc key.

4. Similar to section plots, you may also plot isosurface plots of stresses. Select Plot Iso Surfaces from Display Plots. Type 4 and press Enter to answer the prompt: "Number of Iso Planes [1]">”, accept other defaults.

5. The deformation plot may be plotted on its own using the Deformation command from the Display Plots menu.

Postprocessing in Multiple Windows

1. Select Main, Viewing, Windows, and Window Create. Click the mouse inside the newly created window.

2. Select Shade from Viewing. Choose Flat, Red, and Accept to activate shading.
3. Select **Viewing** and **Boundary Opts**. Choose **Model Boundary, Evaluate Edge**, and **Accept**. Type **45**; and press **Enter** to answer the “Tolerance Angle to Ignore Curvature [20]” message.

4. Select **Main, Meshing, Elements, Plot, All**, and **Accept**. A shaded element plot will be generated in the active window.
Example 6: Thermal Analysis on a Piston

Features:
- Geometry: MicroStation Modeler (from Bentley Systems)
- Meshing: COSMOS/M CAD Interface
- Elements: TETRA4
- Analysis: Thermal Analysis

This example considers the steady-state thermal analysis of a piston. The geometry of the piston will be created in the MicroStation Modeler CAD system.

![Figure 6.23  Geometry of the Piston for Thermal Analysis](image)

** Within MicroStation Modeler **


   To make the model more interesting, we will:
   
   a. Create a vertical blind hole (1.5 dia.) from the bottom face to a depth of 2 units.
   
   b. Create a horizontal through hole (.4 dia.) from the outer surface, center of the hole is at a height of 0.5 units from the base.

2. Choose **File, export, and IGES** in MicroStation to create an IGES file “piston.igs”. Note that the IGES file can be created in any desired directory using **File** and **Export As** in the Export IGES File window, and then clicking the **Export** button.

3. Exit from MicroStation Modeler.
Starting COSMOS/M CAD Interface

Again, before starting CAD Interface, make sure that file “piston.igs” exists in the working directory.

1. Start CAD Interface by typing **COSPRO** from the working directory as before. Choose **Main** and **Read IGES File**. Choose "BSI" from the CAD Program menu and then select "piston" from the available menu.

   The CAD Interface screen should come back with the piston displayed in pink.

Element Attributes

1. Choose **Main**, **Properties**, **Element Groups**, and **Define Group**. Pick **TETRA4** and **Accept** from the available menu. The following message appears in the dialog area:

   
   Element Group [1] >
   (Press Enter to accept all default values in the command)

2. Choose **Main**, **Properties**, and **Material**. Select **Pick Material**, accept the default material set number (1), type **Aluminum** as material name and select FPS system of units.

Meshing

1. Choose **Main**, **Meshing**, **Mesh Density**, and **Polyhedron**. Click **Accept**. The following messages appear in the dialog area:

   
   Avg. element size > 0.15
   Tolerance [.0001] > 0.015
   (Enter 10% of the element size)

2. Choose **Main**, **Meshing**, **Mesh Part**, and **Linear**.

3. After the meshing process is completed, select **Main**, **Viewing**, and **Shade**. Choose **Flat**, **Red**, and **Accept**. A shaded view of the model appears on the screen. If you wish, you can rotate the piston by choosing **Main**, **Viewing**, and **Rotate** and then enter -90° for x-rotation.

Loads and Boundary Conditions

In this problem, we need to study the temperature distribution in the piston when it is subjected to a constant temperature on the top face and convection on the bottom face.


2. Choose **Main**, **Loads/Bconds**, and **Convection**. Select **Define**, **Regions**, **Pick**, and **Accept**. Pick region 28. Enter 32 for convection coefficient and 600 for bulk temperature to specify ambient temperature as 600° F.
The following figure shows the finite element mesh of the piston.

![Finite Element Model of the Piston](image)

**Figure 6.24  Finite Element Model of the Piston**

### Performing Analysis

1. Choose **Main**, **Analysis**, **Thermal**, and **Run Thermal**. Select **Steady State**, **Direct Solver**, and **Run**. Control is now given to FFE Thermal, the heat transfer module of COSMOS/M.

### Postprocessing

Results of the heat transfer analysis are now available in the database.

1. To list the results, choose **Main**, **Postprocessing**, **Lists**, and **List Thermal**. Accept the default value and set number for time step in the dialog area. Results will be listed on the screen. Press **Enter** or click the left button of the mouse to display more pages, or press the **Esc** key to abandon listing.

2. To generate contour plot for temperatures, select **Main**, **Postprocessing**, **Display Plots**, and **Plot Thermal**. Choose **Filled Contour** and **Accept**.

The following figure shows the temperature contour plot of the piston.
Figure 6.25  Temperature Contour Plot
Example 7: AutoCAD Part

Features:
   Geometry: AutoCAD
   Meshing: COSMOS/M CAD Interface
   Elements: TETRA4
   Analysis: Static Analysis

The original part used in this example was created using AutoCAD and was saved as a SAT file.

Within MicroStation Modeler

1. Open a new design file called "cylinder".

2. Choose File, Import, and SAT. MicroStation pops up a new window “Import ACIS SAT file”. Give the name and whole path of the SAT file to be imported (in this case, cylinder.sat). Press OK.

3. The cylinder has now been imported into Modeler. Choose View, Fit, and Active Design and click, using the left mouse button, in any one of the views.

4. Choose Pallettes, Modeler, CreateFeature and use “Create Chamfer Feature” to create a chamfer of length 1 inch on the top outer edge of the cylinder. Note that you can add new features to this SAT model but you are not allowed to edit the original SAT model.

5. Choose File, export, and IGES to create an IGES file “cylinder.igs” in any desired directory as explained in Example 1.
6. Exit from MicroStation.

Starting COSMOS/M CAD Interface

Again, before starting CAD Interface, make sure that file “cylinder.igs” exists in the working directory.

1. Start CAD Interface by typing **COSPRO** from the working directory.

2. Choose **Main** and **Read IGES File**. Choose "BSI" from the CAD Program menu and then select "cylinder" from the available menu.

3. The cylinder model will now be displayed on the screen.

Element Attributes

1. Choose **Main**, **Properties**, **Element Groups**, and **Define Group**. Choose **TETRA4** and accept all other default values by pressing **Enter**.

2. Choose **Main**, **Properties**, and **Material**. Choose **Pick Material** to pick Alloy Steel (A_STEEL) in FPS units.

Meshing

1. Choose **Main**, **Meshing**, **Mesh Density**, and **Polyhedron**. Click **Accept**. The following message appears in the dialog area:

   Avg. element size >
   (Type .25)
   Tolerance [.0001] >
   (Type .025, 10% of element size)

2. Choose **Main**, **Meshing Part**, and **Linear**.

3. The user can look at the shaded views as described in earlier examples.

Loads and Boundary Conditions

The cylinder is rigidly fixed at the top. A downward pressure is applied on the bottom face of the cylinder.

1. Apply rigid constraints to region **10** (top face) using the procedure described in the previous examples.

2. Apply a pressure of **–1000 psi** on region **16** (bottom face).
Performing Analysis

1. Choose **Main**, **Analyze**, **Static**, and **Direct Solver**. Click **Run**. The Interface screen is replaced by the Analysis screen. Once the static analysis is completed, control goes back to the Interface.

Postprocessing

One can look at the **Displacements** and **Stress Contour Plots** by following the procedures described in the previous examples.

![Figure 6.27 Resultant Displacement Contours](image_url)
Chapter 7

About Finite Elements
(This page is intentionally left blank)
Introduction

Knowledge of finite elements is important for using COSMOS/M CAD Interface. While the finite element method is extremely useful and powerful, it is imperative that care is taken in the application of this technique to avoid incorrect results and/or interpretations. This chapter presents some information on the fundamental concepts of finite element analysis and attempts to answer the following general and basic questions about the finite element technique:

- What is finite element analysis and how is it used?
- What factors do I need to consider when I design my finite element model?
  - how do I create the geometry of the model?
  - how do I create the finite element mesh?
  - how do I select the element types?
  - how do I specify the boundary conditions?
  - how do I specify the loading conditions?
  - how do I specify the material properties?
- Is my problem suitable for running on a desktop computer?
- What can go wrong with the analysis?
- How do I obtain accurate results?
- How do I interpret the results of my analysis?

A First Approach To Finite Elements

To illustrate the concept of finite elements, consider the example of finding the area of a circle. Assume that you did not know how to calculate the area of a circle but knew the formula for the area of a triangle.

Since the area of a triangle can be calculated, you can divide the circle into triangles, in two distinct ways, as shown below. To start with, use four triangles. In one method, the triangles are inscribed within the circle, leaving some areas of the circle unaccounted for. This results in an underestimation of the area of the circle. In the second approach the circle lies within the polygon formed by the four triangles. In this case, you will be overestimating the area of the circle. See Figure 7.1.
Figure 7.1  Approximation of the area of a circle with four triangles. Shaded areas indicate excessive areas or areas not accounted for.

If you continue inscribing the circle with an increasing number of triangles (for example six), you will notice that the error, represented by the shaded areas, decreases. See Figure 7.2. If you double the number of triangles, the errors are hardly noticeable. If you continue increasing the number of triangles, you can calculate the area almost exactly. Thus, you can calculate the area of a circle by using a geometric element (a triangle) for which the solution (equation) is known. The triangle acts as a basic element in calculating unknown areas.

Figure 7.2  The approximation of the area of a circle with increasing numbers of triangles.

Thus, for solving problems which do not have a handbook type solution, you can use an approach which uses basic elements for which you have a known solution. The finite element method is based on the same principle. It is a numerical technique for solving mechanical problems in which the structure (or continuum) is subdivided, or idealized, into simple elements whose behavior is very well established. Finite element methods are predominantly used to perform computer based analyses of static, dynamic, electrical, or thermal behavior of physical systems, structures, and components. They are used primarily when hand calculations are inappropriate or cannot provide sufficiently accurate or detailed results. In computer aided design, they are used to integrate the design and the analysis processes.
A More Rigorous Approach

Cantilever Example

Figure 7.3 shows a practical problem of a thin steel sheet loaded in its own plane (Irons & Shrive, 1983). The objective of the analysis is to determine the free end vertical deflections and the reactions at the fixed support. The cantilever is composed of four elements, marked i-iv, which comprise what is called the mesh. When the cantilever deforms, the corner of the elements, known as nodes move. The nodes are numbered 1-10 in this case. The problem assumes that the model can have two unknown deflections at each node, one in each of the x- and y-directions: $u_1, v_1, u_2, ... , v_{10}$, at nodes 1-10, twenty variables in all. These unknown deflections at nodes are called degrees of freedom. The degrees of freedom are the primary unknowns to be determined by the analysis; they may be displacements, rotations, temperatures, pressures or voltages depending on the analysis type. For these twenty variables, the computer must generate and solve twenty simultaneous equations. The coefficients of $u_1, ..., v_{10}$ on the left-hand sides of the equations are called stiffnesses, like those in a simple spring. The problem can then be defined as Stiffness multiplied by Deflection equals Force ($kd = f$), where the right-hand side represents the applied loads. The stiffness, $k$, of each element is an indication of how strong the element is with respect to deformations. The stiffness terms are functions of the material and geometric properties of the structure.

Figure 7.3  A finite element representation of a cantilever shown in side elevation, with a vertical end load. The roman numerals are element numbers. The other numbers define nodes, or points, where the deflection in the x- and y-directions are the variables (Irons & Shrive, 1983).

Of the twenty equations, we eliminate four when we force $u_1, v_1, u_2, v_2$ to be zero due to fixing the left end so it can not move. This condition is known as a boundary condition. Note that the elements are joined and have nodes 3 through 8 in common. For example, $u_5$ is the same in element ii and in element iii. It has to be the same, because the material is continuous across the element boundaries.
After imposing the boundary conditions, there will be sixteen equations in the system $K\mathbf{D} = \mathbf{F}$ to be solved. $K$ is called the structural stiffness matrix, $\mathbf{D}$ is the displacement vector, and $\mathbf{F}$ is known as the load vector. The load vector in this example has only one value, $P$, applied at node 10 in the vertical direction, and the rest of the values are zero. The system of equations, $K\mathbf{D} = \mathbf{F}$, is linear and can be solved easily. Further, $K$ is symmetric and banded, i.e., it has nonzero items clustered about the diagonal. The unknown quantities are always the displacements $\mathbf{D}$. Once the displacements are solved using an equation solver, the forces (and stresses) at each node can be computed.

The results from the analysis can then be compared with known solutions, when possible. In this example, the vertical deflection at the free end (under the load, in nodes 9 and 10) or the reactions at the fixed end (nodes 1 and 2) can be verified with theoretical solutions. However, it is not always possible to compare with theoretical solutions, so engineering judgment must be used to validate the solution.

Thus the principle behind the finite element approximation used in the cantilever example is the same as illustrated in finding the area of a circle. However, there is one important difference you should be aware of. The equation for finding the area of a triangle is considered exact, whereas the equations describing the behavior of finite elements are at best accurate approximations.

**A Formal Definition**

Finite element analysis is broadly defined as a numerical method for approximating the governing differential equations of any continuous system (Baran, 1988). The principle of finite elements is a rigorous mathematical theory based on calculus of variations, energy theorems, principles of elasticity, and other equations of physics and engineering. It is a powerful method for the numerical solution of problems in stress analysis, heat transfer, fluid flow, electric fields, and many other applications.

In general, the objective of finite element analysis is to approximate (with a sufficient degree of accuracy) the unknowns (such as deflections or temperatures) of a governing differential equation at selected locations on a physical system or structure. A mathematical model is created for the structure, which has been divided into nodes and elements. Equations are then generated for the model and solved for each node.

The governing differential equations can define a wide variety of physical phenomena. Poisson's equation, for example, describes deflections of a membrane, bending of a prismatic beam, heat conduction with sources, and many other physical phenomena.

**Finite Element Idealization And Analysis**

Two distinct types of approximations were made in performing the finite element analysis of the cantilever example:

1. The cantilever was replaced by an assemblage of quadrilateral elements which accurately described the geometry, i.e., geometric idealization.

2. The physical nature of the material was approximated by means of mathematical models (e.g., stress-strain equations like Hooke's law), i.e., material idealization.
These two idealizations essentially make up a finite element model. Combined with a solution algorithm after applying the boundary conditions and loads, the finite element model can accurately approximate the behavior of a structure. The accuracy of finite element analysis results depends to a large extent on the accuracy of these idealizations. The solution algorithm also has a significant effect on the accuracy.

With the advancements made in computational geometry, geometric idealization has become relatively straightforward and accurate. Today, almost all types of practical geometries can be accurately represented in a finite element model. Further, many finite element analysis packages include features for creating and meshing the geometry with few commands.

The advancements made in material idealization have also been quite significant. However, the level of accuracy is less than that of geometric idealization. For many materials, the mathematical description is an approximation, at best. In many finite element analyses, linear material models are used to obtain satisfactory results. More accurate material models are usually nonlinear and are computationally intensive.

The practical application of finite element analysis includes a number of disciplines in engineering and applied sciences. As a result, the solution algorithms differ for each type of application. In general, the analysis can be broadly classified as:

- Linear analysis
- Nonlinear analysis

In a linear analysis, the unknown degrees of freedom are computed in a direct manner whereas in a nonlinear analysis, they are computed iteratively. Nonlinearity could be due to different sources. The two principal sources are: nonlinear material properties (which require a materially nonlinear analysis), and large displacements (which require a geometric nonlinear analysis) causing the deformations due to loads to be large. Although all practical problems are nonlinear in nature, in many cases linear analysis provides a good approximation. Frequently, linear analysis is carried out as a first step, followed by a more rigorous nonlinear analysis. However, for many problems, due to their geometric and material properties, nonlinear analysis is necessary.

In a linear analysis, the equations to be solved can be put in the form:

\[
[A]{x} = {F}
\]

where \([A]\) is the system (or structural) stiffness matrix, \({x}\) is a vector containing the unknown degrees of freedom, and \({F}\) is a vector of the known quantities (applied loads, temperatures etc.). The matrix \([A]\) is symmetric, sparse, and banded. The above system of equations is direct and explicitly solvable. Well established procedures are available to solve such equations using minimum computer storage and maximum efficiency. Solution algorithms used in Structural Research and Analysis Corporation are continuously updated to reflect the state-of-the-art techniques.

In a nonlinear analysis, the equations take the form:

\[
[A(x)]{x} = {F}
\]
Note that the system matrix \([A]\) is now a function of the unknown degrees of freedom. As a result, the system of equations are not explicitly solvable. The solution is usually carried out iteratively. The iterations may converge to, or diverge from, a final solution. The user is often expected to provide tolerances for the solution convergence. Engineering judgment is very important for nonlinear problems.

**Basic Steps In Finite Element Analysis**

In order to model and analyze a structure using finite elements, you need to execute the following steps (Kardestuncer, 1987):

1. Identify the problem, its geometric and material properties, and decide if it can be modeled with the finite element software available.

2. Create the problem geometry using a CAD system or a geometrical modeler.

3. Mesh the problem geometry with the user's choice of elements, spacing of nodes, and arrangement.

4. Apply boundary conditions (constraints) on the finite element model.

5. Define the loads on the model.

6. Define the material properties.

7. Submit the completed finite element model for analysis.

8. Interpret and analyze the results.

The operations done before the model is submitted for analysis are performed using a preprocessor. Analyzing the model using finite element procedures is done by the processor. The postprocessor manipulates the analysis results for easy understanding and interpretation.

The steps in a finite element analysis are illustrated in Figure 7.4.
Creating The Finite Element Model

Most of the time spent in developing a finite element model involves creating the model geometry and meshing. Initially, finite element models were generated by hand by the analyst. It involved laying out an element mesh on paper, numbering the nodal points and elements, reading off the nodal coordinates, and then setting up the required input cards for nodal coordinates, element connectivities, material properties, loads and boundary conditions. Since this used to cause errors and delays in performing the analysis, preprocessors for the generation of finite element model input were developed.

The generation of a finite element model can be considered as a four step process. The four steps are (Meyer, 1987):

1. Definition of the problem geometry.
2. Discretization of the problem geometry (i.e., dividing up the object into a set of finite elements).
3. Specification of material properties, loads, and boundary conditions.
4. Specification of the control parameters required by the analysis program and the linking of all the information to the selected analysis program.

Figure 7.4 Finite Element Analysis Steps (Kardestuncer, 1987).
The following sections present more information on each of the above mentioned steps.

**Coordinate System**

A good geometric modeler allows the user to utilize more than one coordinate system during the pre- and postprocessing of finite element models.

Coordinate systems commonly used in finite element analysis are:

- Cartesian coordinate system
- Polar coordinate system
- Cylindrical coordinate system
- Spherical coordinate system

Figures 7.5, 7.6 and 7.7 illustrate the different types of coordinate systems. The Cartesian system is the most commonly used. Often, coordinates in other systems are internally mapped into the Cartesian system during analysis. The coordinates of a point in the Cartesian system are established entirely in units of length for one-, two- and three-dimensional geometries. All other coordinate systems use a combination of length and angular units to represent the coordinates of a point. The polar coordinate system uses a length unit and an angular unit to describe the location of a point (1, 30°). The polar coordinate system is a two dimensional representation of the cylindrical coordinate system in which two length units and an angular unit are used to describe the location of a point in three dimensional space (1, 45°, 3). In the spherical coordinate system, two angular units and one length unit are used for coordinate description (2, 30°, 60°).
Geometry & Using Geometric Entities

Finite element modeling is applicable to almost all practical geometry models. The types of geometries encountered in finite element modeling and analysis can be broadly classified as:

- Discrete geometry
- Lattice geometry
- Continuous geometry

Figure 7.6 The Polar and Cylindrical Coordinate Systems.

Figure 7.7 The Spherical Coordinate System.
Figure 7.8 illustrates the different types of geometries. Discrete geometry includes points used for representing models such as concentrated masses and connecting springs. Lattice geometry refers to a gridwork of line elements. Examples include beams, frames, columns and trusses in two and three dimensional space. Continuous geometry is used for representing two and three dimensional continuous objects. These objects can be planar or solid. Planar objects can be flat (two dimensional) or curved (three dimensional). Solid continuous objects require three dimensional representation.

![Figure 7.8 The three types of geometry classifications.](image)

The geometry of a design model can be quickly constructed using geometric entities, the basic building elements in a geometric modeler. Entities such as point, curve, surface and volume can be conveniently used to quickly construct the model geometry. Most geometric modelers and CAD programs provide geometric entities for model building. In addition, FEA geometry modelers can also apply the loads and boundary conditions to geometric entities to speed up model preparation.

### Units

The finite element analysis algorithms and procedures are unit independent. You can use any consistent system of units for modeling your analysis problems. A consistent set of units for length, mass, force, and time must be used at all times to avoid errors. Appendix B describes sets of consistent units for commonly used engineering quantities.

### Meshing

A mesh is like a two or three dimensional grid system, composed of an arrangement of nodes and elements. The elements are connected to each other at the nodes. The process of creating nodes and elements is commonly referred to as meshing. Most preprocessors of finite element packages provide automatic mesh generation. Using this feature allows the user to instruct the preprocessor to automatically mesh the model geometry with the user's choice of elements and mesh gradation.
The various methods of mesh generation available can be classified into the following categories (Meyer, 1987):

- Unique coordinate transformations
- Blending functions
- Automatic triangulation

Smoothing algorithms are often employed to generate the final location of the nodal points in the interior of a mesh. Of the above three categories, automatic triangulation seems to be a popular choice of many preprocessors.

**General Considerations for Mesh Design**

An ideal mesh involves selecting the number and location of nodes and elements so that the analyses will be sufficiently accurate. The best finite element mesh is the one that uses the least resources (modeling and CPU time) for the design and analysis process. It is also characterized by "economy, simplicity, and discipline", where fewer nodes are used in noncritical areas and the number of nodes are increased in the critical areas undergoing a higher level of change (Kardestuncer, 1987).

Meshes can be classified into four types:

1. Optimum mesh
2. Regular mesh
3. Focused mesh
4. Undisciplined mesh

An *optimum mesh* has a large number of nodes and elements in the areas of the model being effected by the loading. Areas seeing little or no effect would have fewer nodes and elements, so that modeling and computation time is spent obtaining an adequate amount of results for critical areas.

A *regular mesh* has the same discretization error bound for every element of the grid. A simple regular grid has elements with the same type, shape and size.

*Focused meshes* are irregular meshes where the mesh is refined in selected areas. These meshes are developed by refining the parent mesh to make it denser in areas where the coarse mesh provides inadequate data.

*Undisciplined meshes* are irregular, have no parents, and are non-optimum.

The ability to select appropriate meshes is gained by experience. Meshing decisions include where to locate nodes and which type of element should be used. Typically, an optimum mesh would be the ideal choice to use in finite element analysis. However, it is not always possible to start with an optimum mesh. Usually the mesh evolves from a focused mesh to a regular mesh, and then towards an optimum mesh. The common process of evolution involves (Kardestuncer, 1987):

1. Performing a finite element analysis of the system using a relatively coarse mesh and the favored element type.
2. Evaluating the quality of the solution to determine if the mesh is acceptable.
3. Accepting the solution or repeating tasks 1 and 2 using a refined mesh if it is not acceptable.

Thus, mesh evolution requires estimating the accuracy and efficiency of a given mesh along with the criteria for redesign. The goal is to minimize the computer time required to identify an acceptable mesh and perform the finite element analysis. Thus, the process must define where, how, and by how much a given mesh should be changed. Adaptive meshing, to some extent, fulfills these goals. See the section, Adaptive Finite Element Analysis, for more information.

Elements

Elements are the fundamental building blocks of finite element analysis. The elements approximate the geometry of the structure. The mathematical models of the elements describe how the element behaves when individually subjected to loads.

Depending upon the geometry of the physical space, one or more of the following types of finite elements can be used for discretization (Meyer, 1987):

1. One-dimensional (1-D) line elements to model beams, columns, rods, bars, stiffeners, cables and ribs. In this case the one dimension, length, is large compared to the other two.
2. Two-dimensional (2-D) elements for modeling surface structures like plates and shells, with one dimension very small compared to the other two.
3. Axisymmetric solid or ring elements to discretize axisymmetric solid structures.
4. Axisymmetric shell or surface elements to discretize axisymmetric shell and pressure vessel type structures.
5. Three-dimensional (3-D) elements for modeling solids, with all three dimensions being of the same order of magnitude.

The following paragraphs describe and illustrate the different categories of elements with illustrations where appropriate.

One Dimensional Elements

One dimensional elements, also known as line elements, are used to represent lattice type structures such as building frames, beams, transmission towers, bridges, and pipe networks. The structures may occupy two or three dimensional space as shown in Figure 7.9.
Common types of line elements include truss, beam, bar, pipe, and spring elements. Many line elements have built-in features for different cross-sections such as I section, T section, Channel sections and many other commonly used shapes. The line elements may be linear with two nodes per element, quadratic (three nodes) and cubic (four nodes), as shown in Figure 7.10.

Line elements can be used in a model along with two (area) or three dimensional (volume) elements. To model either a plate strengthened by stiffeners or a composite girder-deck slab system as shown in Figure 7.11, you need to use a beam element along with plate or shell elements. Note that the beam element is attached to the plate/shell elements using a rigid link. The rigid link maintains the compatibility of displacements between the girder and the deck slab, i.e., between the beam and plate elements.
Two Dimensional Elements

For finite element modeling of planar (both flat and curved) continuum structures, as shown in Figure 7.12, you need to use two dimensional or area elements. Area elements have small thicknesses which do not require a general three dimensional treatment. Analysis using area elements is widely used in the design industry. Often, many three dimensional finite element analyses are preceded by equivalent two dimensional ones.
Common area elements include plate and shell elements with triangular and quadrilateral geometry as shown in Figure 7.13. These elements may be linear, quadratic, or cubic.

Figure 7.13 2-D and shell elements (Meyer, 1987).

Linear elements are suitable for modeling objects with straight edges. For modeling curved boundaries, higher order elements are better, as illustrated in Figure 7.14.

Figure 7.14 Example of use of higher and lower order elements on curved geometry boundaries (Meyer, 1987).
Three Dimensional Elements

For most practical structures, an analysis involves using elements which consider three dimensional effects. For example, to accurately model a concrete dam, you need to use solid elements as shown in Figure 7.15.

![Concrete Dam](image)

**Figure 7.15** Finite element model of a dam using 3-D solid elements (Meyer, 1987).

The 3-D elements are available in a wide variety of shapes. Common shapes include:

- Tetrahedra (with four triangular faces)
- Pentahedra (with two triangular faces and three quadrilateral faces)
- Hexahedra (with six quadrilateral faces)

Figure 7.16 illustrates the types of solid elements typically used in finite element analysis.
Performing finite element analysis with solid elements is generally computationally intensive and time consuming. However, the analysis can predict the behavior of structures more accurately than equivalent 2-D analyses.

**Axisymmetric Elements**

When modeling objects with symmetry about an axis, axisymmetric elements are commonly used. In modeling the wheel of a railcar, an axisymmetric element can be used as illustrated in the Figure 7.17.
Axisymmetric elements are essentially 2-D elements simulating 3-D behavior. For axisymmetric structures with negligible thickness, axisymmetric shell elements are used. For other axisymmetric structures, axisymmetric solid elements are used. The solid elements can be of triangular or quadrilateral cross-section. Figure 7.18 illustrates the types of axisymmetric elements.

Axisymmetric solid elements can be linear (as shown in Figure 7.18) or quadratic (as shown in Figure 7.19). Higher order (quadratic) axisymmetric elements are used to accurately represent the geometry and obtain better performance.
Material Properties

Since finite element analysis is an idealization and approximation of the actual problem, its success depends on how accurately you model the geometry and material properties. Elements are used to describe the geometry while mathematical models are used to describe the material behavior.

The mathematical models describing physical behavior such as deformation and conduction are called material models or constitutive equations. Hooke's law, which linearly relates stress with strain by means of a constant elastic modulus, is an example of the stress-strain equation, representing a linear model and its constitutive equation.

The types of materials used in different applications range from heterogeneous materials such as concrete to homogeneous materials such as steel. Material properties may be direction dependent. An isotropic material has identical properties in all directions. Orthotropic materials, on the other hand, have different properties in two different directions. Anisotropic material properties may vary in each coordinate direction.

A general purpose finite element program provides the following types of commonly used material models:

- Linear elastic isotropic models
- Linear elastic orthotropic models
- Linear elastic anisotropic models
- Nonlinear material models
- Isotropic heat conduction material properties
- Orthotropic heat conduction material properties
- Convection and radiation material properties
- Isotropic thermal expansion/contraction material properties
- Orthotropic thermal expansion/contraction material properties
- Various types of damping and mass material properties for dynamics analyses
The use of nonlinear material models requires a nonlinear solution algorithm. The linear elastic models are usually computed from the user specified data, such as elastic modulus and Poisson's ratio. Linear elastic material models can also include the effects of temperature for thermal stress analysis.

For many practical applications with homogeneous materials, isotropic linear models are sufficient. If you are performing stress analysis of a design component using finite elements, for example, a linear elastic isotropic model may be adequate. In this case, you need only to input the elastic constant and Poisson's ratio to describe the material.

**Imposing Boundary Conditions**

The finite element model must be sufficiently supported, or *constrained*, before an analysis can be performed. This is done by applying the boundary conditions existing on the structure. The type of constraint to be applied or enforced depends on the type of analysis. For stress, frequency, buckling, and thermal analyses, the boundary conditions can be classified into the following categories:

- Displacement (deformational) boundary conditions
- Thermal boundary conditions

In many FEA programs, it is possible to apply the boundary conditions in a user specified coordinate system.

The support conditions usually encountered in stress analysis problems are known as displacement boundary conditions. The term *essential* boundary conditions is also used to indicate these same type of boundary conditions.

Structures can be supported in many different ways. A hinged support allows rotational movements of the structure whereas a roller support allows for seasonal variations of temperature on structures, as shown in Figure 7.20 (Baran, 1988).
Boundary conditions for thermal (heat transfer) finite element analysis include thermal insulation, specified temperature and heat flux, and convective boundary conditions.

**Applying Loads**

The word *load* is a general term used to indicate applied force, displacement, pressure, moment, temperature and other types of applied loads. After meshing your design and imposing the boundary conditions, you need to apply loads on the finite element model to complete the input phase.

The types of loads that can be applied on a finite element model are:

- **Mechanical loads**
  - time independent (static/quasistatic) loads
  - time dependent (dynamic) loads

- **Thermal loads**
  - time independent (steady state) thermal loads
  - time dependent (transient) thermal loads

---

**Simply Supported** \(U_x = U_y = 0\)

**Fixed Support** \(U_x = U_y = \theta_z = 0\)

**Roller Support** \(U_y = \theta_z = 0\)

Figure 7.20 Examples of structural supports and the corresponding movements constrained by the supports (Baran, 1988).
Examples of mechanical loads include nodal forces and moments, imposed displacements, pressure loading, and gravity loading. Thermal loading includes prescribed nodal and element temperatures and heat fluxes.

**Performing An Analysis**

After all the input data has been prepared, the finite element model is submitted for analysis. Commercial FEA programs feature data checking to review the input data for errors before performing an analysis. The analysis is typically done by the processor of the finite element software. The processor, as shown in Figure 7.4, performs operations such as generating element shape functions and coordinate transformations, assembling element equations, applying boundary conditions, and performing solution procedures.

Typically, small jobs (about 1000 equations) are run interactively while larger ones may be submitted for execution in a batch mode. With the advancements made in high performance computing, some finite element calculations can be made in parallel, exploiting the parallel architecture of the computer. Some finite element calculations can also be vectorized, exploiting the array processing feature of a supercomputer which makes the computation faster and more efficient.

**Node Resequeencing**

Before performing an analysis, most finite element analysis programs resequence the node numbers generated by the preprocessor in a computationally efficient manner, as explained below.

The finite element system matrix $[A]$ is symmetric, sparse (has many zero entries) and banded (i.e., all non-zero entries cluster about the diagonal of the matrix) for most analyses. The bandwidth of this matrix depends on the way in which the nodes are numbered in a finite element mesh. Finite element computations can be made efficient by keeping the bandwidth as small as possible. Bandwidth minimization is achieved by renumbering the nodes in such a way that the difference in node numbers associated with any element is the smallest possible for that mesh. Figure 7.21 illustrates how the node numbers of a mesh can be resequenced to produce the minimum bandwidth.

**Figure 7.21** How the node numbers of a mesh can be resequenced to produce the minimum bandwidth.
As a user, you do not need to renumber the nodes yourself to produce minimum bandwidth. Most finite element analyzers, including COSMOS/M CAD Interface, feature automatic node resequencing (or renumbering) algorithms which minimize the bandwidth of the system matrix. The node renumbering is done internally during computations, and the results are output with respect to the originally generated node numbers.

**Postprocessing**

The purpose of postprocessing is to allow the user to review the results of the analysis both in tabular and graphic forms. This lets the user obtain a better understanding of the model's behavior. Information obtained from an analysis can be condensed and presented in a number of ways (Meyer, 1987):

1. Displaced (or deformed) shape plots
2. Contour plots (of stress, displacement, temperature, etc.)
3. Vector plots (of principal stresses, flow lines, etc.)
4. Animation
5. Thresholding of results (observing extreme values of the calculated results)
6. Automatic checks against design codes
7. Automatic generation of result reports

*Deformed shape* plots are very helpful in understanding how the structure is likely to behave in real life. The components of displacements are shown by an exaggerated deformed mesh. *Contour* plots can be used to show points of the same stress, temperature or displacement (within a certain tolerance) in the finite element mesh in order to observe the distribution of such quantities throughout the structure.

*Vector* plots are an alternative depiction of vector fields, such as displacements or velocities, with arrows of appropriate length and direction displayed at each key point. The understanding of the dynamic behavior of a structure can be greatly enhanced with animation. For example, the animation of the natural vibration modes of a structure provides better understanding of its vibrational behavior. Evaluation of extreme values by thresholding of results quickly draws attention to critical regions of a problem, while automatic design checks indicate the adequacy of the design. *Report generators* are used in conjunction with formatting procedures to reduce the effort of putting the results into a final form for presentation.

**Errors In Finite Element Analysis**

Except for rare occasions, finite element models yield only approximate solutions to many practical problems. Experienced FEA users know that nearly all solutions are at best only good approximations, and that it is important to estimate the errors in those approximations.

In general, the errors in finite element modeling and analysis may be classified as (Kardestuncer, 1987):
<table>
<thead>
<tr>
<th>Error</th>
<th>Source</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mathematical modeling errors</td>
<td>Theory</td>
</tr>
<tr>
<td>Mesh (Discretization) errors</td>
<td>User</td>
</tr>
<tr>
<td>Roundoff errors</td>
<td>Computer</td>
</tr>
<tr>
<td>Solution error</td>
<td>Total error minus mathematical modeling error</td>
</tr>
</tbody>
</table>

The analysis results are based on a mathematical representation of the actual physical problem. Many simplifying assumptions are made concerning the geometry of the solution domain, system parameters, and other relevant quantities. The difference between the measured quantities of the physical system and the corresponding solution from the mathematical abstraction is known as the mathematical modeling error.

Discretization (meshing) errors are caused by representing the infinitely many degrees of freedom of the mathematical model by the finite number of degrees of freedom in the FEA model.

Roundoff errors are caused by the limitation of digital computers to represent real numbers. Due to the limits on the computer hardware, only a part of the real number can be represented on the computers. This lack of complete representation of real numbers may lead to roundoff errors in finite element analysis which requires high numerical precision.

The solution error is defined as the total error less the mathematical modeling error. In fact, these errors are the sum of the meshing and roundoff errors.

The mathematical modeling and roundoff errors cannot be improved by the user. However, you have some control over the discretization and meshing errors. (See the following section, Adaptive Finite Element Analysis for more information on how a user can minimize meshing errors.) In general, you should be aware of the following facts about errors in finite element analysis (Kardestuncer, 1987):

1. The errors in the computed results originate from two sources: the roundoff errors in the numerical representation and the discretization errors caused by representing the infinite degree of freedom (DOF) system with a finite degree of freedom model. These errors may be represented as equation errors.

2. Depending upon system characteristics, the equation errors reflect on the solution by changing their magnitudes.

3. The algorithms used in obtaining the solution from the equations contribute further to the solution errors in two ways: by causing more roundoff errors and by altering the system characteristics.

4. Very little has been done in assessing the equation errors in the finite element method, although the mechanisms by which these reflect on the solution are very well understood.

5. In order to assess the errors by computer experiments, the same problem should be solved at least three times by gradually refining the finite element mesh. Then, the response-mesh-size plots should be produced for each mesh point of the crudest mesh, if possible.

6. In the response-mesh-size plots, an erratic appearance signals the presence of excessive roundoff errors.
Adaptive Finite Element Analysis

The adaptive finite element method can be linked to a predictor-corrector type procedure after an analysis is carried out. Mesh areas where errors in excess of the specified limit are identified are automatically improved to achieve convergence to an acceptable solution. In short, the analysis iteratively adapts itself to a particular problem and solves it within specified error bounds.

A number of methods have been proposed and tested for making improvements in the mesh. One method is to retain the original mesh and improve the meshing in the areas of selected elements. Others reconfigure the mesh while maintaining the same element topology or define an entirely new mesh. The following sections describe the two main approaches to adaptive finite element analysis. Please note that currently only the P-method is supported by some of the elements in COSMOS/M CAD Interface.

The H-Method

In the H-version of adaptive finite element analysis, the improvement within the selected area of a mesh is achieved by subdividing the elements into a number of smaller elements of the same type. This approach is referred to as an H-version enrichment scheme because the meshing is improved by reducing the element size (often measured in terms of a side length $h$).

Figure 7.22 shows a finite element model improved by the H-version of adaptive finite element analysis. The model shows a plate with a hole subjected to a uniform tension. High stresses can be expected near the hole due to stress concentration effects. As a result, more elements are required around the hole, and the adaptive H-refinement shown in Figure 7.22 verifies this observation.

Note that when a selected element is subdivided, the new element may have an element aspect ratio greater than that of the original element.

![Figure 7.22 Plate model showing adaptive H-Method mesh refinement.](image)
The aspect ratio is defined as the ratio of the length of the longest element side to the length of the shortest element side. The degree of sensitivity to aspect ratio is a function of the element type and the number of digits of accuracy available.

Note that with the H-refinement scheme, there is an additional computing cost associated with the application of boundary conditions, loads, and constraint equations on the refined mesh. Most finite element programs with adaptive refinement perform this task automatically and the user does not need to interfere.

**The P-Method**

The other approach to mesh improvement is P-refinement. In this approach, the element geometry is left unaltered, but the polynomial order of the element is increased.

An obvious advantage of the P-version approach is that it does not require the definition of additional elements. The convergence rate of the P-version for uniform refinements is generally superior to that of H-version refinements for some problems. However, for general problems requiring selective mesh improvements, there is no proof that either one of these two methods is superior.

**The H-P Method**

The H-P Method is a combination of the H- and P-Methods.

**Suggested Further Reading**

To learn more about the finite element method, the following books and journals are suggested:


3. *Manager's Guide To FEA*, from CAE Magazine at Penton Publishing, 100 Superior Ave, Cleveland, Ohio 44114 or (216) 696-7000. This provides a simple introduction to finite element analysis.


7. *Finite Element Idealization, edited by Christian Meyer for the American Society Of Civil Engineers (ASCE), 1987.* This is a thorough static and dynamic handbook for engineers who regularly use FEA.

8. *An Introduction to the Finite Element Method using Basic Programs, by David K. Brown, Surrey University Press (Blackie).* This book gives an overview for FEA methods for the new user.


13. *Finite Elements in Analysis and Design (Periodical).*
Appendix A

Glossary & Acronyms
## Common Acronyms

<table>
<thead>
<tr>
<th>Acronym</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>AISC</td>
<td>American Institute of Steel Construction</td>
</tr>
<tr>
<td>ANSI</td>
<td>American National Standards Institute</td>
</tr>
<tr>
<td>ASCII</td>
<td>American Standard Code for Information Interchange</td>
</tr>
<tr>
<td>CAD</td>
<td>Computer Aided Design</td>
</tr>
<tr>
<td>CAE</td>
<td>Computer Aided Engineering</td>
</tr>
<tr>
<td>CAM</td>
<td>Computer Aided Manufacturing</td>
</tr>
<tr>
<td>DOF</td>
<td>Degrees of freedom</td>
</tr>
<tr>
<td>E</td>
<td>Young's modulus</td>
</tr>
<tr>
<td>FEA</td>
<td>Finite Element Analysis</td>
</tr>
<tr>
<td>FEM</td>
<td>Finite Element Modeling</td>
</tr>
<tr>
<td>G</td>
<td>shear modulus</td>
</tr>
<tr>
<td>GKS</td>
<td>Graphical Kernel System</td>
</tr>
<tr>
<td>IGES</td>
<td>Initial Graphics Exchange Specification</td>
</tr>
<tr>
<td>k</td>
<td>stiffness or spring constant</td>
</tr>
<tr>
<td>K</td>
<td>stiffness matrix, square, positive definite</td>
</tr>
<tr>
<td>MCAE</td>
<td>Mechanical Computer Aided Engineering</td>
</tr>
<tr>
<td>MCAD</td>
<td>Mechanical Computer Aided Design</td>
</tr>
<tr>
<td>NC</td>
<td>Numerical Control</td>
</tr>
<tr>
<td>RAM</td>
<td>Random Access Memory</td>
</tr>
<tr>
<td>ROM</td>
<td>Read Only Memory</td>
</tr>
<tr>
<td>u,v,w</td>
<td>deflections</td>
</tr>
</tbody>
</table>
Glossary Of Terminology

A

Absolute coordinates: the location of a point or node in reference to a fixed coordinate system in space.

Adaptive finite element method: a solver that iteratively performs finite element analysis, determines the areas of the mesh where the solution is not sufficiently accurate, and improves the mesh in those areas until the prespecified degree of accuracy is obtained. See also Adaptive meshing.

Adaptive meshing: a method of mesh refinement (usually automatic) so that the resulting finite element solutions meet specified error tolerances. See also Adaptive finite element method.

Aspect ratio: the ratio between the longest and shortest finite element dimensions. The element aspect ratio is a useful measure of the quality of the shape of an element.

Association: the relationship of two entities so that changes in one are reflected in the other.

Automatic meshing: creation of elements and nodes automatically by the software. See also Finite element analysis and Finite element modeling.

Automatic node renumbering: a node renumbering scheme to minimize the bandwidth and optimize the profile of the system stiffness matrix (this process is completely transparent to you).

Axial symmetry: a solid or shell of revolution is axially symmetric if its geometry, material properties, and loading conditions are all constant along the circumferential direction.

Axisymmetric 2D element: a two dimensional element that can be used to simulate body of revolution problems.

B

Banded matrix: a matrix in which the nonzero elements cluster around the diagonal and elements away from the diagonal are zeros.

Beam element: a one dimensional finite element defined by three nodes or two nodes and an angle. A beam element has six degrees of freedom at each of the two end nodes. The third node or the angle is used to define the orientation of the cross section in space. Beam elements are capable of resisting axial, bending, shear, and torsional forces.

Body force: force due to the mass of the body or object being modeled. Body forces include gravitational and centrifugal forces.

Boundary conditions: conditions imposed on some areas of the design. For example displacement boundary conditions are applied to structural problems to restrain its motion and stabilize it.
**Buckling**: sudden deformation of slender members or thin walled structures under compressive loading due to sudden release of internal energy in the form of bending deformation.

**Bulk Modulus**: a measure of the volumetric deformability of a material. It is the ratio of the volumetric stress to volumetric strain.

**C**

**Cartesian coordinate system**: a coordinate system using three orthogonal axes (X,Y,Z) to specify locations of points in a three dimensional space.

**Coefficient of thermal expansion**: the change in length per unit length per unit rise in temperature.

**Collapse of structure**: a state of mechanism in which the structure cannot support any load. Most programs warn of a singular structural stiffness matrix when the collapse load is reached or the system is in a mechanism state.

**Composite material**: a material that is obtained by combining two or more materials on a macroscopic scale. Composite materials are principally manmade and, therefore, the constituents can be selected and combined to produce a useful material that has the desired properties, such as high strength, high stiffness, low weight, and so on.

**Computational Fluid Dynamics (CFD)**: a computer based numerical study of fluid flow using approximate methods such as the finite element method, the finite difference method, the boundary element method, the finite volume method, and so on.

**Computer aided design (CAD)**: the use of computer software to aid (e.g. solid modeling) in the creation or modification of a design.

**Computer aided engineering (CAE)**: the use of computer software (e.g. finite element analysis) in the analysis and design of engineering parts and components.

**Computer aided manufacturing (CAM)**: the use of a computer software (e.g. numerical control) to aid in the manufacturing of engineering parts.

**Conduction (Heat)**: the process of heat transfer within solid media due to a temperature gradient.

**Consistent mass matrix**: a representation of the element mass in which the mass of the element is assumed to be continuously distributed throughout the element (see *Lumped mass*). The computation of the consistent mass matrix uses the same shape functions used in the computation of the stiffness matrix.

**Constraints**: the restrictions imposed on the degrees of freedom of some nodes in a finite element model.

**Contour**: a series of curves forming closed loops in a plane or 3-D surface. Stress contours for example represent lines connecting stresses and displacements of the same magnitude.
**Contour plot:** a graphical display that shows equal values of the plotted quantity in the same color. Line contour plots connect locations of selected equal values of the plotted quantity (isolines).

**Convection:** heat transfer due to the motion of fluids. Fluid particles act as carrier.

**Critical buckling loads:** minimum loads that cause buckling (see *Buckling*).

**Cyclic symmetry:** a repetition of geometry and loading even though no axis of reflective symmetry exists, as seen in some structures, such as impellers in centrifugal pumps. Since stresses and displacements also display cyclic symmetry, you only need to analyze one repeating structure.

**Cylindrical coordinate system:** a coordinate system in which the coordinates are specified by a radial distance (r), a circumferential angle (θ), and a distance along the cylindrical axis (z).

**D**

**Dead load:** the self weight of the structure. See also *Body force*.

**Default:** a value assumed by the program for an input in the absence of specific user instructions.

**Deflection:** the distance a structure deforms in a given direction.

**Degrees of freedom (DOF):** the number of independent displacement measurements that serve to describe all possible displacement configurations of a structure.

**Density:** the mass per unit volume.

**Discretization:** the process of dividing the model into finite elements. See also *Mesh*.

**Displacement:** translation and/or rotations.

**Dynamic analysis:** a method of analysis in which, due to the time dependency of applied forces, the inertia effects of accelerating masses must be considered, leading to the solution of equations of motion.

**Dynamic response:** the time dependent response of a dynamic system in terms of its displacement, velocity, or acceleration at any given point of the system.

**E**

**Eigenvalue:** a root of the characteristic equation of a matrix. For frequency analysis, the eigenvalues are the squares of the resonant frequencies. For buckling analysis, the eigenvalues represent the critical buckling load factors.

**Eigenvector:** a mode shape that corresponds to a resonant frequency or buckling mode. The terms eigenvector and mode shape are equivalent.
Elastic behavior: the deformational behavior of a structure in which stresses vary linearly with strain, i.e. displacements, strains, and stresses will double if loads are doubled.

Elastic modulus: the ratio of stress to strain, or the stress required to cause a unit strain. The elastic modulus is also called Young’s modulus and is the slope of the straight line portion of the stress-strain curve.

Element: the basic building block of the finite element analysis model.

Emissivity: the ratio of the heat flux emitted by a body to the heat flux emitted by a black body at the same temperature.

F

Face (of an element): a side of a finite element generally used to apply loads and/or boundary conditions.

Failure criteria: a mathematical model formulated from the combination of stress components to establish a failure criterion. This criterion is generally used in connection with the behavior of a material to define the condition of failure for that material.

Finite element analysis (FEA): a computer based analysis method which calculates the response of the system by solving the set of simultaneous equations that represent the behavior of the structure under the applied loads. See also Finite element modeling and Nonlinear finite element analysis.

Finite element modeling (FEM): the process of setting up a model for analysis with the finite element method, typically involving graphical generation of the model geometry, meshing it into finite elements, defining material properties, and applying loads and boundary conditions.

Fixed boundary conditions: all degrees of freedom (translations and rotations) are set to zero.

Flexibility matrix: a matrix whose coefficients represent displacements due to unit forces applied to the structure at its coordinates.

Flexural rigidity: represents the stiffness of the structure in bending.

Fundamental frequency: the lowest resonant (natural) frequency. The corresponding mode shape is called the fundamental mode of vibration. See also Natural frequency.

G

Gaussian elimination: a method of solving a linear system of equations of the form AX=F.

Gauss quadrature: a numerical method of integration used to evaluate the stiffness matrices of finite elements.
**Gauss points**: points selected in the element domain to minimize errors from numerical integration.

**Geometric nonlinearity**: the consideration of large displacement effects in the analysis of structural systems. The strain-displacement relations will include second or higher order terms. Geometric nonlinearity is particularly important for slender or thin walled structural members under compressive in-plane (membrane type) loading.

**Geometric properties**: various shape dependent properties such as thickness, cross-sectional area, sectional moment of inertia, and location of center of gravity.

**Global coordinate system**: the default Cartesian coordinate relative to which the complete finite element system.

**H**

**Hardcopy**: images or text printed or plotted onto a media such as paper or film.

**H-convergence**: convergence towards a more accurate solution by subdividing the elements into a number of smaller elements. This approach is referred to as h-convergence because of improved discretization due to reduced element size (often measured in terms of a side length h).

**H-P convergence**: the best possible convergence that can be obtained by combining the H- and P-Methods into a procedure that, in a given iteration, will use the method that gives the best result in that step. See also *H-convergence* and *P-convergence*.

**Higher order elements**: finite elements with higher order terms in their shape functions or elements that include additional nodes (such as mid-side nodes) resulting in greater overall DOF describing the element. These elements are more accurate than their lower order counterparts.

**Hinged connections**: connections which permit rotations but not translations.

**H-Method**: a finite element method which uses h-convergence for improving solutions. See also *H-convergence*.

**Hoop stresses**: circumferential stresses in a cylindrical pipe or spherical dome due to internal or external pressures.

**Hourglass mode**: a zero-energy deformation mode (or kinematic mode or mechanism) which produces a strain field that is zero at all quadrature (e.g., Gauss) points in elements. Hourglass modes are characterized by introducing singularity in the stiffness matrix so that solutions for static cases may not be possible.

**I**

**Icon**: a pictorial display of a frequently used function. It provides for simple and quick execution of that function by highlighting the icon with the mouse cursor.
Incompressible materials: materials which exhibit no volumetric change when subjected to compressive loads.

In-core solution: solution of a system of equations using the main memory available in the computer.

Inelastic analysis: stress analysis in which the effects of material nonlinearity (such as plasticity) are included.

Isoparametric elements: elements in which the same (iso) parameters are employed for both displacement and geometric approximation. Isoparametric elements are higher order elements that provide better accuracy in the solution of models involving complicated geometries.

Isotropic material: a material which has the same properties in all directions.

J

Jacobian: a matrix relating the local coordinates of an element to the global coordinate system.

L

Lanczos method: a numerical method for extracting eigenvalues and eigenvectors.

Limit analysis: an analysis which gives lower and upper bounds to the collapse load of a structure. The information obtained is confined to the collapse load and the collapse mechanism.

Line loads: distributed loads along a line on the structure.

Linear analysis: analysis in which the response of the design is a linear function of the applied loads.

Load cases: a feature that allows you to apply different sets of loading environments on the same structure in one analysis. Each load application is known as a load case.

Local coordinate system: a coordinate system defined relative to the global coordinate system for the convenient application of loads or boundary conditions.

Lumped mass: a method of representing the mass in a dynamic finite element analysis where the element mass is assumed to be concentrated at the nodes.

M

Mass element: an element lumped at a node representing the effect of a concentrated mass in different coordinate directions.

Material nonlinearity: the consideration of the material behaving nonlinearly such as elastic plastic materials, in the analysis of structural systems. The stress-strain relations are no longer
accurate by only considering its linear part. Material nonlinearity is important particularly when analyzing structural systems built from nonmetallic materials such as rubbers and plastics.

**Matrix**: an array of numbers generally arranged in the form of a rectangular table.

**Mechanical computer aided engineering (MCAE)**: the use of computers in mechanical design, especially the tight coupling of geometric modeling and analysis.

**Mechanism**: a structure which, for certain patterns of deformation, undergoes no stress inducement. Hence, it cannot support loads and is considered an unstable structure. See also **Zero-energy mode**.

**Meshing**: creating a mesh or grid of elements on a finite element model represented by its surface or solid geometry. See also **Finite element analysis**.

**Mesh convergence**: See **H-convergence**.

**Modal analysis**: same as frequency analysis: the calculation of resonant frequencies and corresponding mode shapes. The results of modal analysis can be used to predict the response of the design to dynamic loads.

**Mode shape**: a shape that describes the deformed configuration of the structure for a particular natural frequency. See also **Eigenvector**.

**Modulus of elasticity**: stress required to produce a unit strain.

**N**

**Natural frequency**: the frequency of vibration of a structure under undamped harmonic conditions.

**Node**: a point generated from the intersection of grid lines used for subdividing a continuous system into a series of finite elements.

**Node renumbering**: the process of reassigning the node numbers in a finite element mesh for the purpose of improving computational efficiency.

**Nonlinear finite element analysis**: the analysis of structures involving geometric and material nonlinearity using finite element analysis.

**Normal modes analysis**: analysis involving the calculation of natural frequencies and corresponding mode shapes of a structure. See also **Modal analysis**.

**Numerical integration**: an approximate method of calculating the value of an integral.

**O**

**Orthotropic material**: a material with different properties in two or three mutually perpendicular directions.
Out-of-core solution: the solution of equations with arrays larger than the core memory, thus dividing the system of equations into a number of blocks, writing them on the hard disk, and swapping blocks in and out of core memory.

P

P-convergence: a method of solution convergence in which the finite element mesh is unaltered but the order of the polynomial representing the displacement function of each element is increased.

Patch: a bounded section of a surface.

Patch test: a test in which a patch of elements is subjected to a set of nodal point displacements that in an exact analysis correspond to the constant strain condition. If the element strains calculated by FEA do actually represent the constant strain condition, the patch test is passed, i.e., the completeness condition is satisfied by the element assemblage.

Plane strain: a condition where the strains are zero in one of the three orthogonal axes. This condition means that there is no out-of-plane deformation of the cross-section. In general, there are nonzero stresses in the direction of zero strain. The assumption of plane strain is valid for long cylindrical or prismatic bodies uniformly loaded perpendicular to the axis of zero strain.

Plane stress: a condition where stresses in one of the three orthogonal axes are zero. The strains, however, are not zero due to the Poisson effect. A thin plate loaded by membrane type forces (forces in the plane of the plate) is an example of a plane stress condition.

Plate (bending) element: a flat planar element with resistance to bending and membrane forces.

P-Method: a method of finite element analysis which uses p-convergence to iteratively minimize the error of analysis. See also P-convergence.

Poisson's ratio: the ratio of the lateral or perpendicular strain to the longitudinal or axial strain.

Polar coordinate system: a planar coordinate system in which the coordinates are defined by a radial distance (r) and a circumferential angle (θ).

Postprocessing: evaluating the results of a finite element analysis through tabular and/or graphical displays. Postprocessing contains all steps after performing the analysis.

Preprocessing: preparing finite element input data involving model creation, mesh generation, material definition, and load and boundary condition application. Preprocessing contains all steps before performing the analysis.

Principal stress: normal stresses which act on planes free of shear stresses.
R

Radiation (thermal): transfer of heat through electromagnetic waves.

Reactions: forces produced at supports of a structure under external loading so as to bring it into a state of equilibrium.

Relative coordinates: coordinates specified by differences in distances and/or angles measured with respect to the previous configuration rather than the original configuration.

Renumbering: See Node renumbering.

Restart capability: the ability of a finite element program to restart the solution of the problem after partial execution of a transient or nonlinear analysis.

Rigid-body modes: modes of vibration in which the entire model moves without deformation. The frequency of a rigid body mode is zero (period is infinite: no oscillatory motion).

Rigid-body motion: free movement of the object in space without any relative motion between any two particles in the model.

Rigid connection: a connection with relatively high stiffness, allowing very little or no deformations under loading. See also Rigid elements/links.

Rigid elements/links: elements that have relatively high stiffnesses so that little or no deformations are permitted. See also Rigid connection.

Rotational degrees of freedom: degrees of freedom associated with angular deformation.

S

Shell element: elements used for modeling thin walled surface structures such as pressure vessels and domes. Shell elements provide load bearing capacity through both membrane and bending actions.

Shear modulus: the ratio of shear stress to the corresponding shear strain.

Small deformation theory: in a linear structural analysis, the stress calculations and the equilibrium conditions are based on the original undeformed shape of the structure, i.e., the deformations are so small compared to the size of the structure so that the initial configuration can be used to calculate the stiffness.

Solid elements: finite elements for modeling three dimensional solid designs.

Solid modeling: a modeling technique in which three dimensional solid structures are constructed by the union, intersection, and/or other combinations of certain solid primitives, such as cylinders, cubes, and cones. Information regarding all facets of the solid structure is accessible from the associated database files.

Solids of revolution: solids generated by sweeping a two dimensional model about the axis of revolution.
**Solvers:** computerized solution algorithm for solving a system of simultaneous equations.

**Specific heat:** the heat required to raise the temperature of a unit mass by a unit temperature.

**Spherical coordinate system:** a coordinate system in which the coordinates are specified by a radial distance (r), a circumferential angle (q), and a longitudinal angle (f).

**Spring elements:** elements for modeling rigid supports or spring connections. The spring element is an axial spring with two nodes and a user defined axial and rotational stiffness.

**Static analysis:** analysis of designs to solve for displacements, strains, and stresses while eliminating the effect of inertia. Loads are statically applied and do not change with time.

**Steady State heat transfer:** determination of the temperature distribution of a design which has reached thermal equilibrium under the action of specified thermal loads and boundary conditions.

**Stefan-Boltzmann constant:** a constant in the analysis of radiation heat transfer representing the constant of proportionality between the heat flux emitted by a black body and the fourth power of the absolute temperature of the black body.

**Stiffness:** force (or moment) required to produce unit translation (or rotation).

**Stiffness matrix:** an array of stiffness terms relating the element force components to the corresponding element displacement components.

**Strain:** a dimensionless quantity calculated as the ratio of change in length to the original length.

**Strain energy:** again considering a bar of uniform cross-section held fixed at one end and subjected to a slowly applied axial force at the other end, the area under the load-deformation diagram for this bar represents the work done in elongating the bar or the strain energy stored in the bar.

**Stress:** intensity of internal forces in a body (force per unit area).

**Stress analysis:** the computation of stresses due to applied loads.

**Stress concentration:** magnification of stresses near concentrated loading, sharp corners, narrow sections, holes, or other discontinuities.

**Stress contour plot:** a plot of a stress component by a series of color filled contours representing regions of equal stress.

**Stress-strain relations:** a mathematical relation between stresses and strains.

**Structural stability:** study of stable, unstable, and neutral equilibrium states of structures. See also *Buckling*.

**Sturm sequence check:** a property in eigenvalue extraction which computes the number of eigenvalues in an arbitrary range. By itself, the Sturm sequence method is not efficient for eigenvalue extraction, particularly for large bandwidths.
Subspace iteration: a method of eigenvalue extraction where the iterations are performed on a q-dimensional subspace. The number of starting q iteration vectors are always larger than the number of p eigenvalues and eigenvectors that are to be calculated (q > p).

Substructuring: an efficient way of solving large finite element analysis problems by breaking the model into several parts or substructures, analyzing each one individually, and then combining them for the final results. It eliminates computational problems such as memory, speed, and size limitations.

Surface modeling: the geometric modeling technique in which the model is created in terms of its surfaces only, with no volume definition.

Symmetric matrix: a matrix in which elements in the ith column and jth row are equal to elements in the jth column and ith row.

T

Temperature contour plots: a plot showing contour lines connecting points of equal temperature.

Thermal conductivity: amount of heat conducted per unit area, per unit rise in temperature over a unit length.

Thermal stress analysis: the computation of stresses and displacements due to changes in thermal conditions.

Transient heat transfer: heat transfer problems in which temperature distribution varies as a function of time.

Transient response: refers to the time varying response of a system subjected to time varying loading.

Translational degrees of freedom: degrees of freedom representing translations along specified coordinate axes.

Truss element: a one dimensional line element defined by two nodes resisting axial loads only.

V

Vector: a vector quantity is defined by its magnitude and direction (force) as opposed to a scalar quantity defined by magnitude (temperature).

View point: a point located in 3D space from which a model is viewed.

von Mises stress: an effective stress quantity defined as the second invariant of the stress tensor. In the analysis of 2D and 3D structures, von Mises stress is used as an effective stress quantity relative to which the state of stress of the structure is measured.
W

Warping: the out-of-plane deformation of the cross-section of bars due to shear stresses.

Window: a computer screen area that functions independently, almost as a separate screen.

Wireframe: a geometric representation of 3D models as outlined by outer edges.

Y

Young's modulus: the stress required to produce a unit strain. See also Modulus of elasticity and Elastic modulus.

Z

Zero-energy mode: a mode of deformation which produces no strain energy, i.e., a mechanism. See also Hourglass mode.
Introduction

In COSMOS/M CAD Interface and COSMOS/M products in general, the user is free to adopt any system of units for the input. The user can use standard or non-standard systems, but should be consistent and interpret the output accordingly. In this appendix, a brief guide for the commonly used units in FEA is given. It should be mentioned that if material (or section) properties are specified by selecting them from a library, all other input must be consistent with the units used in that library.

In solving the basic equations in structural mechanics, the units of force, mass, length and time can not be all chosen arbitrarily. Three of the four can be optionally selected, but the fourth one is then derived. Two systems of consistent units commonly used are the International System of units (SI units) and the US customary units. Tables illustrating the units of various input and output quantities used in COSMOS/M CAD Interface are given in this appendix for the SI and US customary systems of units.

International System Of Units

The basic units in this system are the units of length, mass and time and are, respectively, the meter (m), the kilogram (kg), and the second (s). Other units are derived from these basic units. The unit of acceleration, for example, is the unit of length divided by the squared unit of time (m/s/s); the unit of force, called a Newton (N), is defined as the force required to give a unit of acceleration to a unit of mass (kg.m/s/s). The weight of a body, like any other force, must be expressed in Newtons which is the weight of a one kilogram mass multiplied by the acceleration of gravity or 1.0 x 9.81 kg.m/s².

The SI system is an absolute system of units since measurements are independent of the location. The units in this system can be used anywhere on Earth, as well as on other planets, and will always have the same significance.

Principal SI Units used in structural mechanics are illustrated in Table B.1.

U.S. Customary System Of Units

The basic units in this system are the foot (ft) for length, the pound (lb) for weight, and the second (s) for time. The weight (not mass) is a basic system, contrary to the SI system. The system is gravitational (and not absolute) since the weight of a body depends on the value of the acceleration of gravity which varies slightly on Earth. The weight of a body on Earth is about six times its weight on the moon. The pound is defined as the weight of a platinum standard, called the standard pound. The mass of the standard pound is 0.45359243 kg. Since the weight depends on the gravity field, it is stated that the pound is the weight of that mass at sea level and latitude of 45 degrees.

While the standard pound is used as a unit of mass in many commercial transactions, it should not be so used in engineering practice. The mass in the US customary system is the slug. The slug is defined as the mass which attains a unit acceleration (ft/s²) when subjected to a unit force (lb).
Principal US customary units used in mechanics and their equivalent in the SI system are shown in Table B.2.

<table>
<thead>
<tr>
<th>Quantity</th>
<th>Unit</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>Length</td>
<td>Meter</td>
<td>m</td>
</tr>
<tr>
<td>Mass</td>
<td>Kilogram</td>
<td>kg</td>
</tr>
<tr>
<td>Time</td>
<td>Second</td>
<td>s</td>
</tr>
<tr>
<td>Area</td>
<td>Square meter</td>
<td>m²</td>
</tr>
<tr>
<td>Solids volume</td>
<td>Cubic meter</td>
<td>m³</td>
</tr>
<tr>
<td>Liquid volume</td>
<td>Liter</td>
<td>L=10⁻³ m³</td>
</tr>
<tr>
<td>Velocity</td>
<td>Meter per second</td>
<td>m/s</td>
</tr>
<tr>
<td>Acceleration</td>
<td>Meter per second squared</td>
<td>(m/s²)</td>
</tr>
<tr>
<td>Angle</td>
<td>Radian</td>
<td>rad</td>
</tr>
<tr>
<td>Angular velocity</td>
<td>Radian per second</td>
<td>rad/s</td>
</tr>
<tr>
<td>Angular acceleration</td>
<td>Radians per second squared</td>
<td>rad/s²</td>
</tr>
<tr>
<td>Density</td>
<td>Kilogram per cubic meter</td>
<td>kg/m³</td>
</tr>
<tr>
<td>Force</td>
<td>Newton</td>
<td>N=kg m/s²</td>
</tr>
<tr>
<td>Moment of a force</td>
<td>Newton-meter</td>
<td>N m</td>
</tr>
<tr>
<td>Stress and pressure</td>
<td>Pascal</td>
<td>Pa=N/m²</td>
</tr>
<tr>
<td>Frequency</td>
<td>Hertz</td>
<td>Hz=cycle/s</td>
</tr>
<tr>
<td>Impulse</td>
<td>Newton-second</td>
<td>N s</td>
</tr>
<tr>
<td>Work</td>
<td>Joule</td>
<td>J=N m</td>
</tr>
<tr>
<td>Power</td>
<td>Watt</td>
<td>W=J/s</td>
</tr>
<tr>
<td>Thermal conductivity (K)</td>
<td>Watt per meter per degree Centigrade</td>
<td>W/m °C</td>
</tr>
<tr>
<td>Specific heat (C)</td>
<td>Joule per kilogram per degree Centigrade</td>
<td>J/kg °C</td>
</tr>
<tr>
<td>Convection film coefficient</td>
<td>Watt per meter squared per degree Centigrade</td>
<td>W/m² °C</td>
</tr>
<tr>
<td>Heat generation per node (Q)</td>
<td>Watt</td>
<td>W</td>
</tr>
<tr>
<td>Heat flux (heat generation per area)</td>
<td>Watt per meter squared</td>
<td>W/m²</td>
</tr>
<tr>
<td>Heat generation per unit volume (QE)</td>
<td>Watt per cubic meter</td>
<td>W/m³</td>
</tr>
</tbody>
</table>
### Table B.2 Conversion Table for U.S. Customary and SI Units

<table>
<thead>
<tr>
<th>Quantity</th>
<th>US Customary Unit</th>
<th>SI Equivalent</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Length</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>ft</td>
<td>0.3048 m</td>
</tr>
<tr>
<td></td>
<td>in</td>
<td>0.0254 m</td>
</tr>
<tr>
<td></td>
<td>mi</td>
<td>1.609 km</td>
</tr>
<tr>
<td><strong>Area</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Area</td>
<td>ft²</td>
<td>0.0929 m²</td>
</tr>
<tr>
<td></td>
<td>in²</td>
<td>645.2 mm²</td>
</tr>
<tr>
<td><strong>Volume</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Volume</td>
<td>ft³</td>
<td>0.02832 m³</td>
</tr>
<tr>
<td></td>
<td>in³</td>
<td>16.39 cm³</td>
</tr>
<tr>
<td><strong>Liquid Volumes</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Liquid Volumes</td>
<td>gal</td>
<td>3.785 L</td>
</tr>
<tr>
<td></td>
<td>qt</td>
<td>0.9464 L</td>
</tr>
<tr>
<td><strong>Velocity</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Velocity</td>
<td>ft/s</td>
<td>0.3048 m/s</td>
</tr>
<tr>
<td></td>
<td>in/s</td>
<td>0.0254 m/s</td>
</tr>
<tr>
<td></td>
<td>mi/h (mph)</td>
<td>0.4470 m/s</td>
</tr>
<tr>
<td></td>
<td>mi/h</td>
<td>1.609 km/h</td>
</tr>
<tr>
<td><strong>Acceleration</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Acceleration</td>
<td>ft/s²</td>
<td>0.3048 m/s²</td>
</tr>
<tr>
<td></td>
<td>in/s²</td>
<td>0.0254 m/s²</td>
</tr>
<tr>
<td><strong>Mass</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Mass</td>
<td>oz mass</td>
<td>28.35 g</td>
</tr>
<tr>
<td></td>
<td>lb mass</td>
<td>0.4536 kg</td>
</tr>
<tr>
<td></td>
<td>slug</td>
<td>14.59 kg</td>
</tr>
<tr>
<td></td>
<td>ton</td>
<td>907.2 kg</td>
</tr>
<tr>
<td><strong>Force</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Force</td>
<td>kip</td>
<td>4.448 kN</td>
</tr>
<tr>
<td></td>
<td>lb weight</td>
<td>4.448 N</td>
</tr>
<tr>
<td></td>
<td>oz weight</td>
<td>0.2780 N</td>
</tr>
<tr>
<td><strong>Moment of Force</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Moment of Force</td>
<td>lb ft</td>
<td>1.356 N m</td>
</tr>
<tr>
<td></td>
<td>lb in</td>
<td>0.1130 N m</td>
</tr>
<tr>
<td><strong>Area moment of Inertia</strong></td>
<td></td>
<td>0.4162 x 10⁶ mm⁴</td>
</tr>
<tr>
<td>Mass moment of Inertia</td>
<td></td>
<td>1.356 kg m²</td>
</tr>
<tr>
<td><strong>Pressure or Stress</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Pressure or Stress</td>
<td>lb/ft²</td>
<td>47.88 Pa</td>
</tr>
<tr>
<td></td>
<td>lb/in² (psi)</td>
<td>6.895 kPa</td>
</tr>
<tr>
<td><strong>Work or Energy</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Work or Energy</td>
<td>ft lb</td>
<td>1.356 J</td>
</tr>
<tr>
<td><strong>Impulse or Momentum</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Impulse or Momentum</td>
<td>lb s</td>
<td>4.448 N s</td>
</tr>
<tr>
<td><strong>Power</strong></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Power</td>
<td>ft lb/s</td>
<td>1.356 W</td>
</tr>
</tbody>
</table>
Engineering Prefixes

Multiple and submultiple units commonly used in engineering practice are shown in Table B.3.

### Table B.3 Common Engineering Prefixes and Symbols

<table>
<thead>
<tr>
<th>Multiplication</th>
<th>Prefix</th>
<th>Symbol</th>
</tr>
</thead>
<tbody>
<tr>
<td>$10^{12}$</td>
<td>tera</td>
<td>T</td>
</tr>
<tr>
<td>$10^9$</td>
<td>giga</td>
<td>G</td>
</tr>
<tr>
<td>$10^6$</td>
<td>mega</td>
<td>M</td>
</tr>
<tr>
<td>$10^3$</td>
<td>kilo</td>
<td>k</td>
</tr>
<tr>
<td>$10^2$</td>
<td>hekto</td>
<td>h</td>
</tr>
<tr>
<td>$10^1$</td>
<td>deka</td>
<td>da</td>
</tr>
<tr>
<td>$10^{-1}$</td>
<td>deci</td>
<td>d</td>
</tr>
<tr>
<td>$10^{-2}$</td>
<td>centi</td>
<td>c</td>
</tr>
<tr>
<td>$10^{-3}$</td>
<td>milli</td>
<td>m</td>
</tr>
<tr>
<td>$10^{-6}$</td>
<td>micro</td>
<td>μ</td>
</tr>
<tr>
<td>$10^{-9}$</td>
<td>nano</td>
<td>n</td>
</tr>
<tr>
<td>$10^{-12}$</td>
<td>pico</td>
<td>p</td>
</tr>
<tr>
<td>$10^{-15}$</td>
<td>femto</td>
<td>f</td>
</tr>
<tr>
<td>$10^{-18}$</td>
<td>atto</td>
<td>a</td>
</tr>
</tbody>
</table>

Tables of Consistent Units for COSMOS/M Modules

As mentioned earlier, standard as well as nonstandard systems of units can be used in COSMOS/M products. The following tables list proposed consistent units for input and output quantities used in COSMOS/M CAD Interface.

The following applies to Tables B.4 through B.8.

- Units are consistent with the COSMOS/M material library database.
- FPS refers to the U.S. customary system of units.
- SI refers to the International system of units.
- MKS refers to the Metric system of units.
- CGS refers to the French system of units.
Table B.4  Table of Consistent Units for Linear Static Analysis

<table>
<thead>
<tr>
<th>Measure</th>
<th>COSMOS Name</th>
<th>* FPS (gravitational)</th>
<th>* SI (absolute)</th>
<th>* MKS (gravitational)</th>
<th>CGS (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Material Properties</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>EX, EY, EZ</td>
<td>lbs/in²</td>
<td>Newton/m² or Pascal</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>GXY, GYZ, GXZ</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>NUXY, NUYZ, NUXZ</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
<tr>
<td>Mass Density</td>
<td>DENS</td>
<td>lbs sec²/in⁴</td>
<td>kg/m³</td>
<td>g sec²/cm⁴</td>
<td>g/cm³</td>
</tr>
<tr>
<td>Coeff. of Thermal Expansion</td>
<td>ALPX, ALPY, ALPZ</td>
<td>in/(in °F)</td>
<td>m/(m °K)</td>
<td>cm/(cm °C)</td>
<td>cm/(cm °K)</td>
</tr>
<tr>
<td>Material Angle</td>
<td>Beta</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Anisotropic Material</td>
<td>MC11, …, MC66</td>
<td>lbs/in² (total of 21 constants)</td>
<td>Newton/m² or Pascal</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Stiffness Matrix Constants</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Loads and Boundary Conditions</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>Temp</td>
<td>°F</td>
<td>°K</td>
<td>°C</td>
<td>°K</td>
</tr>
<tr>
<td>Translational Displacements</td>
<td>UX, UY, UZ</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Rotational Displacements</td>
<td>RX, Ry, RZ</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
</tr>
<tr>
<td>Forces (nodal)</td>
<td>FX, FY, FZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Moments (nodal)</td>
<td>MX, MY, MZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Pressure</td>
<td>P</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>ACCEL</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td>Angular Velocity</td>
<td>OMEGA, CGOMEGA</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
</tr>
<tr>
<td>Angular Acceleration</td>
<td>DOMEGA, DCGOMEGA</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
</tr>
<tr>
<td><strong>Results</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction Forces</td>
<td>UX, UY, UZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Reaction Moments</td>
<td>RX, Ry, RZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Displacements</td>
<td>UX, UY, UZ, RES</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Stresses</td>
<td>SX, SY, SZ, TXY, TYZ, TXZ, P1, P2, P3, VON, INT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Strains</td>
<td>EPSX, EPSY, EPSZ, GMXY, GMYZ, GMXZ, ESTRN</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
</tbody>
</table>
## Table B.5  Table of Consistent Units for Heat Transfer Analysis

<table>
<thead>
<tr>
<th>Description</th>
<th>COSMOS Name</th>
<th>* FPS(^1) (gravitational)</th>
<th>* SI(^2) (absolute)</th>
<th>* MKS(^3) (gravitational)</th>
<th>CGS(^4) (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Measure</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>X, Y, Z</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td><strong>Material Properties</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Thermal Conductivity</td>
<td>KX, KY, KZ</td>
<td>BTU/in sec °F</td>
<td>W/m °K</td>
<td>Cal/cm sec °C</td>
<td>W/cm °K</td>
</tr>
<tr>
<td>Specific Heat</td>
<td>C</td>
<td>BTU in/lb sec(^2) °F</td>
<td>J/kg °K</td>
<td>Cal cm/kg sec(^2) °C</td>
<td>J/g °K</td>
</tr>
<tr>
<td>Mass Density</td>
<td>DENS</td>
<td>lbs sec(^2)/in(^4)</td>
<td>kg/m(^3)</td>
<td>kg sec(^2)/cm(^4)</td>
<td>g/cm(^3)</td>
</tr>
<tr>
<td>Emissivity</td>
<td>EMIS</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
</tr>
<tr>
<td>Stefan Boltzmann Constant</td>
<td>SBCONST</td>
<td>3.3063e-15</td>
<td>5.6703e-8</td>
<td>1.3552e-12</td>
<td>5.6703e-12</td>
</tr>
<tr>
<td>Convection Film Coefficient</td>
<td>HC (for CLINK elements)</td>
<td>BTU/sec in(^2) °F</td>
<td>W/m(^2) °K</td>
<td>Cal/sec cm(^2) °C</td>
<td>W/cm(^2) °K</td>
</tr>
<tr>
<td>Material Angle</td>
<td>Beta</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Electric Conductivity</td>
<td>ECONX</td>
<td>mho/in</td>
<td>mho/m</td>
<td>mho/m</td>
<td>10(^{-13}) mho/cm</td>
</tr>
<tr>
<td><strong>Loads and Boundary Conditions</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>Temp, TREF, NT</td>
<td>°F</td>
<td>°K</td>
<td>°C</td>
<td>°K</td>
</tr>
<tr>
<td>Absolute Temperature Offset</td>
<td>TOFFSET</td>
<td>460 °R</td>
<td>none</td>
<td>273 °K</td>
<td>none</td>
</tr>
<tr>
<td>Time</td>
<td>Time</td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
</tr>
<tr>
<td>Concentrated Heating</td>
<td>Nodal Heat, Q</td>
<td>BTU/sec</td>
<td>Watt</td>
<td>Cal/sec</td>
<td>Watt</td>
</tr>
<tr>
<td>Volume Heating</td>
<td>Element Heat, QE</td>
<td>BTU/sec in(^3)</td>
<td>Wm(^3)</td>
<td>Cal/sec cm(^3)</td>
<td>W/cm(^3)</td>
</tr>
<tr>
<td>Surface Heating</td>
<td>Heat Flux, HX</td>
<td>BTU/sec in(^2)</td>
<td>W/m(^2)</td>
<td>Cal/sec cm(^2)</td>
<td>W/cm(^2)</td>
</tr>
<tr>
<td>Convection Film Coefficient</td>
<td>CE</td>
<td>BTU/sec in(^2) °F</td>
<td>W/m(^2) °K</td>
<td>Cal/sec cm(^2) °C</td>
<td>W/cm(^2) °K</td>
</tr>
<tr>
<td>Radiation Emissivity</td>
<td>RE</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
</tr>
<tr>
<td><strong>Results</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>Temp</td>
<td>°F</td>
<td>°K</td>
<td>°C</td>
<td>°K</td>
</tr>
<tr>
<td>Temperature Gradient</td>
<td>GRADX, GRADY, GRADZ</td>
<td>°F/in</td>
<td>°K/m</td>
<td>°C/cm</td>
<td>°K/cm</td>
</tr>
<tr>
<td>Heat Flux</td>
<td>HFLUXX, HFLUXY, HFLUXZ, HFLUXN</td>
<td>BTU/sec in(^2)</td>
<td>W/m(^2)</td>
<td>Cal/sec cm(^2)</td>
<td>W/cm(^2)</td>
</tr>
</tbody>
</table>
Table B.6 Table of Consistent Units for Linear Dynamic Analysis

<table>
<thead>
<tr>
<th>Description</th>
<th>COSMOS Name</th>
<th>*FPS¹ (gravitational)</th>
<th>*SI² (absolute)</th>
<th>*MKS³ (gravitational)</th>
<th>CGS⁴ (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measure</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>X, Y, Z</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Material Properties</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>EX, EY, EZ</td>
<td>lbs/in²</td>
<td>Newton/m² or Pascal</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>GXY, GYZ, GXZ</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>NUXY, NUYZ, NUXZ</td>
<td>in/in</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
<tr>
<td>Mass Density</td>
<td>DENS</td>
<td>lbs sec²/in⁴</td>
<td>kg/m³</td>
<td>kg sec²/cm⁴</td>
<td>g/cm³</td>
</tr>
<tr>
<td>Material Angle</td>
<td>Beta</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Loads and Boundary Conditions</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td></td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
</tr>
<tr>
<td>Frequency</td>
<td></td>
<td>Hz or rad/sec</td>
<td>Hz or rad/sec</td>
<td>Hz or rad/sec</td>
<td>Hz or rad/sec</td>
</tr>
<tr>
<td>Translational Displacements</td>
<td>DISP (see INITIAL or Base Excitation)</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Linear Velocity</td>
<td>VEL (see INITIAL or Base Excitation)</td>
<td>in/sec</td>
<td>m/sec</td>
<td>cm/sec</td>
<td>cm/sec</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>ACC (see INITIAL or Base Excitation)</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td>Forces (nodal)</td>
<td>FX, FY, FZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Moments (nodal)</td>
<td>MX, MY, MZ</td>
<td>lbs</td>
<td>m N</td>
<td>cm kg</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Pressure</td>
<td>P</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Concentrated Damper</td>
<td>P-dyn Damping, DX, DY, DZ</td>
<td>lbs sec/in</td>
<td>N sec/m</td>
<td>kg sec/cm</td>
<td>dyne sec/cm</td>
</tr>
<tr>
<td>Damping Ratio</td>
<td>P-dyn Damping, (fraction of critical damping, C_r)</td>
<td>C/C_r (no units)</td>
<td>C/C_r (no units)</td>
<td>C/C_r (no units)</td>
<td>C/C_r (no units)</td>
</tr>
<tr>
<td>Phase Angle Input</td>
<td>Phase</td>
<td>degrees</td>
<td>degrees</td>
<td>degrees</td>
<td>degrees</td>
</tr>
<tr>
<td>Power Spectral Density, (PSD)</td>
<td>Acc²/Freq, (see P-dyn Analysis Type)</td>
<td>(in/sec² rad/sec)/Hz</td>
<td>(m/sec² rad/sec)/Hz</td>
<td>(cm/sec² rad/sec)/Hz</td>
<td>(cm/sec² rad/sec)/Hz</td>
</tr>
<tr>
<td>Results</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td></td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
<td>seconds</td>
</tr>
<tr>
<td>Reaction Forces</td>
<td>UX, UY, UZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Reaction Moments</td>
<td>RX, RY, RZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm kg</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Displacements</td>
<td>(DISP), UX, UY, UZ</td>
<td>in/sec</td>
<td>m/sec</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Linear Velocity</td>
<td>(VEL), UX, UY, UZ</td>
<td>in/sec</td>
<td>m/sec</td>
<td>cm/sec</td>
<td>cm/sec</td>
</tr>
<tr>
<td>Angular Velocity</td>
<td>(VEL), RX, RY, RZ</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>(ACC), UX, UY, UZ</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td>Angular Acceleration</td>
<td>(ACC), RX, RY, RZ</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
</tr>
<tr>
<td>Stresses</td>
<td>SX, SY, SZ, TXY, TYZ, TXZ, P1, P2, P3, VON, INT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Strains</td>
<td>EPSX, EPSY, EPSZ, GMXY, GMYZ, GMXZ, ESTRN</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
</tbody>
</table>
Table B.7 Table of Consistent Units for Nonlinear Static Analysis

<table>
<thead>
<tr>
<th>Description</th>
<th>COSMOS Name</th>
<th>* FPS¹ (gravitational)</th>
<th>* SI² (absolute)</th>
<th>* MKS³ (gravitational)</th>
<th>CGS⁴ (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Measure</td>
<td>X, Y, Z</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Material Properties</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>EX, EY, EZ</td>
<td>lbs/in²</td>
<td>Newton/m² or Pascal</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>GXY, GYZ, GXZ</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>NUXY, NUYZ, NUXZ</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
<tr>
<td>Mass Density</td>
<td>DENS</td>
<td>lbs sec²/in⁴</td>
<td>kg/m³</td>
<td>kg sec²/cm⁴</td>
<td>g/cm³</td>
</tr>
<tr>
<td>Coeff. of Thermal Expansion</td>
<td>ALPX, ALPY, ALPZ</td>
<td>in/(in °F)</td>
<td>m/(m °K)</td>
<td>cm/(cm °C)</td>
<td>cm/(cm °K)</td>
</tr>
<tr>
<td>Tangent Modulus</td>
<td>ETAN</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Yield Stress</td>
<td>SIGYLD</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>SIGXT, SIGYT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Compressive Strength</td>
<td>SIGXG, SIGYC</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Tensile Shear Strength</td>
<td>SIGXYT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Compressive Shear Strength</td>
<td>SIGXYC</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Mooney-Rivlin Hyperelastic Constants</td>
<td>MOONEY_A, MOONEY_B</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Creep Constants</td>
<td>CREEPC, CREEPX</td>
<td>Units are determined by the creep law used</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cohesive Strength (Drucker-Prager model)</td>
<td>COHESN</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Friction Angle (Drucker-Prager model)</td>
<td>FRCANG</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Material Angle</td>
<td>Beta</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Loads and Boundary Conditions</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Temperature</td>
<td>Temp</td>
<td>°F</td>
<td>°K</td>
<td>°C</td>
<td>°K</td>
</tr>
<tr>
<td>Load Step</td>
<td>Time</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
<td>no units</td>
</tr>
<tr>
<td>Translational Displacements</td>
<td>UX, UY, UZ</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Rotational Displacements</td>
<td>RX, RY, RZ</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
</tr>
<tr>
<td>Forces (nodal)</td>
<td>FX, FY, FZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Moments (nodal)</td>
<td>MX, MY, MZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm kg</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Pressure</td>
<td>P</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Distributed Beam Load</td>
<td>PB</td>
<td>lbs/in</td>
<td>N/m</td>
<td>kg/cm</td>
<td>dyne/cm</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>ACCEL</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td>Angular Velocity</td>
<td>OMEGA, CGOMEGA</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
</tr>
<tr>
<td>Angular Acceleration</td>
<td>DOMEGA, DCGOMEGA</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
</tr>
<tr>
<td>Results</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction Forces</td>
<td>UX, UY, UZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Reaction Moments</td>
<td>RX, RY, RZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm kg</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Displacements</td>
<td>UX, UY, UZ, RES</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Stresses</td>
<td>SX, SY, SZ, TXY, TYZ, TXZ, P1, P2, P3, VON, INT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Strains</td>
<td>EPSX, EPSY, EPSZ, GMYX, GMYZ, GMXZ, ESTRN</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
</tbody>
</table>
### Table B.8 Table of Consistent Units for Nonlinear Dynamic Analysis

<table>
<thead>
<tr>
<th>Description</th>
<th>COSMOS Name</th>
<th><em>FPS</em> (gravitational)</th>
<th><em>SI</em> (absolute)</th>
<th><em>MKS</em> (gravitational)</th>
<th>CGS (absolute)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Measure</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Length</td>
<td>X, Y, Z</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td><strong>Material Properties</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Elastic Modulus</td>
<td>EX, EY, EZ</td>
<td>lbs/in²</td>
<td>Newton/m² or Pascal</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Shear Modulus</td>
<td>GXY, GYZ, GXZ</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Poisson's Ratio</td>
<td>NUXY, NUYZ, NUXZ</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
<tr>
<td>Mass Density</td>
<td>DENS</td>
<td>lbs sec²/in⁴</td>
<td>kg/m³</td>
<td>kg sec²/cm⁴</td>
<td>g/cm³</td>
</tr>
<tr>
<td>Tangent Modulus</td>
<td>ETAN</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Yield Stress</td>
<td>SIGYLD</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Tensile Strength</td>
<td>SIGXT, SIGYT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Compressive Strength</td>
<td>SIGXC, SIGYC</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Tensile Shear Strength</td>
<td>SIGXYT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Compressive Shear Strength</td>
<td>SIGXYC</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Mooney-Rivlin Hyperelastic Constants</td>
<td>MOONEY_A, MOONEY_B</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Creep Constants</td>
<td>CREEPC, CREEPX</td>
<td>Units are determined by the creep law used</td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Cohesive Strength (Drucker-Prager model)</td>
<td>COHESN</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Friction Angle (Drucker-Prager model)</td>
<td>FRCANG</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td>Material Angle</td>
<td>Beta</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
<td>degree</td>
</tr>
<tr>
<td><strong>Loads and Boundary Conditions</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Time</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Translational Displacements</td>
<td>UX, UY, UZ</td>
<td>in</td>
<td>m</td>
<td>cm</td>
<td>cm</td>
</tr>
<tr>
<td>Rotational Displacements</td>
<td>RX, RY, RZ</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
<td>radians</td>
</tr>
<tr>
<td>Forces (nodal)</td>
<td>FX, FY, FZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Moments (nodal)</td>
<td>MX, MY, MZ</td>
<td>in lbs</td>
<td>m N</td>
<td>cm kg</td>
<td>cm dyne</td>
</tr>
<tr>
<td>Pressure</td>
<td>P</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Distributed Beam Load</td>
<td>PB</td>
<td>lbs/in</td>
<td>N/m</td>
<td>kg/cm</td>
<td>dyne/cm</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>ACEL</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td><strong>Results</strong></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
<tr>
<td>Reaction Forces</td>
<td>UX, UY, UZ</td>
<td>lbs</td>
<td>Newton</td>
<td>kg</td>
<td>dyne</td>
</tr>
<tr>
<td>Reaction Moments</td>
<td>RX, RY, RZ</td>
<td>lbs m N</td>
<td>cm kg</td>
<td>cm dyne</td>
<td></td>
</tr>
<tr>
<td>Displacements</td>
<td>UX, UY, UZ, RES</td>
<td>in m</td>
<td>cm</td>
<td>cm</td>
<td></td>
</tr>
<tr>
<td>Linear Velocity</td>
<td>(VEL), UX, UY, UZ</td>
<td>in/sec m/sec</td>
<td>cm/sec</td>
<td>cm/sec</td>
<td></td>
</tr>
<tr>
<td>Angular Velocity</td>
<td>(VEL), RX, RY, RZ</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
<td>rad/sec</td>
</tr>
<tr>
<td>Linear Acceleration</td>
<td>(ACC), UX, UY, UZ</td>
<td>in/sec²</td>
<td>m/sec²</td>
<td>cm/sec²</td>
<td>cm/sec²</td>
</tr>
<tr>
<td>Angular Acceleration</td>
<td>(ACC), RX, RY, RZ</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
<td>rad/sec²</td>
</tr>
<tr>
<td>Stresses</td>
<td>SX, SY, SZ, TXY, TYZ, TXZ, P1, P2, P3, VON, INT</td>
<td>lbs/in²</td>
<td>N/m² or Pa</td>
<td>kg/cm²</td>
<td>dyne/cm²</td>
</tr>
<tr>
<td>Strains</td>
<td>EPSX, EPSY, EPSZ, GMXY, GMYX, GMYZ, GMXZ, ESTRN</td>
<td>in/in (no units)</td>
<td>m/m (no units)</td>
<td>cm/cm (no units)</td>
<td>cm/cm (no units)</td>
</tr>
</tbody>
</table>
(This page is intentionally left blank)
Appendix C

Material Library
Introduction

The COSMOS/M built-in material library comes with the interface and contains 45 common materials. The Pick Material command should be used to define a material by picking a material from the library.

A material browser supported by a more extensive material library, compiled by Information Indexing Inc. (InfoDex), is optionally available from SRAC. The browser may be started directly from the Interface using the Run Material command and material properties are automatically defined upon selecting a material and exiting the browser. Contact SRAC for more information.

In this appendix, the contents of the COSMOS/M Material Library are listed.

<table>
<thead>
<tr>
<th>Material Name: A_STEEL</th>
<th>Alloy Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.30E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.28</td>
</tr>
<tr>
<td>GXY</td>
<td>0.12E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.74E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.73E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.67E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>42. BTU/in/lbf/s/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ACRYLIC</th>
<th>Acrylic (Medium-High Impact)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.35E+06 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.35</td>
</tr>
<tr>
<td>GXY</td>
<td>0.12E+06 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.29E-04 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.11E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.28E-05 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>0.14E+03 BTU/in/lbf/s/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ALUMINUM</th>
<th>Aluminum Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.10E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.39E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.13E-04 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.25E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.27E-02 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>83. BTU/in/lbf/s/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_1345</th>
<th>Aluminum 1345 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.10E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.39E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.13E-04 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.25E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.29E-02 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>93. BTU/in/lbf/s/s/F</td>
</tr>
<tr>
<td>Material Name: AL_1350          Aluminum 1350 Alloy</td>
<td></td>
</tr>
<tr>
<td>-----------------------------------------------</td>
<td></td>
</tr>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.10E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.39E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.13E-04 °Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.25E-03 lbf*s/s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.31E-02 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>93. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_2014          Aluminum 2014 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>NUXY</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>KX</td>
</tr>
<tr>
<td>C (Cp)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_2018          Aluminum 2018 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>NUXY</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>KX</td>
</tr>
<tr>
<td>C (Cp)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_2024          Aluminum 2024 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>NUXY</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>KX</td>
</tr>
<tr>
<td>C (Cp)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_3003          Aluminum 3003 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>NUXY</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>KX</td>
</tr>
<tr>
<td>C (Cp)</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: AL_6061          Aluminum 6061 Alloy</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
</tr>
<tr>
<td>EX</td>
</tr>
<tr>
<td>NUXY</td>
</tr>
<tr>
<td>GXY</td>
</tr>
<tr>
<td>ALPX</td>
</tr>
<tr>
<td>DENS</td>
</tr>
<tr>
<td>KX</td>
</tr>
<tr>
<td>C (Cp)</td>
</tr>
</tbody>
</table>
### Material Name: AL_7079  Aluminum 7079 Alloy

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.10E+08 psi</td>
<td>0.73E+06 Kg/cm/cm</td>
<td>0.72E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
<td>0.33</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.39E+07 psi</td>
<td>0.27E+06 Kg/cm/cm</td>
<td>0.27E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.14E-04 /Fahrenheit</td>
<td>0.25E-04 /Centigrade</td>
<td>0.25E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.26E-03 lbf*s/s/in**4</td>
<td>0.28E-05 Kgf*s/s/cm**4</td>
<td>0.27E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.16E-02 BTU/in/s/F</td>
<td>0.29 Cal/cm/s/C</td>
<td>0.12E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>89</td>
<td>0.23E+06 Cal*cm/kgf/s/C</td>
<td>0.96E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: AL_BRONZE  Aluminum Bronze

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.17E+08 psi</td>
<td>0.12E+07 Kg/cm/cm</td>
<td>0.11E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.30</td>
<td>0.30</td>
<td>0.30</td>
</tr>
<tr>
<td>GXY</td>
<td>0.62E+07 psi</td>
<td>0.44E+06 Kg/cm/cm</td>
<td>0.43E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.95E-05 /Fahrenheit</td>
<td>0.17E-04 /Centigrade</td>
<td>0.17E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.70E-03 lbf*s/s/in**4</td>
<td>0.76E-05 Kgf*s/s/cm**4</td>
<td>0.74E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.75E-03 BTU/in/s/F</td>
<td>0.13 Cal/cm/s/C</td>
<td>56. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>35</td>
<td>0.88E+05 Cal*cm/kgf/s/C</td>
<td>0.38E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: BRASS  Silicon Brass & Bronze

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.15E+08 psi</td>
<td>0.11E+07 Kg/cm/cm</td>
<td>0.10E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.37</td>
<td>0.37</td>
<td>0.37</td>
</tr>
<tr>
<td>GXY</td>
<td>0.54E+07 psi</td>
<td>0.38E+06 Kg/cm/cm</td>
<td>0.37E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.10E-04 /Fahrenheit</td>
<td>0.18E-04 /Centigrade</td>
<td>0.18E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.80E-03 lbf*s/s/in**4</td>
<td>0.87E-05 Kgf*s/s/cm**4</td>
<td>0.85E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.15E-02 BTU/in/s/F</td>
<td>0.26 Cal/cm/s/C</td>
<td>0.11E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>36</td>
<td>0.90E+05 Cal*cm/kgf/s/C</td>
<td>0.39E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: BRONZE  Bronze

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.16E+08 psi</td>
<td>0.11E+07 Kg/cm/cm</td>
<td>0.11E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.37</td>
<td>0.37</td>
<td>0.37</td>
</tr>
<tr>
<td>GXY</td>
<td>0.59E+07 psi</td>
<td>0.41E+06 Kg/cm/cm</td>
<td>0.41E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.18E-04 /Fahrenheit</td>
<td>0.32E-04 /Centigrade</td>
<td>0.32E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.81E-03 lbf*s/s/in**4</td>
<td>0.88E-05 Kgf*s/s/cm**4</td>
<td>0.87E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.35E-03 BTU/in/s/F</td>
<td>0.62E-01 Cal/cm/s/C</td>
<td>26. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>32</td>
<td>0.81E+05 Cal*cm/kgf/s/C</td>
<td>0.34E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: CA_STEEL  Cast Alloy Steel (Below 8% Content)

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.28E+08 psi</td>
<td>0.20E+07 Kg/cm/cm</td>
<td>0.19E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.26</td>
<td>0.26</td>
<td>0.26</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
<td>0.79E+06 Kg/cm/cm</td>
<td>0.78E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.81E-05 /Fahrenheit</td>
<td>0.15E-04 /Centigrade</td>
<td>0.15E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.68E-03 lbf*s/s/in**4</td>
<td>0.74E-05 Kgf*s/s/cm**4</td>
<td>0.73E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.50E-03 BTU/in/s/F</td>
<td>0.90E-01 Cal/cm/s/C</td>
<td>38. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>41</td>
<td>0.10E+06 Cal*cm/kgf/s/C</td>
<td>0.44E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: COBALT  Cobalt

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.33E+08 psi</td>
<td>0.23E+07 Kg/cm/cm</td>
<td>0.23E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
</tr>
<tr>
<td>GXY</td>
<td>0.13E+08 psi</td>
<td>0.90E+06 Kg/cm/cm</td>
<td>0.88E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.68E-05 /Fahrenheit</td>
<td>0.12E-04 /Centigrade</td>
<td>0.12E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.83E-03 lbf*s/s/in**4</td>
<td>0.91E-05 Kgf*s/s/cm**4</td>
<td>0.89E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.92E-03 BTU/in/s/F</td>
<td>0.16 Cal/cm/s/C</td>
<td>65. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>38</td>
<td>0.96E+05 Cal*cm/kgf/s/C</td>
<td>0.41E+03 J/kgm/K</td>
</tr>
</tbody>
</table>
### Material Name: Copper

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.16E+08 psi</td>
<td>0.11E+07 Kg/cm/cm</td>
<td>0.11E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.37</td>
<td>0.37</td>
<td>0.37</td>
</tr>
<tr>
<td>GXY</td>
<td>0.58E+07 psi</td>
<td>0.41E+06 Kg/cm/cm</td>
<td>0.40E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.13E-04 /Fahrenheit</td>
<td>0.24E-04 /Centigrade</td>
<td>0.24E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.84E-03 lbf*s/s/in**4</td>
<td>0.91E-05 Kgf*s/s/cm**4</td>
<td>0.89E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.52E-02 BTU/in/s/F</td>
<td>0.92 Cal/cm/s/C</td>
<td>0.39E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>36</td>
<td>0.90E+05 Cal*cm/kgf/s/s/C</td>
<td>0.39E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: CS_Steel

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.28E+08 psi</td>
<td>0.20E+07 Kg/cm/cm</td>
<td>0.19E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.26</td>
<td>0.26</td>
<td>0.26</td>
</tr>
<tr>
<td>GXY</td>
<td>0.12E+08 psi</td>
<td>0.81E+06 Kg/cm/cm</td>
<td>0.79E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.84E-05 /Fahrenheit</td>
<td>0.15E-04 /Centigrade</td>
<td>0.15E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.73E-03 lbf*s/s/in**4</td>
<td>0.79E-05 Kgf*s/s/cm**4</td>
<td>0.77E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.50E-03 BTU/in/s/F</td>
<td>0.90E-01 Cal/cm/s/C</td>
<td>0.37   W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>48</td>
<td>0.12E+06 Cal*cm/kgf/s/s/C</td>
<td>0.52E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: GC_Iron

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.17E+08 psi</td>
<td>0.12E+07 Kg/cm/cm</td>
<td>0.12E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.27</td>
<td>0.27</td>
<td>0.27</td>
</tr>
<tr>
<td>GXY</td>
<td>0.72E+07 psi</td>
<td>0.51E+06 Kg/cm/cm</td>
<td>0.50E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.66E-05 /Fahrenheit</td>
<td>0.12E-04 /Centigrade</td>
<td>0.12E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.67E-03 lbf*s/s/in**4</td>
<td>0.73E-05 Kgf*s/s/cm**4</td>
<td>0.72E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.60E-03 BTU/in/s/F</td>
<td>0.11 Cal/cm/s/C</td>
<td>0.37   W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>47</td>
<td>0.12E+06 Cal*cm/kgf/s/s/C</td>
<td>0.51E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: Glass

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.92E+07 psi</td>
<td>0.64E+06 Kg/cm/cm</td>
<td>0.63E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.22</td>
<td>0.22</td>
<td>0.22</td>
</tr>
<tr>
<td>GXY</td>
<td>0.15E+08 psi</td>
<td>0.10E+07 Kg/cm/cm</td>
<td>0.10E+12 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.55E-05 /Fahrenheit</td>
<td>0.99E-05 /Centigrade</td>
<td>0.99E-05 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.28E-03 lbf*s/s/in**4</td>
<td>0.31E-05 Kgf*s/s/cm**4</td>
<td>0.30E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.22E-04 BTU/in/s/F</td>
<td>0.38E-02 Cal/cm/s/C</td>
<td>0.16   W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>70</td>
<td>0.18E+06 Cal*cm/kgf/s/s/C</td>
<td>0.75E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: Gold

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.11E+08 psi</td>
<td>0.76E+06 Kg/cm/cm</td>
<td>0.74E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.42</td>
<td>0.42</td>
<td>0.42</td>
</tr>
<tr>
<td>GXY</td>
<td>0.38E+07 psi</td>
<td>0.27E+06 Kg/cm/cm</td>
<td>0.26E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.79E-05 /Fahrenheit</td>
<td>0.14E-04 /Centigrade</td>
<td>0.14E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.18E-02 lbf*s/s/in**4</td>
<td>0.20E-04 Kgf*s/s/cm**4</td>
<td>0.19E+05 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.40E-02 BTU/in/s/F</td>
<td>0.71 Cal/cm/s/C</td>
<td>0.30E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>12</td>
<td>0.30E+05 Cal*cm/kgf/s/s/C</td>
<td>0.13E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: Iron

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.17E+08 psi</td>
<td>0.12E+07 Kg/cm/cm</td>
<td>0.12E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
<td>0.78E+06 Kg/cm/cm</td>
<td>0.77E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.61E-05 /Fahrenheit</td>
<td>0.11E-04 /Centigrade</td>
<td>0.11E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.74E-03 lbf*s/s/in**4</td>
<td>0.80E-05 Kgf*s/s/cm**4</td>
<td>0.79E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.10E-02 BTU/in/s/F</td>
<td>0.18 Cal/cm/s/C</td>
<td>0.36E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>42</td>
<td>0.11E+06 Cal*cm/kgf/s/s/C</td>
<td>0.45E+03 J/kgm/K</td>
</tr>
</tbody>
</table>
### Material Name: LEAD  Pure Lead

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.20E+07 psi</td>
<td>0.14E+06 Kgf/cm/cm</td>
<td>0.14E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.40</td>
<td>0.40</td>
<td>0.40</td>
</tr>
<tr>
<td>GXY</td>
<td>0.71E+06 psi</td>
<td>0.50E+05 Kgf/cm/cm</td>
<td>0.49E+10 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.29E-04 /Fahrenheit</td>
<td>0.53E-04 /Centigrade</td>
<td>0.53E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.11E-02 lbf*s/s/in**4</td>
<td>0.12E-04 Kgf*s/s/cm**4</td>
<td>0.11E+05 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.47E-03 BTU/in/s/F</td>
<td>0.83E-01 Cal/cm/s/C</td>
<td>35. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>12. BTU*in/lbf/s/F</td>
<td>0.30E+05 Cal*cm/kgf/s/C</td>
<td>0.13E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: MAGNES  Magnesium Alloy - Wrought or Cast

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.65E+07 psi</td>
<td>0.46E+06 Kgf/cm/cm</td>
<td>0.45E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.35</td>
<td>0.35</td>
<td>0.35</td>
</tr>
<tr>
<td>GXY</td>
<td>0.24E+07 psi</td>
<td>0.17E+06 Kgf/cm/cm</td>
<td>0.17E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.14E-04 /Fahrenheit</td>
<td>0.25E-04 /Centigrade</td>
<td>0.25E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.16E-03 lbf*s/s/in**4</td>
<td>0.18E-05 Kgf*s/s/cm**4</td>
<td>0.17E+04 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.21E-02 BTU/in/s/F</td>
<td>0.38 Cal/cm/s/C</td>
<td>47. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>97. BTU*in/lbf/s/F</td>
<td>0.25E+06 Cal*cm/kgf/s/C</td>
<td>0.10E+04 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: MC_IRON  Malleable Cast Iron (ASTM - A220)

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.27E+08 psi</td>
<td>0.19E+07 Kgf/cm/cm</td>
<td>0.19E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.27</td>
<td>0.27</td>
<td>0.27</td>
</tr>
<tr>
<td>GXY</td>
<td>0.13E+08 psi</td>
<td>0.88E+06 Kgf/cm/cm</td>
<td>0.86E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.67E-05 /Fahrenheit</td>
<td>0.12E-04 /Centigrade</td>
<td>0.12E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.68E-03 lbf*s/s/in**4</td>
<td>0.74E-05 Kgf*s/s/cm**4</td>
<td>0.73E+04 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.63E-03 BTU/in/s/F</td>
<td>0.11 Cal/cm/s/C</td>
<td>47. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>47. BTU*in/lbf/s/F</td>
<td>0.12E+06 Cal*cm/kgf/s/C</td>
<td>0.51E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: MN_BRONZE  Manganese Bronze

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.15E+08 psi</td>
<td>0.11E+07 Kgf/cm/cm</td>
<td>0.11E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.37</td>
<td>0.37</td>
<td>0.37</td>
</tr>
<tr>
<td>GXY</td>
<td>0.56E+07 psi</td>
<td>0.39E+06 Kgf/cm/cm</td>
<td>0.39E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.12E-04 /Fahrenheit</td>
<td>0.22E-04 /Centigrade</td>
<td>0.22E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.78E-03 lbf*s/s/in**4</td>
<td>0.85E-05 Kgf*s/s/cm**4</td>
<td>0.83E+04 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.13E-03 BTU/in/s/F</td>
<td>0.23E-01 Cal/cm/s/C</td>
<td>9.6 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>35. BTU*in/lbf/s/F</td>
<td>0.88E+05 Cal*cm/kgf/s/C</td>
<td>0.38E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: MOLYBDENUM  Molybdenum

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.47E+08 psi</td>
<td>0.33E+07 Kgf/cm/cm</td>
<td>0.32E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.38</td>
<td>0.38</td>
<td>0.38</td>
</tr>
<tr>
<td>GXY</td>
<td>0.17E+08 psi</td>
<td>0.12E+07 Kgf/cm/cm</td>
<td>0.12E+12 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.28E-05 /Fahrenheit</td>
<td>0.50E-05 /Centigrade</td>
<td>0.50E-05 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.96E-03 lbf*s/s/in**4</td>
<td>0.10E-04 Kgf*s/s/cm**4</td>
<td>0.10E+05 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.20E-02 BTU/in/s/F</td>
<td>0.35 Cal/cm/s/C</td>
<td>0.15E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>25. BTU*in/lbf/s/F</td>
<td>0.64E+05 Cal*cm/kgf/s/C</td>
<td>0.27E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: MONEL  Monel 400 Alloy of Nickel

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.26E+08 psi</td>
<td>0.18E+07 Kgf/cm/cm</td>
<td>0.18E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.34</td>
<td>0.34</td>
<td>0.34</td>
</tr>
<tr>
<td>GXY</td>
<td>0.97E+07 psi</td>
<td>0.68E+06 Kgf/cm/cm</td>
<td>0.67E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.77E-05 /Fahrenheit</td>
<td>0.14E-04 /Centigrade</td>
<td>0.14E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.83E-03 lbf*s/s/in**4</td>
<td>0.90E-05 Kgf*s/s/cm**4</td>
<td>0.88E+04 Kgm/m**3</td>
</tr>
<tr>
<td>X</td>
<td>0.29E-01 BTU/in/s/F</td>
<td>0.52E-01 Cal/cm/s/C</td>
<td>22. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>39. BTU*in/lbf/s/F</td>
<td>0.10E+06 Cal*cm/kgf/s/C</td>
<td>0.43E+03 J/kgm/K</td>
</tr>
</tbody>
</table>
### Material Name: NICKEL  
**Nickel**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.31E+08 psi</td>
<td>0.21E+07 Kg/cm/cm</td>
<td>0.21E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.31</td>
<td>0.31</td>
<td>0.31</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
<td>0.81E+06 Kg/cm/cm</td>
<td>0.79E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.92E-05 /Fahrenheit</td>
<td>0.17E-04 /Centigrade</td>
<td>0.17E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.80E-03 lbf*s/s/in**4</td>
<td>0.87E-05 Kg<em>s</em>s/cm**4</td>
<td>0.85E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.58E-03 BTU/in/s/F</td>
<td>0.10 Cal/cm/s/C</td>
<td>43. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>42. BTU*in/lbf/s/F</td>
<td>0.11E+06 Cal*cm/kgf/s/s/C</td>
<td>0.46E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: D_NICKEL  
**Duranickel 301**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.30E+08 psi</td>
<td>0.21E+07 Kg/cm/cm</td>
<td>0.21E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.34</td>
<td>0.34</td>
<td>0.34</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
<td>0.79E+06 Kg/cm/cm</td>
<td>0.77E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.82E-05 /Fahrenheit</td>
<td>0.15E-04 /Centigrade</td>
<td>0.15E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.77E-03 lbf*s/s/in**4</td>
<td>0.84E-05 Kg<em>s</em>s/cm**4</td>
<td>0.82E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.32E-03 BTU/in/s/F</td>
<td>0.57E-01 Cal/cm/s/C</td>
<td>24. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>54. BTU*in/lbf/s/F</td>
<td>0.14E+06 Cal*cm/kgf/s/s/C</td>
<td>0.59E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: NYLON  
**Nylon 6/10**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.12E+07 psi</td>
<td>0.84E+05 Kg/cm/cm</td>
<td>0.83E+10 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.28</td>
<td>0.28</td>
<td>0.28</td>
</tr>
<tr>
<td>GXY</td>
<td>0.47E+06 psi</td>
<td>0.33E+05 Kg/cm/cm</td>
<td>0.32E+10 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.16E+04 /Fahrenheit</td>
<td>0.30E+04 /Centigrade</td>
<td>0.30E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.13E-03 lbf*s/s/in**4</td>
<td>0.14E-05 Kg<em>s</em>s/cm**4</td>
<td>0.14E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.70E-05 BTU/in/s/F</td>
<td>0.13E-02 Cal/cm/s/C</td>
<td>0.53 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>0.14E+03 BTU*in/lbf/s/F</td>
<td>0.34E-06 Cal*cm/kgf/s/s/C</td>
<td>0.15E+04 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: PC_STEEL  
**Plain Carbon Steel**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.30E+08 psi</td>
<td>0.21E+07 Kg/cm/cm</td>
<td>0.21E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.28</td>
<td>0.28</td>
<td>0.28</td>
</tr>
<tr>
<td>GXY</td>
<td>0.12E+08 psi</td>
<td>0.81E+06 Kg/cm/cm</td>
<td>0.79E+11 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.74E-05 /Fahrenheit</td>
<td>0.13E-04 /Centigrade</td>
<td>0.13E-04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.73E-03 lbf*s/s/in**4</td>
<td>0.79E-05 Kg<em>s</em>s/cm**4</td>
<td>0.78E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.57E-03 BTU/in/s/F</td>
<td>0.10 Cal/cm/s/C</td>
<td>43. W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>41. BTU*in/lbf/s/F</td>
<td>0.10E+06 Cal*cm/kgf/s/s/C</td>
<td>0.44E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: PORCELAIN  
**Ceramic Porcelain**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.15E+08 psi</td>
<td>0.11E+07 Kg/cm/cm</td>
<td>0.10E+12 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.22</td>
<td>0.22</td>
<td>0.22</td>
</tr>
<tr>
<td>GXY</td>
<td>0.15E+08 psi</td>
<td>0.10E+07 Kg/cm/cm</td>
<td>0.10E+12 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.35E-05 /Fahrenheit</td>
<td>0.63E-05 /Centigrade</td>
<td>0.63E-05 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.24E-03 lbf*s/s/in**4</td>
<td>0.26E-05 Kg<em>s</em>s/cm**4</td>
<td>0.25E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.68E-03 BTU/in/s/F</td>
<td>0.12E-01 Cal/cm/s/C</td>
<td>5.0 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>81. BTU*in/lbf/s/F</td>
<td>0.21E+06 Cal*cm/kgf/s/s/C</td>
<td>0.88E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

### Material Name: RUBBER  
**Rubber**

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>EX</td>
<td>0.88E+03 psi</td>
<td>62. Kg/cm/cm</td>
<td>0.61E+07 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.4</td>
<td>0.49</td>
<td>0.49</td>
</tr>
<tr>
<td>GXY</td>
<td>0.42E+03 psi</td>
<td>29. Kg/cm/cm</td>
<td>0.29E+07 Pascals</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.37E-03 /Fahrenheit</td>
<td>0.67E-03 /Centigrade</td>
<td>0.67E-03 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.93E-04 lbf*s/s/in**4</td>
<td>0.10E-05 Kg<em>s</em>s/cm**4</td>
<td>0.10E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.19E-05 BTU/in/s/F</td>
<td>0.34E-03 Cal/cm/s/C</td>
<td>0.14 W/m/K</td>
</tr>
<tr>
<td>Material Name: SILVER</td>
<td>Pure Silver</td>
<td></td>
<td></td>
</tr>
<tr>
<td>--------------------------------</td>
<td>---------------------------------------</td>
<td></td>
<td></td>
</tr>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
<td><strong>Value (MKS)</strong></td>
<td><strong>Value (SI)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.10E+08 psi</td>
<td>0.70E+06 Kgf/cm/cm</td>
<td>0.69E+11 Pascals</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.37</td>
<td>0.37</td>
<td>0.25E+11 Pascals</td>
</tr>
<tr>
<td>GXY</td>
<td>0.36E+07 psi</td>
<td>0.26E+06 Kgf/cm/cm</td>
<td>0.20E-04 Kgf/cm/cm</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.11E-04 /Fahrenheit</td>
<td>0.20E-04 /Centigrade</td>
<td>0.05E+04 /Kelvin</td>
</tr>
<tr>
<td>DENS</td>
<td>0.98E-03 lbf*s/in**4</td>
<td>0.11E-04 Kgf*s/cm**4</td>
<td>0.11E+05 Kgm/m°</td>
</tr>
<tr>
<td>KX</td>
<td>0.56E-02 BTU/in/s/F</td>
<td>1.0 Cal/cm/s/C</td>
<td>0.42E+03 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>22. BTU*in/lbf/s/F</td>
<td>0.55E+05 Cal*kgf/s/C</td>
<td>0.23E+03 J/kgm/K</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: STEEL</th>
<th>Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.29E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.32</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.84E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.74E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.11E-04 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>46. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ST_1020</th>
<th>Steel, AISI C1020 (Hot Worked)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.29E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.29</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.84E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.74E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.39E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>39. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ST_304</th>
<th>Steel, AISI 304 (Sheet)</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.29E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.29</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.84E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.74E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.39E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>39. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ST_ST</th>
<th>Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.29E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.28</td>
</tr>
<tr>
<td>GXY</td>
<td>0.11E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.84E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.74E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.39E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>39. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: T_BRONZE</th>
<th>Tin Bearing Bronze</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Property Name</strong></td>
<td><strong>Value (FPS)</strong></td>
</tr>
<tr>
<td>EX</td>
<td>0.13E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.54E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.10E-04 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.78E-03 lbf*s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.11E-04 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>35. BTU*in/lbf/s/F</td>
</tr>
<tr>
<td>Material Name: TITANIUM</td>
<td>Titanium</td>
</tr>
<tr>
<td>------------------------</td>
<td>---------</td>
</tr>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.16E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.30</td>
</tr>
<tr>
<td>GXY</td>
<td>0.62E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.49E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.43E-03 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.29E-01 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>2. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: TUNGSTEN</th>
<th>Tungsten</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.59E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.28</td>
</tr>
<tr>
<td>GXY</td>
<td>0.23E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.25E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.18E-02 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.27E-01 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>12. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: VANADIUM</th>
<th>Vanadium</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.20E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.36</td>
</tr>
<tr>
<td>GXY</td>
<td>0.74E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.46E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.57E-03 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.41E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>46. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: W_COPPER</th>
<th>Wrought Copper</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.16E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.33</td>
</tr>
<tr>
<td>GXY</td>
<td>0.58E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.11E-04 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.79E-03 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.31E-02 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>35. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: WS_STEEL</th>
<th>Wrought Stainless Steel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.29E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.26</td>
</tr>
<tr>
<td>GXY</td>
<td>0.12E+08 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.60E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.75E-03 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.25E-03 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>46. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Material Name: ZIRCONIUM</th>
<th>Zirconium</th>
</tr>
</thead>
<tbody>
<tr>
<td>Property Name</td>
<td>Value (FPS)</td>
</tr>
<tr>
<td>EX</td>
<td>0.14E+08 psi</td>
</tr>
<tr>
<td>NUXY</td>
<td>0.34</td>
</tr>
<tr>
<td>GXY</td>
<td>0.53E+07 psi</td>
</tr>
<tr>
<td>ALPX</td>
<td>0.61E-05 /Fahrenheit</td>
</tr>
<tr>
<td>DENS</td>
<td>0.62E-03 lbf<em>s</em>s/in**4</td>
</tr>
<tr>
<td>KX</td>
<td>0.22E-04 BTU/in/s/F</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>27. BTU*in/lbf/s/F</td>
</tr>
</tbody>
</table>
### Material Name: WATER (300 K)

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENS</td>
<td>0.93E-04 lbf<em>s</em>s/in**4</td>
<td>0.10E-05 Kgf<em>s</em>s/cm**4</td>
<td>1.0E+04 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.82E-05 BTU/in/s/F</td>
<td>0.15E-02 Cal/cm/s/C</td>
<td>0.61 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>0.39E+03 BTU*lbf/s/F</td>
<td>0.98E+06 Cal*kgf/s/C</td>
<td>0.42E+04 J/kgm/K</td>
</tr>
<tr>
<td>VISC</td>
<td>0.13E-06 lbf*s/in</td>
<td>0.88E-08 kgf*s/cm</td>
<td>0.87E-03 kgm/m/s</td>
</tr>
<tr>
<td>BETA</td>
<td>0.15E-03 / Rankin</td>
<td>0.27E-03 /Kelvin</td>
<td></td>
</tr>
</tbody>
</table>

### Material Name: AIR (300 K)

<table>
<thead>
<tr>
<th>Property Name</th>
<th>Value (FPS)</th>
<th>Value (MKS)</th>
<th>Value (SI)</th>
</tr>
</thead>
<tbody>
<tr>
<td>DENS</td>
<td>0.10E-06 lbf<em>s</em>s/in**4</td>
<td>0.11E-08 Kgf<em>s</em>s/cm**4</td>
<td>1.1 Kgm/m**3</td>
</tr>
<tr>
<td>KX</td>
<td>0.36E-06 BTU/in/s/F</td>
<td>0.64E-04 Cal/cm/s/C</td>
<td>0.27E-01 W/m/K</td>
</tr>
<tr>
<td>C (Cp)</td>
<td>93. BTU*lbf/s/F</td>
<td>0.24E+06 Cal*kgf/s/C</td>
<td>0.10E+04 J/kgm/K</td>
</tr>
<tr>
<td>VISC</td>
<td>0.27E-08 lbf*s/in</td>
<td>0.19E-09 kgf*s/cm</td>
<td>0.19E-04 kgm/m/s</td>
</tr>
<tr>
<td>BETA</td>
<td>0.19E-02 / Rankin</td>
<td>0.33E-02 /Kelvin</td>
<td>0.33E-02 /Kelvin</td>
</tr>
<tr>
<td>GAMMA</td>
<td>1.40</td>
<td>1.40</td>
<td>1.40</td>
</tr>
</tbody>
</table>
Appendix D

GEOSTAR Mode Commands
Introduction

This appendix provides a brief description for all GEOSTAR commands. GEOSTAR is the pre- and postprocessor for COSMOS/M. All GEOSTAR commands are accessible in the GEOSTAR Command mode available in the COSMOS/M CAD Interface. Note that no menus or on-line help is available for GEOSTAR commands in the COSMOS/M CAD Interface. Some selected GEOSTAR commands are explained in more detail in Chapter 3. Full help is available in the COSMOS/M manuals. The user must activate the GEOSTAR Command mode and type in the command name. The user is then prompted to specify the required input. GEOSTAR commands are based upon the following main menus.

- GEOMETRY to build the model's geometry.
- MESHING to mesh models; create and modify nodes and elements.
- PROPSETS for element groups, material, and real constant sets.
- LOADS-BC for loadings and boundary conditions.
- CONTROL to save, activate, select, interact with FE packages, ...
- DISPLAY to control the grid, view, and display parameters.
- ANALYSIS to specify and execute desired analysis.
- RESULTS to activate, list, and plot analysis results.

GEOMETRY Menu

This menu includes the following submenus:

- GRID A menu to define a plane and control the grid type.
- POINTS A menu to generate and edit keypoints.
- CURVES A menu to generate and edit curves.
- SURFACES A menu to generate and edit surfaces.
- VOLUMES A menu to generate and edit volumes.
- CONTOURS A menu to generate and edit contours.
- REGIONS A menu to generate and edit regions.
- POLYHEDRA A menu to generate and edit polyhedra.
- PARTS A menu to generate and edit parts.
- COORD_SYS A menu to generate and edit coordinate systems.

GRID Submenu

- PLANE Defines a plane parallel to the X-Y, X-Z, or Y-Z planes.
- GRIDON Plots a grid in the active plane.
- GRIDOFF Erases the active grid from the screen.

POINTS Submenu

- PT Creates a keypoint at specified coordinates.
- PTMERGE Merges keypoints within specified tolerance.
- PTTOL Specifies tolerance for keypoint merging.

The POINTS menu also includes the following submenus:

- PTGENR A menu to generate keypoints from existing ones.
- PTEDIT A menu to edit existing keypoints.
Appendix D   GEOSTAR Mode Commands

PTGENR Sub-submenu

PTRELOC Relocates keypoints by translation and/or rotation.
PTMOVE Moves keypoints from one coordinate system to another.
PTFLIP Flips keypoints about a plane.
PTGEN Generates a pattern of keypoints from existing ones.
PTCOPY Copies keypoints from one coordinate system to another.
PTSYM Creates keypoints by symmetry about a plane.
PTINTCC Creates keypoints at intersections of curves.
PTINTCS Creates keypoints at intersections of a curve with surfaces.
PTONCR Creates keypoints on curves using parametric coordinates.
PTONSF Creates keypoints on surfaces using parametric coordinates.
PTND Creates a keypoint at the location of a node.

PTEDIT Sub-submenu

PTIDENT Highlights and displays coordinates and label of a keypoint.
PTCOMPRESS Compresses keypoints on the screen.
PTPLOT Plots keypoints on the screen.
PTLIST Lists keypoints on the screen.
PTDEL Deletes keypoints from the database.
PTUNDEL Undeletes keypoints from the database.
PTERASE Erases keypoints from the screen.

CURVES Submenu

CRPCORD Snap-draws continuous lines to an active grid.
CRSKETCH Sketches continuous lines/arcs by specifying dimensions.
CRSPOLY Snap-draws continuous lines/arcs to an active grid.
CRLINE Connects 2 keypoints by a line.
CRPLINE Connects a series of keypoints by lines.
CR4PT Creates a cubic curve passing through 4 keypoints.
CRARC Creates a circular arc.
CRCONIC Creates a parabola, hyperbola, or ellipse.
CRELLIPSE Creates an ellipse from a center and 2 points.
CRHELIX Creates a helical curve using 3 keypoints and an angle.
CRGFORM Creates a curve by specifying 12 geometric parameters.
CRFITCORD Snap-draws points for curve fit to an active grid.
CRFIT Fits a curve over a series of keypoints.

The CURVES menu also includes the following submenus:

CIRCLES A menu to create circular curves.
SPLINES A menu to generate spline curves.
CRMANIP A menu to manipulate existing curves.
CRGENR A menu to generate curves from existing curves.
CREDIT A menu to edit existing curves.

CIRCLES Sub-submenu

CRSCIRCLE Snap-draws a circle using the center and an edge point.
CRARCCPT Creates a circular arc using a center and two edge points.
CRARCCPT Creates a circular arc passing through 3 points.
CRCIRCLE Creates a circular arc in space.
CRPCIRCLE  Creates a circular arc using 2 points, radius and an angle.
CRPCIRDIA  Creates a circular arc using 2 points and an angle.

SPLINES Sub-submenu
CRBEZCORD  Snap-draws 4 keypoints and a Bezier curve to an active grid.
CRSPLCORD  Snap-draws keypoints and joins them by splines.
CRBSPLCORD  Snap-draws keypoints and joins them by B-splines.
CRBEZIER  Creates a Bezier curve using 4 controlling keypoints.
CRSPLINE  Generates a spline from a series of keypoints.
CRBSPLINE  Generates a B-spline from a series of controlling keypoints.

CRMANIP Sub-submenu
CRREPAR  Reverses the direction of a curve.
CRBLEND  Blends 2 curves.
CREXTEND  Extends a curve.
CRJOIN  Joins 2 curves by a new curve.
CRFILLET  Creates a fillet between 2 curves.
CRTANPT  Creates a tangent from a keypoint to a curve.
CRNORMPT  Creates a normal from a keypoint to a curve.
CRTANLIN  Creates a tangent common to 2 curves.
CRWAVRG  Creates a curve by weighted averaging of 2 curves.
CRBRK  Breaks a curve into a number of equal segments.
CRPTBRK  Breaks a curve near a keypoint.
CRNUBRK  Breaks a curve into segments using parametric coordinates.
CRMERGE  Merges curves within the specified tolerance.

CRGENR Sub-submenu
CREXTR  Generates lines by extruding keypoints along an axis.
CRRELOC  Relocates curves by translation and/or rotation.
CRMOVE  Moves curves from one coordinate system to another.
CRRESIZ  Resizes a pattern of existing curves.
CRFLIP  Flips curves about a plane.
CRGEN  Generates a pattern of curves from existing ones.
CRCOPY  Copies curves from one coordinate system to another.
CRSCALE  Generates curves from existing ones with a scale factor.
CRSYM  Creates curves by symmetry about a plane.
CRINTCC  Breaks curves at the intersections with a primary curve.
CRINTSC  Breaks curves at the intersections of curves with a surface.
CRINTSS  Generates curves at the intersection of surfaces.
CRONSF  Creates a curve on a surface.

CREDIT Sub-submenu
CRIDENT  Highlights a curve and displays its label.
CRCOMPRESS  Compresses curves by removing numbering gaps.
CRPLOT  Plots curves on the screen.
CRLIST  Lists curves on the screen.
CRDEL  Deletes curves from the database.
CRUNDEL  Undeletes curves from the database.
CRERASE  Erases curves from the screen.
SURFACES Submenu

SF3CORD Snap-draws a 3-sided surface to an active grid.
SF4CORD Snap-draws a 4-sided surface to an active grid.
SF3PT Creates a 3-sided surface from 3 keypoints.
SF4PT Creates a 4-sided surface from 4 keypoints.
SF16PT Creates a surface from 12 boundary and 4 interior keypoints.
SFPTCR Creates a 3-sided surface using a keypoint and a curve.
SF2CR Creates a surface using 2 curves.
SF3CR Creates a 3-sided surface using 3 curves.
SF4CR Creates a 4-sided surface using 4 curves.
SF4PCR Creates a surface passing through 4 (parallel) curves.
SFGFORM Creates a surface by specifying 48 geometric parameters.

The SURFACES menu also includes the following submenus:

SFMANIP A menu to manipulate existing surfaces.
SFGENR A menu to generate surfaces using existing ones.
SFEDIT A menu to edit existing surfaces.

SFMANIP Sub-submenu

SFREORNT Changes the surface orientation.
SFREPAR Changes the parametric curve of a surface.
SFFILLET Creates a fillet between 2 surfaces.
SFWAVRG Creates a surface by the weighted averaging of 2 surfaces.
SFBRK Breaks a surface using parametric coordinates of a keypoint.
SFPTBRK Breaks a surface at a specified keypoint.

SFGENR Sub-submenu

SFXEXTR Generates surfaces by extruding curves along an axis.
SFSWEEP Creates surfaces by sweeping curves about an axis.
SFGLIDE Creates surfaces by gliding curves along curves.
SFDRAG Creates surfaces by dragging curves along curves.
SFRELOC Relocates surfaces by translation and/or rotation.
SFMOVE Moves surfaces from one coordinate system to another.
SFRESIZ Resizes a pattern of existing surfaces.
SFFLIP Flips surfaces about a plane.
SFGEN Generates a pattern of surfaces from existing ones.
SFCOPY Copies surfaces from one coordinate system to another.
SFSCALE Generates surfaces from existing ones with a scale factor.
SFSYM Creates surfaces by symmetry about a plane.

SFEDIT Sub-submenu

SFIDENT Highlights a surface and displays its label.
SFCOMPRESS Compresses surfaces by removing numbering gaps.
SFPLT Plots surfaces on the screen.
SFLIST Lists surfaces on the screen.
SFDEL Deletes surfaces from the database.
SFUNDL Undeletes surfaces from the database.
SFERASE Erases surfaces from the screen.
VOLUMES Submenu

VL8PT  Creates a volume using 8 keypoints.
VL4CR  Creates a volume using 4 (parallel) curves.
VL2SF  Creates a volume using 2 surfaces.
VL4SF  Creates a volume using 4 surfaces.
VLPTSF Creates a volume (pyramid) using a keypoint and a surface.
VLCRSF Creates a volume (prism) using a curve and a surface.
VLGFORM Creates a volume by specifying 192 geometric parameters.

The VOLUMES menu also includes the following submenus:

VLGENR A menu to generate and manipulate volumes.
VLEDIT  A menu to edit existing volumes.

VLGENR Sub-submenu

VLEXTR  Generates volumes by extruding surfaces along an axis.
VLSWEEP Creates volumes by sweeping surfaces about an axis.
VLGLIDE Creates volumes by gliding surfaces along curves.
VLDRAG  Creates volumes by dragging surfaces along curves.
VLRELOC Relocates volumes by translation and/or rotation.
VMOVE   Moves volumes from one coordinate system to another.
VLRESIZ Resizes a pattern of existing volumes.
VLFLIP   Flips volumes about a plane.
VLGEN   Generates a pattern of volumes from existing ones.
VLCOPY  Copies volumes from one coordinate system to another.
VLSCALE Generates volumes from existing ones with a scale factor.
VLSYM   Creates volumes by symmetry about a plane.

VLEDIT Sub-submenu

VLIDENT Highlights a volume and displays its label.
VLCOMPRESS Compresses volumes by removing numbering gaps.
VLLOT    Plots volumes on the screen.
VLLIST   Lists volumes on the screen.
VLDEL    Deletes volumes from the database.
VLUNDEL  Undeletes volumes from the database.
VLERASE  Erases volumes from the screen.

CONTOURS Submenu

CT     Defines a contour with uniform element distribution.
CTNU   Defines a contour with nonuniform element distribution.
CTMODIFY Replaces one of the contour's curves by other curves.
CTTOL  Specifies contour closure tolerance.

The CONTOURS menu also includes the following submenu to edit existing contours:

CTEDIT Sub-submenu

CTIDENT Highlights a contour and displays its label.
CTPLOT Plots contours on the screen.
CTLIST Lists contours on the screen.
CTDEL Deletes contours from the database.
CTUNDEL Undeletes contours from the database.
CTERASE Erases contours from the screen.
CTCOMPRESS Compresses contours by removing numbering gaps.

REGIONS Submenu

RG Defines a region using existing contours.
RGSF Defines a region using existing surface.
RGREORNT Reorients a region to reverse the direction of the normal.

The REGIONS menu also includes the following submenus:

RGGENR A menu to generate and manipulate regions.
RGEDIT A menu to edit existing regions.

RGGENR Sub-submenu

RGRELOCT Relocates regions by translation and/or rotation.
RGMOVE Moves regions from one coordinate system to another.
RGRESIZ Resizes a pattern of existing regions.
RGFLIP Flips regions about a plane.
RGGEN Generates a pattern of regions from existing ones.
RGCOPY Copies regions from one coordinate system to another.
RGSCALE Generates regions from existing ones with a scale factor.
RGSYM Creates regions by symmetry about a plane.

RGEDIT Sub-submenu

RGIDENT Highlights a region and displays its label.
RGCOMPRESS Compresses regions by removing numbering gaps.
RGPLOT Plots regions on the screen.
RGLIST Lists regions on the screen.
RGDEL Deletes regions from the database.
RGUNDEL Undeletes regions from the database.
RGERASE Erases regions from the screen.

POLYHEDRA Submenu

PH Creates a polyhedron from surfaces and/or regions.
PHTOL Defines tolerance for closing polyhedra.
PHPLOT Plots a pattern of polyhedra on the screen.
PHLIST Lists a pattern of polyhedra.
PHDEL Deletes a pattern of polyhedra from the database.
PHEXTR Creates polyhedra by surface/region extrusion along an axis.
PHSWEEP Creates polyhedra by surface/region sweeping about an axis.
PHDRAG Creates polyhedra by surface/region dragging along a curve.
PHGLIDE Creates polyhedra by surface/region gliding along a curve.

PARTS Submenu

PART Creates a 3D geometric part entity.
PARTPLOT Plots a pattern of parts on the screen.
PARTLIST Lists a pattern of parts.
PARTDEL Deletes a pattern of parts from the database.

COORD-SYS Submenu

CSYS Defines a local coordinate system using 3 keypoints.
CSANGL Defines coordinate systems using the origin and 3 rotations.
CSMATRIX Defines coordinate systems based on a transformation matrix.
CSPLLOT Plots coordinate systems on the screen.
CSLIST Lists coordinate systems on the screen.
CSDEL Deletes coordinate systems from the database.
CSERASE Erases coordinate systems from the screen.

MESHING Menu

This menu includes the following submenus:

MESH_DENSITY A menu to set mesh density parameters.
PARAM_MESH A menu to generate parametric meshes for geometric entities.
AUTO_MESH A menu to generate automatic meshes for geometric entities.
NODES A menu to generate and edit nodes.
ELEMENTS A menu to generate and edit elements.

MESH_DENSITY Submenu

CRDENSITY Resets element size for automesh based on specified curve.
CTDENSITY Resets element size for automesh based on specified contour.
RGDENSITY Resets element size for automesh based on specified region.
PHDENSITY Resets element size for automesh based on specified polyhedron.

PARAM_MESH Submenu

M_PT Creates 1-node elements at specified keypoints.
M_CR Meshes curves to generate 1D elements.
M_SF Meshes surfaces to generate 2D elements.
M_VL Meshes volumes to generate 3D elements.
MPTDEL Deletes 1-node elements associated with keypoints.
MCRDEL Deletes 1D elements associated with curves.
MSFDEL Deletes 2D elements associated with surfaces.
MVLDEL Deletes 3D elements associated with volumes.

AUTO_MESH Submenu

MA_CR Automeshes curves to generate 2/3-node 1D elements.
MA_RG Automeshes regions to generate 3-node elements.
MA_PTRG Automeshes a region radiating from a keypoint in the region.
MA_CTRG Automeshes a region radiating from an inner contour.
### Appendix D  GEOSTAR Mode Commands

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MA_SF</td>
<td>Automeshes surfaces generating 3-node elements.</td>
</tr>
<tr>
<td>MA_NUSF</td>
<td>Automeshes a surface with different number of elements.</td>
</tr>
<tr>
<td>MA_PTSF</td>
<td>Automeshes a surface radiating from a keypoint.</td>
</tr>
<tr>
<td>MA_CRSF</td>
<td>Automeshes a surface radiating from one of its sides.</td>
</tr>
<tr>
<td>MA_PH</td>
<td>Automeshes a polyhedron using 3-node shell elements.</td>
</tr>
<tr>
<td>MA_PART</td>
<td>Automeshes parts using 4 or 10-node tetrahedral elements.</td>
</tr>
<tr>
<td>MARGCH</td>
<td>Changes region mesh from 3-node triangles to 4-node quads.</td>
</tr>
<tr>
<td>MASFCH</td>
<td>Changes surface mesh from 3-node triangles to 4-node quads.</td>
</tr>
<tr>
<td>MARGDEL</td>
<td>Deletes nodes and elements associated with a region.</td>
</tr>
<tr>
<td>MASFDEL</td>
<td>Deletes nodes and elements associated with a surface.</td>
</tr>
</tbody>
</table>

### NODES Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>ND</td>
<td>Creates a node at the specified coordinates.</td>
</tr>
<tr>
<td>NIDENT</td>
<td>Highlights a node and displays its label.</td>
</tr>
<tr>
<td>NCOMPRESS</td>
<td>Compresses nodes by removing numbering gaps.</td>
</tr>
<tr>
<td>NMODIFY</td>
<td>Modifies nodal coordinates.</td>
</tr>
<tr>
<td>NPTPUSH</td>
<td>Pushes nodes to keypoints.</td>
</tr>
<tr>
<td>NCRPUSH</td>
<td>Pushes nodes to a curve.</td>
</tr>
<tr>
<td>NSFPUSH</td>
<td>Pushes nodes to a surface.</td>
</tr>
<tr>
<td>NMERGE</td>
<td>Merges nodes within the specified tolerance.</td>
</tr>
<tr>
<td>NLIST</td>
<td>Lists nodes on the screen.</td>
</tr>
<tr>
<td>NPLIC</td>
<td>Plots nodes on the screen.</td>
</tr>
<tr>
<td>NDELETE</td>
<td>Deletes nodes from the database.</td>
</tr>
<tr>
<td>NREASSOC</td>
<td>Reassociates nodes to Surface/Region entities.</td>
</tr>
<tr>
<td>SHOW_MERGE</td>
<td>Shows probable merged nodes or free edges.</td>
</tr>
</tbody>
</table>

The NODES menu also includes the following submenu to generate nodes from existing ones:

### NDGENR Sub-submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>NPRELOC</td>
<td>Relocates nodes by translation and/or rotation.</td>
</tr>
<tr>
<td>NRESIZ</td>
<td>Relocates nodes by scaling and translating or rotating.</td>
</tr>
<tr>
<td>NMOVE</td>
<td>Moves nodes from one coordinate system to another.</td>
</tr>
<tr>
<td>NFLIP</td>
<td>Flips nodes about a plane.</td>
</tr>
<tr>
<td>NGEN</td>
<td>Generates a pattern of nodes from existing ones.</td>
</tr>
<tr>
<td>NSCALE</td>
<td>Generates nodes from existing pattern with a scale factor.</td>
</tr>
<tr>
<td>NDCOPY</td>
<td>Copies nodes from one coordinate system to another.</td>
</tr>
<tr>
<td>NSYM</td>
<td>Creates nodes by symmetry about a plane.</td>
</tr>
<tr>
<td>NPT</td>
<td>Creates a node at the location of a keypoint.</td>
</tr>
</tbody>
</table>

### ELEMENTS Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>EL</td>
<td>Defines an element by specifying its nodes.</td>
</tr>
<tr>
<td>EIDENT</td>
<td>Highlights an element and displays its label.</td>
</tr>
<tr>
<td>ECOMPRESS</td>
<td>Compresses elements by removing numbering gaps.</td>
</tr>
<tr>
<td>ELIST</td>
<td>Lists elements on the screen.</td>
</tr>
<tr>
<td>SETEPEAT</td>
<td>Sets Element plot options.</td>
</tr>
<tr>
<td>EPLET</td>
<td>Plots elements on the screen.</td>
</tr>
<tr>
<td>EDELETE</td>
<td>Deletes elements from the database.</td>
</tr>
<tr>
<td>ECHECK</td>
<td>Checks the aspect ratio of elements.</td>
</tr>
</tbody>
</table>
ACTECLR  Controls the activation of default colors for elements.
SETECLR  Sets element coloring.
ECHANGE  Changes the order of the elements.
ESMOOTH  Smoothens an existing mesh.
EREFINE  Refines selected elements in an existing mesh.
EMERGE   Merges coincident elements with identical nodal connectivity.
ALIGNSHELL  Aligns faces of shell elements with common edges.

The ELEMENTS menu also includes the following submenu to generate elements from existing ones:

**ELGENR Sub-submenu**

ELEXTR  Creates 2D or 3D elements by extruding 1D or 2D elements.
ELSWEEP Creates 2D or 3D elements by sweeping 1D or 2D elements.
ELDRAG   Creates 2D or 3D elements by dragging 1D or 2D elements.
ELGLIDE  Creates 2D or 3D elements by gliding 1D or 2D elements.
ELRELOC  Relocates elements by translation and/or rotation.
ELMOVE   Moves elements from one coordinate system to another.
ELRESIZ  Resizes a pattern of existing elements.
ELFLIP   Flips elements about a plane.
ELGEN    Creates a pattern of elements from existing ones.
ELCOPY   Copies elements from one coordinate system to another.
ELSCALE  Generates elements from existing ones with a scale factor.
ELSYM    Creates elements by symmetry about a plane.

**PROPSETS Menu**

This menu includes the following commands:

EGROUP   Defines an element group.
MPROP    Defines a material property set.
RCONST   Defines a real constant set.
PICK_MAT Picks a material from the COSMOS/M material library.
USER_MAT Picks a material from user-created material library.
_R_MATLIB Runs the material browser.
PICK_SEC Picks a section from the AISC tables.
EGLIST   Lists defined element groups on the screen.
MPLIST   Lists defined material property sets on the screen.
RCLIST   Lists defined real constant sets on the screen.
EGDEL    Deletes element groups from the database.
MPDEL    Deletes material property sets from the database.
RCDL    Deletes real constant sets from the database.
EPROPCHANGE Changes the property set association for elements.
EPROPSET Assigns attributes to elements generated from existing ones.
BMSECDEF Defines beam section dimensions.
BMSECLIST Lists defined beam sections.

**LOADS-BC Menu**

This menu includes the following submenus:
STRUCTURAL A menu for structural loadings and boundary conditions.
THERMAL A menu for thermal loadings and boundary conditions.
FLUID FLOW A menu for fluid flow loadings and boundary conditions.
E_MAGNETIC A menu for electromagnetic loadings and boundary conditions.
LOAD OPS A menu for associated load options.
FUNC_CURVES A menu to generate time, temperature, or B-H curves.

### STRUCTURAL Submenu

This menu includes the following submenus:

- **DISPLMNTS** A menu to apply displacement constraints.
- **FORCES** A menu to apply nodal force loading.
- **PRESSURE** A menu to apply element pressure loading.
- **MASTER_DOF** A menu to specify degrees of freedom for Guyan reduction.
- **COUPLING** A menu to apply coupling constraints.
- **BONDING** A menu to apply bonding constraints.
- **GRAVITY** A menu to apply gravity loading.

#### DISPLMNTS Sub-submenu

- **DND** Specifies displacements at a pattern of nodes.
- **DPT** Specifies displacements at nodes associated with keypoints.
- **DCR** Specifies displacements at nodes associated with curves.
- **DSF** Specifies displacements at nodes associated with surfaces.
- **DCT** Specifies displacements at nodes associated with contours.
- **DRG** Specifies displacements at nodes associated with regions.
- **DNDEL** Deletes displacements at a pattern of nodes.
- **DPDEL** Deletes displacements at nodes associated with keypoints.
- **DCDEL** Deletes displacements at nodes associated with curves.
- **DSDEL** Deletes displacements at nodes associated with surfaces.
- **DCTDEL** Deletes displacements at nodes associated with contours.
- **DRDEL** Deletes displacements at nodes associated with regions.
- **DPLOT** Plots prescribed nodal displacements on the screen.
- **DLIST** Lists prescribed nodal displacements on the screen.

#### FORCES Sub-submenu

- **FND** Specifies forces at a pattern of nodes.
- **FPT** Specifies forces at nodes associated with keypoints.
- **FCR** Specifies forces at nodes associated with curves.
- **FSF** Specifies forces at nodes associated with surfaces.
- **FCT** Specifies forces at nodes associated with contours.
- **FRG** Specifies forces at nodes associated with regions.
- **FNDEL** Deletes forces at a pattern of nodes.
- **FPDEL** Deletes forces at nodes associated with keypoints.
- **FCDEL** Deletes forces at nodes associated with curves.
- **FSDEL** Deletes forces at nodes associated with surfaces.
- **FCTDEL** Deletes forces at nodes associated with contours.
- **FRDEL** Deletes forces at nodes associated with regions.
- **FPLOT** Plots prescribed nodal forces on the screen.
- **FLIST** Lists prescribed nodal forces on the screen.
PRESSURE Sub-submenu

PEL Specifies pressure on elements.
PCR Specifies pressure on elements associated with curves.
PSF Specifies pressure on elements associated with surfaces.
PRG Specifies pressure on elements associated with regions.
PBEL Specifies beam element loading.
PEDEL Deletes pressure on a pattern of elements.
PCDEL Deletes pressure on elements associated with curves.
PSDEL Deletes pressure on elements associated with surfaces.
PRDEL Deletes pressure on elements associated with regions.
PBEDEL Deletes beam element loading.
PPLOT Plots prescribed pressures on the screen.
PLIST Lists prescribed pressures on the screen.
PBELIST Lists beam element loading.

MASTER_DOF Sub-submenu

MDOFND Defines master degrees of freedom at nodes for Guyan reduction.
MDOFPT Defines master degrees of freedom at nodes associated with keypoints.
MDOFCR Defines master degrees of freedom at nodes associated with curves.
MDOFSF Defines master degrees of freedom at nodes associated with surfaces.
MDOFCT Defines master degrees of freedom at nodes associated with contours.
MDOFRG Defines master degrees of freedom at nodes associated with regions.
MDOFNDEL Deletes master degrees of freedom defined at nodes.
MDOFPDEL Deletes master degrees of freedom at nodes associated with keypoints.
MDOFCDEL Deletes master degrees of freedom at nodes associated with curves.
MDOFSDEL Deletes master degrees of freedom at nodes associated with surfaces.
MDOFCTDEL Deletes master degrees of freedom at nodes associated with contours.
MDOFRDEL Deletes master degrees of freedom at nodes associated with regions.
MDOFPLOT Plots symbols at defined master degrees of freedom for Guyan reduction.
MDOFLIST Lists defined master degrees of freedom for Guyan reduction.

COUPLING Sub-submenu

CPDOF Defines a pattern of coupled degrees of freedom.
CPCNS Defines a point-to-(point, curve, or surface) constraint.
CPEQN Defines a constraint equation.
CPEQNV Defines the right hand side of a constraint equation.
CPDOFLIST Lists coupled degrees of freedom.
CPCNSLIST Lists point-to-(point, curve, or surface) constraints.
CPEQNLIST Lists constraint equations.
CPDOFDEL Deletes sets of coupled degrees of freedom.
Appendix D  GEOSTAR Mode Commands

CPCNSDEL Deletes point-to-(point, curve, or surface) constraints.
CPEQNDEL Deletes constraint equations.
CPDOFPLOT Plots a symbol for defined coupled degrees of freedom.

BONDING Sub-submenu
BONDDEF Defines Bonding of independently meshed geometric entities.
BONDLIST Lists Bonding parameters.
BONDDEL Deletes Bonding parameters.

GRAVITY Sub-submenu
ACEL Defines gravitational acceleration.
OMEGA Defines angular velocity.
DOMEGA Defines angular acceleration.
CGOMEGA Defines angular velocity of the global Cartesian system.
DCGOMEGA Defines angular acceleration of the global Cartesian system.
CGLOC Defines the origin of the inertial reference frame.
GRVLIST Lists components of gravity and centrifugal loadings.

THERMAL Submenu
This menu includes the following submenus:
TEMPERATURE A menu to apply temperature boundary conditions.
NODAL_HEAT A menu to apply nodal heat loading.
ELEM_HEAT A menu to apply element heat loading.
HEAT_FLUX A menu to apply heat flux loading.
CONVECTION A menu to apply convection loading.
RADIATION A menu to apply radiation loading.

TEMPERATURE Sub-submenu
NTND Specifies a temperature at a pattern of nodes.
NTPT Specifies a temperature at nodes associated with keypoints.
NTCR Specifies a temperature at nodes associated with curve.
NTSF Specifies a temperature at nodes associated with surfaces.
NTVL Specifies a temperature at nodes associated with volumes.
NTCT Specifies a temperature at nodes associated with contours.
NTRG Specifies a temperature at nodes associated with regions.
NTNDEL Deletes temperatures at a pattern of nodes.
NTPDEL Deletes temperatures at nodes associated with keypoints.
NTCDEL Deletes temperatures at nodes associated with curves.
NTSDEL Deletes temperatures at nodes associated with surfaces.
NTVDEL Deletes temperatures at nodes associated with volumes.
NTCTDDEL Deletes temperatures at nodes associated with contours.
NTRDDEL Deletes temperatures at nodes associated with regions.
NTPLOT Plots a symbol at nodes with specified temperatures.
NTLIST Lists prescribed nodal temperatures.

NODAL_HEAT Sub-submenu
QND Specifies heat generation rate at a pattern of nodes.
QPT Specifies heat generation rate at nodes on keypoints.
Appendix D  GEOSTAR Mode Commands

QCR  Specifies heat generation rate at nodes on curves.
QSF  Specifies heat generation rate at nodes on surfaces.
QLV  Specifies heat generation rate at nodes in volumes.
QCT  Specifies heat generation rate at nodes on contours.
QRG  Specifies heat generation rate at nodes in regions.
QNDEL Deletes heat generation rate at nodes.
QPDEL Deletes heat generation rate at nodes associated with keypoints.
QCDEL Deletes heat generation rate at nodes associated with curves.
QSDEL Deletes heat generation rate at nodes associated with surfaces.
QVDEL Deletes heat generation rate at nodes associated with volumes.
QCTDEL Deletes heat generation rate at nodes associated with contours.
QRDEL Deletes heat generation rate at nodes associated with regions.
QPLT  Plots symbols at nodes with specified heat generation rates.
QLST  Lists prescribed nodal rates of heat generation.

**ELEM_HEAT Sub-submenu**

QEL  Specifies heat generation rate for a pattern of elements.
QECCR Specifies heat generation rate for elements on a curve.
QESF Specifies heat generation rate for elements on a surface.
QEVL Specifies heat generation rate for elements in a volume.
QECT Specifies heat generation rate for elements on a contour.
QERG Specifies heat generation rate for elements in a region.
QEDEL Deletes heat generation rate for a pattern of elements.
QECDEL Deletes heat generation rate for elements on a curve.
QESDEL Deletes heat generation rate for elements on a surface.
QEVDEL Deletes heat generation rate for elements in a volume.
QECTDEL Deletes heat generation rate for elements on a contour.
QERDEL Deletes heat generation rate for elements in a region.
QPT  Plots symbols at elements with prescribed heat generation rate.
QELST Lists elements with prescribed heat generation rate.

**HEAT_FLUX Sub-submenu**

HXEL Applies heat flux on the specified face of elements.
HXCR Applies heat flux on elements associated with curves.
HXSF Applies heat flux on elements associated with surfaces.
HXRG Applies heat flux on elements associated with regions.
HEDEL Deletes heat flux for a pattern of elements.
HXCDEL Deletes heat flux for elements associated with curves.
HXSDEL Deletes heat flux for elements associated with surfaces.
HRDEL Deletes heat flux for elements associated with regions.
HXPLOT Plots symbols on elements with heat flux.
HXLST Lists elements with prescribed heat flux.

**CONVECTION Sub-submenu**

CEL Specifies convection parameters for a pattern of elements.
CECR Specifies convection parameters for elements on a curve.
CESF Specifies convection parameters for elements on a surface.
CERG Specifies convection parameters for elements in a region.
CEDEL Deletes convection parameters for a pattern of elements.
CECDEL Deletes convection parameters for elements on a curve.
CESDEL Deletes convection parameters for elements on a surface.
CERDEL Deletes convection parameters for elements in a region.
CEPLOT Plots symbols on elements with convection parameters.
CELIST Lists elements with prescribed convection parameters.

RADIATION Sub-submenu
REL Specifies radiation parameters for a pattern of elements.
RECR Specifies radiation parameters for elements on a curve.
RESF Specifies radiation parameters for elements on a surface.
RERG Specifies radiation parameters for elements in a region.
REDEL Deletes radiation parameters for a pattern of elements.
RECDEL Deletes radiation parameters for elements on a curve.
RESDEL Deletes radiation parameters for elements on a surface.
RERDEL Deletes radiation parameters for elements in a region.
REPLOT Plots symbols on elements with radiation parameters.
RELST Lists elements with radiation parameters.

FLUID_FLOW Submenu
This menu includes the following submenus:

VELOCITY A menu to specify velocity vectors.
TK_ENERGY A menu to specify turbulence kinetic energy.
DISSIP_RATE A menu to specify dissipation rate.
PRESSURE A menu to specify nodal pressure.
DENSITY A menu to specify nodal density.
BOUND_EL A menu to specify zero normal velocity.

VELOCITY Sub-submenu
VND Specifies velocity components for a pattern of nodes.
VPT Specifies velocity components at nodes associated with keypoints.
VCR Specifies velocity components at nodes associated with curves.
VSF Specifies velocity components at nodes associated with surfaces.
VVL Specifies velocity components at nodes associated with volumes.
VCT Specifies velocity components at nodes associated with contours.
VRG Specifies velocity components at nodes associated with regions.
VNDEL Deletes velocities for a pattern of nodes.
VPDEL Deletes velocities at nodes associated with keypoints.
VCDEL Deletes velocities at nodes associated with curves.
VSDEL Deletes velocities at nodes associated with surfaces.
VVDEL Deletes velocities at nodes associated with volumes.
VCTDEL Deletes velocities at nodes associated with contours.
VRDEL Deletes velocities at nodes associated with regions.
VPLT Plots symbols at nodes with prescribed velocities.
VLST Lists nodes with prescribed velocities.
TK_ENERGY Sub-submenu

TKEND Specifies turbulence kinetic energy for a pattern of nodes.
TKEPT Specifies turbulence kinetic energy at nodes on keypoints.
TKECR Specifies turbulence kinetic energy at nodes on curves.
TKESF Specifies turbulence kinetic energy at nodes on surfaces.
TKECT Specifies turbulence kinetic energy at nodes on contours.
TKERG Specifies turbulence kinetic energy at nodes in regions.
TKENDEL Deletes turbulence kinetic energy for a pattern of nodes.
TKEPDEL Deletes turbulence kinetic energy at nodes on keypoints.
TKECDEL Deletes turbulence kinetic energy at nodes on curves.
TKESDEL Deletes turbulence kinetic energy at nodes on surfaces.
TKECTDEL Deletes turbulence kinetic energy at nodes on contours.
TKERDEL Deletes turbulence kinetic energy at nodes in regions.
TKEPLOT Plots symbols at nodes with turbulence kinetic energy.
TKELIST Lists nodes with prescribed turbulence kinetic energy value.

DISSIP_RATE Sub-submenu

EPSND Specifies dissipation rate for a pattern of nodes.
EPSPT Specifies dissipation rate at nodes associated with keypoints.
EPSCR Specifies dissipation rate at nodes associated with curves.
EPSSF Specifies dissipation rate at nodes associated with surfaces.
EPSCT Specifies dissipation rate at nodes associated with contours.
EPSRG Specifies dissipation rate at nodes associated with regions.
EPSNDEL Deletes dissipation rate for a pattern of nodes.
EPSLIST Lists nodes with prescribed dissipation rates.
EPSPLOT Plots symbols at nodes with prescribed dissipation rates.
EPSPDEL Deletes dissipation rate at nodes associated with keypoints.
EPSCDEL Deletes dissipation rate at nodes associated with curves.
EPSSDEL Deletes dissipation rate at nodes associated with surfaces.
EPSCTDEL Deletes dissipation rate at nodes associated with contours.
EPSRDEL Deletes dissipation rate at nodes associated with regions.

PRESSURE Sub-submenu

NPRNND Specifies nodal pressure for a pattern of nodes.
NPRPT Specifies nodal pressure at nodes on keypoints.
NPRCR Specifies nodal pressure at nodes associated with curves.
NPRCT Specifies nodal pressure at nodes associated with contours.
NPRNDEL Deletes nodal pressure for a pattern of nodes.
NPRPDEL Deletes nodal pressure at nodes associated with keypoints.
NPRCDEL Deletes nodal pressure at nodes associated with curves.
NPRCTDEL Deletes nodal pressure at nodes associated with contours.
NPRPLOT Plots symbols at nodes with prescribed nodal pressures.
NPRLIST Lists nodes with prescribed nodal pressures.

DENSITY Sub-submenu

DNSND Specifies density at a pattern of nodes.
DNSPT Specifies density at nodes associated with keypoints.
DNSCR Specifies density at nodes associated with curves.
DNSSF Specifies density at nodes associated with surfaces.
DNSCT Specifies density at nodes associated with contours.
DNSRG Specifies density at nodes associated with regions.
Appendix D   GEOSTAR Mode Commands

DNSNDEL Deletes density value at a pattern of nodes.
DNSPDEL Deletes density value at nodes associated with keypoints.
DNSCDEL Deletes density value at nodes associated with curves.
DNSSDEL Deletes density value at nodes associated with surfaces.
DNSCTDEL Deletes density value at nodes associated with contours.
DNSRDEL Deletes density value at nodes associated with regions.
DNSPLOT Plots symbols at nodes with prescribed densities.
DNSLIST Lists nodes with prescribed densities.

BOUND_EL Sub-submenu

BEL Specifies zero normal velocity for a pattern of elements.
BECR Specifies zero normal velocity for elements on curves.
BESF Specifies zero normal velocity for elements on surfaces.
BERG Specifies zero normal velocity for elements in regions.
BEDE Deletes zero normal velocity for a pattern of elements.
BECDEL Deletes zero normal velocity for elements on curves.
BESDEL Deletes zero normal velocity for elements on surfaces.
BERDEL Deletes zero normal velocity for elements in regions.
BEPLOT Plots symbols on elements with zero normal velocity.
BELIST Lists elements with zero normal velocity condition.

E_MAGNETIC Submenu

JSDEF Defines 3D current sources for magnetostatic analysis.
JSDEL Deletes 3D current sources.
JSLIST Lists prescribed 3D current sources.
MCPDEF Couples the magnetic potential at nodes in two patterns.
MCDEL Deletes magnetic coupling at a pattern of nodes.
MCPLIST Lists magnetic coupling at a pattern of nodes.

The E_MAGNETIC menu also includes the following submenus:

M_POTENTIAL A menu to specify voltage or magnetic potential.
ND_CURRENT A menu to define nodal currents.
EL_CURRENT A menu to specify currents or charges at elements.
HF_CB A menu for high frequency boundary conditions.

M_POTENTIAL Sub-submenu

NPND Specifies voltage/magnetic potentials at nodes.
NPPT Specifies voltage/magnetic potentials at nodes on keypoints.
NPCR Specifies voltage/magnetic potentials at nodes on curves.
NPSF Specifies voltage/magnetic potentials at nodes on surfaces.
NPCR Specifies voltage/magnetic potentials at nodes on contours.
NPRG Specifies voltage/magnetic potentials at nodes in regions.
NPNDE Deletes voltage/magnetic potentials at nodes.
NPPDEL Deletes voltage/magnetic potentials at nodes on keypoints.
NPCDEL Deletes voltage/magnetic potentials at nodes on curves.
NPSDEL Deletes voltage/magnetic potentials at nodes on surfaces.
NPCCTDEL Deletes voltage/magnetic potentials at nodes on contours.
NPREDL Deletes voltage/magnetic potentials at nodes in regions.
NPPLT Plots symbols at nodes with voltage or magnetic potentials.
NPLIST Lists nodes with prescribed voltage or magnetic potentials.
Appendix D   GEOSTAR Mode Commands

**ND_CURRENT Sub-submenu**

- **NJIND** Specifies input current at a pattern of nodes.
- **NJPT** Specifies input current at nodes associated with keypoints.
- **NJCR** Specifies input current at nodes associated with curves.
- **NJSF** Specifies input current at nodes associated with surfaces.
- **NJCT** Specifies input current at nodes associated with contours.
- **NJRG** Specifies input current at nodes associated with regions.
- **NJDDEL** Deletes input current at a pattern of nodes.
- **NJPDEL** Deletes input current at nodes associated with keypoints.
- **NJCDEL** Deletes input current at nodes associated with curves.
- **NJSDEL** Deletes input current at nodes associated with surfaces.
- **NJCTDEL** Deletes input current at nodes associated with contours.
- **NJRDDEL** Deletes input current at nodes associated with regions.
- **NJPLOT** Plots symbols at nodes with prescribed currents.
- **NJLIST** Lists nodes with prescribed currents.

**EL_CURRENT Sub-submenu**

- **JEL** Specifies current density for a pattern of elements.
- **JESF** Specifies current density at elements on surfaces.
- **JERG** Specifies current density at elements in regions.
- **JEDEL** Deletes current density for a pattern of elements.
- **JESDEL** Deletes current density for elements on surfaces.
- **JERDEL** Deletes current density for elements in regions.
- **JEPLOT** Plots symbols on elements with current densities.
- **JELIST** Lists elements with prescribed current densities.

**HF_CB Sub-submenu**

- **CBEL** Defines an HF boundary condition on element faces.
- **CBCR** Defines an HF boundary condition on curves.
- **CBSF** Defines an HF boundary condition on surfaces.
- **CBRG** Defines an HF boundary condition on regions.
- **CBEDEL** Deletes HF boundary conditions on element faces.
- **CBCDEL** Deletes HF boundary conditions on curves.
- **CBSDEL** Deletes HF boundary conditions on surfaces.
- **CBRDEL** Deletes HF boundary conditions on regions.
- **CBPLOT** Plots HF boundary conditions.
- **CBLIST** Lists HF boundary conditions.

**LOAD_OPS Submenu**

- **TREF** Defines the reference temperature for the model.
- **TUNIF** Defines a uniform temperature for all nodes in the model.
- **TIMES** Specifies time parameters for nonlinear and transient analyses.
- **TIMELIST** Lists time parameters for nonlinear and transient analysis.
- **INITIAL** Defines initial conditions at a pattern of nodes.
- **INITDEL** Deletes specified initial conditions at a pattern of nodes.
- **INITLIST** Lists specified initial conditions at a pattern of nodes.
- **NPRREAD** Provides fluid-structural coupling.
- **EMFREAD** Provides magneto-structural coupling.
- **TEMPREAD** Provides thermal-structural coupling.
Appendix D  GEOSTAR Mode Commands

TEMPRDLIST  Lists time step/load case assignments for thermal loading.
TEMPINIT   Provides as initial temperature, results from a previous run.

**FUNC_CURVES Submenu**

CURDEF     Defines (time, temperature, or B-H) curves.
MPC        Defines a strain-stress curve at a reference temperature.
MPCTYP     Sets the type of material property curves.
CURDEL     Deletes a pattern of previously defined curves.
MPCDEL     Deletes a pattern of material property curves.
CURLIST    Lists previously defined curves.
MPCLIST    Lists material property curves.
MAKE_CYCLIC Repeats a pattern of (time, temperature, or B-H) curves.

**CONTROL Menu**

EXIT       Saves the generated model and exits to the operating system.

The CONTROL menu also includes the following submenus:

UTILITY    A menu to general utility commands.
ACTIVE     A menu to control activation of sets.
SELECT     A menu to create and edit selection lists.
PARAMETRIC A menu to deal with symbolic variables and expressions.
CAD_SYS    A menu to translate to or from IGES and DXF files.
FEM_INP    A menu to translate to or from ANSYS/NASTRAN/PATRAN files.
DEVICES    A menu to hardcopies and (PostScript, META, DXF, ...) files.
MEASURE    A menu to geometry and mesh measurements.
MISC       A menu to miscellaneous commands.

**UTILITY Submenu**

FILE       Reads commands from a file.
EDIT       Transfers control from GEOSTAR to the editor.
SAVE       Updates the session file and copies database to a new name.
HELP       Provides description of the command.
STATUS1    Information on flags associated with geometric entities.
STATUS2    Information on plotting of boundary conditions, forces, etc.
CMDLIST    Lists commands in the session file.
NEWPROB    Switches to a new problem.
GFORM_OUT  Generates a compact file similar to the session file.
SYMSIZ     Sets symbol size for plotting various entities.
OSCOMMAND  Executes operating system commands.
SYSTEM     Enables using the operating system without exiting GEOSTAR.

**ACTIVE Submenu**

ACTSET     Activates a particular set.
ACTPLOT    Controls the plotting of lower order entities.
ACTMARK    Controls plotting of marks to identify parametric coordinates.
ACTNUM
ACTKEEP
ACTCDMESH
ACTSEL

ACTNUM Controls the plotting of entity numbers.
ACTKEEP Controls keeping lower entities when deleting higher ones.
ACTCDMESH Controls default meshing of newly created entities.
ACTSEL Controls the activation of selection lists.

SELECT Submenu

INITSEL Status3
SELSETOP
SELINP
SELPIC
SELWIN
SELREF
SELREF
SELREF
SELREF

INITSEL Initializes a selection list.
STATUS3 Information on selection lists.
SELSETOP Performs Selection set operations.
SELINP Adds members to a selection list by specifying their labels.
SELPIC Adds members to a selection list by picking.
SELWIN Adds members to a selection list using a window.
SELREF Selects members associated with a reference entity.
SELREF Adds members to a selection list by specifying x, y, z ranges.
ESELPROP Selects elements associated with specified property set.
UNSELINP Removes members from a selection list using their labels.
UNSELPIC Removes members from a selection list by picking.
UNSELWIN Removes members from a selection list using a window.
UNSELREF Removes entity-associated members from a selection list.
UNSELREF Removes members from a selection list using x, y, z ranges.
UNESELPROP Removes property set-associated elements from selection.

PARAMETRIC Submenu

PARASSIGN
PARLIST
PARDEL
ARRDEF
ARRASSIGN
ARRLIST
ARRDEL
FUNCDEF
FUNCLIST
FUNCDEL
CALLMACRO

PARASSIGN Assigns a numeric value to a parameter.
PARLIST Lists defined parameters and their numeric values.
PARDEL Deletes parameters from the database.
ARRDEF Declares an array.
ARRASSIGN Assigns value to array elements.
ARRLIST Lists arrays.
ARRDEL Deletes arrays.
FUNCDEF Defines analytical functions.
FUNCLIST Lists defined functions.
FUNCDEL Deletes functions from the database.
CALLMACRO Calls a pre-defined macro with corresponding input.

CAD_SYS Submenu

CAD_INP SAT2 IGS
IGES_INP IGES_OUT
DXF_INP DXF_OUT
PRO_INP

CAD_INP Reads solid and shell-type geometry from CAD systems.
SAT2 IGS Creates an IGES file from a SAT file which includes ACIS-based geometry.
IGES_INP Reads and executes an IGES file.
IGES_OUT Generates an IGES file for the geometry of problem.
DXF_INP Reads and executes a DXF file.
DXF_OUT Generates a DXF file for the geometry of problem.
PRO_INP Reads and executes Pro/ENGINEER - COSMOS/M geometry file.
Appendix D  GEOSTAR Mode Commands

FEM_INP Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MODINPUT</td>
<td>Generates an input file for MODSTAR module.</td>
</tr>
<tr>
<td>ANSYSINP</td>
<td>Generates an input file for ANSYS program.</td>
</tr>
<tr>
<td>NASTRANINP</td>
<td>Generates an input file for NASTRAN program.</td>
</tr>
<tr>
<td>ABAQUSINP</td>
<td>Generates an input file for ABAQUS program.</td>
</tr>
<tr>
<td>PATRANINP</td>
<td>Generates an input file for PATRAN program.</td>
</tr>
<tr>
<td>SINDAINP</td>
<td>Generates an input file for SINDA program.</td>
</tr>
<tr>
<td>TEAPINP</td>
<td>Generates an input file for TEAP program.</td>
</tr>
<tr>
<td>R_MDITRANS</td>
<td>Two-way translation between COSMOS/M and ADAMS.</td>
</tr>
</tbody>
</table>

DEVICES Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PAPER_SETUP</td>
<td>Sets paper size, orientation and margins for hardcopies.</td>
</tr>
<tr>
<td>PAINTJET</td>
<td>Generates a hardcopy using an HP_PAINTJET printer.</td>
</tr>
<tr>
<td>LASERJET</td>
<td>Generates a hardcopy using an HP_LASERJET printer.</td>
</tr>
<tr>
<td>DESKJET</td>
<td>Generates a hardcopy using an HP_DESKJET printer.</td>
</tr>
<tr>
<td>HPGL_CRTRG</td>
<td>Generates a hardcopy using an HPGL cartridge.</td>
</tr>
<tr>
<td>NEC201</td>
<td>Generates a hardcopy using an NEC201 printer (NEC Version).</td>
</tr>
<tr>
<td>SCREENPLOT</td>
<td>Dumps the screen to a printer or plotter.</td>
</tr>
</tbody>
</table>

DEVICE_FILE Sub-submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>METAFILE</td>
<td>Saves an image in an ASCII META file.</td>
</tr>
<tr>
<td>HPGLFILE</td>
<td>Saves an image in an ASCII HPGL file.</td>
</tr>
<tr>
<td>POSTSCRIPT</td>
<td>Saves an image in the PostScript format.</td>
</tr>
<tr>
<td>DXF_2D</td>
<td>Saves an image in the DXF_2D format used by CAD packages.</td>
</tr>
<tr>
<td>VIEW_META</td>
<td>Views an existing meta image file on the screen.</td>
</tr>
<tr>
<td>PLT_METAFIL</td>
<td>Generates a hardcopy from an ASCII META file.</td>
</tr>
<tr>
<td>PLT_FILE</td>
<td>Generates a hardcopy from an ASCII HPGL or PostScript file.</td>
</tr>
<tr>
<td>IMAGESAV</td>
<td>Saves a selected part of the screen into an image file.</td>
</tr>
<tr>
<td>IMAGERES</td>
<td>Restores an image file on the screen.</td>
</tr>
<tr>
<td>MULTIPRINT</td>
<td>Prints previously created image files in various formats.</td>
</tr>
</tbody>
</table>

MEASURE Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MASSPROP</td>
<td>Lists mass property information for a pattern of elements.</td>
</tr>
<tr>
<td>DISTANCE</td>
<td>Computes distance between keypoints and/or nodes.</td>
</tr>
<tr>
<td>ANGLE</td>
<td>Computes angle between 3 keypoints, 3 nodes or 2 lines.</td>
</tr>
<tr>
<td>LENGTH</td>
<td>Computes length of a specified curve.</td>
</tr>
<tr>
<td>AREA</td>
<td>Computes area of a specified surface.</td>
</tr>
</tbody>
</table>

MISC Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>MENUTYPE</td>
<td>Specifies the mode of menu access and brief help.</td>
</tr>
<tr>
<td>WRTEXT</td>
<td>Writes text on the screen.</td>
</tr>
<tr>
<td>WRTXTLIST</td>
<td>Lists messages input by WRTEXT command.</td>
</tr>
<tr>
<td>ERASE</td>
<td>Erases whatever is inside of a window from the screen.</td>
</tr>
<tr>
<td>TITLE</td>
<td>Gives a title to the current problem.</td>
</tr>
<tr>
<td>SUBTITLE</td>
<td>Gives a subtitle to the current problem.</td>
</tr>
<tr>
<td>COMMENT</td>
<td>Writes a comment to the session file.</td>
</tr>
</tbody>
</table>
### DISPLAY Menu

This menu includes the following submenus:

- **VIEW_PAR** A menu to control the view parameters.
- **DISP_PAR** A menu to control the display parameters.
- **WINDOWS** A menu to create and use up to 4 windows.
- **XY_PLOTS** A menu to plot pre- and postprocessing graphs.

#### VIEW_PAR Submenu

- **VIEW** Defines the view direction.
- **VIEWSAVE** Saves current viewing direction.
- **VIEWREST** Restores specified viewing direction.
- **AXIS** Controls plotting of the global coordinate system axes.
- **ASPECT** Defines a ratio for the x- and y- scales.
- **EXTENTS** Clips an object beyond a specified range.
- **REPAINT** Replots the picture using the current view parameters.
- **RESET** Resets view parameters to their default values.
- **CLS** Clears the screen.
- **FILTER** Filters plots to be exclusively ored with a specified color.
- **SETCOLOR** Sets colors for various entities.
- **FCLR** Sets the foreground color of the window.
- **BCLR** Sets the background color of the window.

#### DISP_PAR Submenu

- **TRANSLATE** Translates the current picture by the specified distance.
- **ROTATE** Rotates the current picture by the specified angles.
- **SCALE** Replots the current picture after scaling it.
- **PSCALE** Replots picture after scaling it to the window size.
- **ZOOMIN** Zooms into a part of the current picture.
- **ZOOMOUT** Recovers previously zoomed windows.
- **SHRINK** Shrinks elements by a specified factor for plotting.
- **HIDE_OPT** Sets element hidden face removal options.
- **HIDDEN** Controls plotting of hidden elements.
- **SHADE** Activates or deactivates shading of element plots.
- **LIGHT** Specifies the location of light source for shaded elements.
- **EVAL_BOUND** Evaluates boundary element faces or edges.
- **BOUNDARY** Controls plotting of element boundaries and faces.

#### WINDOWS Submenu

- **WCREATE** Creates up to 4 new windows.
- **WOPEN** Opens a closed window.
- **WMOVE** Moves an open window.
WRESIZE       Resizes an open window.
WPUSH         Pushes a window to the background.
WPOP          Activates a window and pops it to the foreground.
WCLOSE        Closes an open window.
WDELETE       Deletes a window.

**XY_PLOTS Submenu**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>INITXYPLOT</td>
<td>Restores default setting for X-Y plots.</td>
</tr>
<tr>
<td>ACTXYPRE</td>
<td>Loads preprocessing curves for X-Y plots.</td>
</tr>
<tr>
<td>ACTXYPOST</td>
<td>Loads postprocessing curves for X-Y plots.</td>
</tr>
<tr>
<td>ACTXYUSR</td>
<td>Loads user-defined curves for Y-Y plots.</td>
</tr>
<tr>
<td>SETXYPLOT</td>
<td>Sets flags and parameters for X-Y plots.</td>
</tr>
<tr>
<td>XYRANGE</td>
<td>Specifies the graph ranges and scale factors.</td>
</tr>
<tr>
<td>XYREFLINE</td>
<td>Plots a reference line parallel to the X or Y axis.</td>
</tr>
<tr>
<td>XYIDENTIFY</td>
<td>Identifies the coordinates of a point in the graph range.</td>
</tr>
<tr>
<td>XYLIST</td>
<td>Lists the available information to produce X-Y plots.</td>
</tr>
<tr>
<td>XYPTLIST</td>
<td>Lists a pattern of points for loaded graphs.</td>
</tr>
<tr>
<td>XYPLOT</td>
<td>Plots the activated X-Y graph curve.</td>
</tr>
</tbody>
</table>

**ANALYSIS Menu**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>RESTART</td>
<td>Sets restart flag for different analysis options.</td>
</tr>
<tr>
<td>RENUMBER</td>
<td>Minimizes bandwidth of the stiffness matrix.</td>
</tr>
<tr>
<td>DATA_CHECK</td>
<td>Checks element groups, material and real constant sets, etc.</td>
</tr>
<tr>
<td>R_CHECK</td>
<td>Performs analysis related database checks.</td>
</tr>
<tr>
<td>A_LIST</td>
<td>Lists active options for various types of analyses.</td>
</tr>
</tbody>
</table>

The ANALYSIS menu also includes the following submenus:

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>OUTPUT_OPS</td>
<td>A menu to control writing output results in the (.OUT) file.</td>
</tr>
<tr>
<td>STATIC</td>
<td>A menu for Linear Static analysis.</td>
</tr>
<tr>
<td>FREQ/BUCK</td>
<td>A menu for Frequency and Buckling analysis.</td>
</tr>
<tr>
<td>POST_DYN</td>
<td>A menu for Post-Dynamic analysis.</td>
</tr>
<tr>
<td>NONLINEAR</td>
<td>A menu for Nonlinear Static and Dynamic Structural analyses.</td>
</tr>
<tr>
<td>OPTM/SENS</td>
<td>A menu for Design Optimization and Sensitivity.</td>
</tr>
<tr>
<td>FATIGUE</td>
<td>A menu for Fatigue analysis.</td>
</tr>
<tr>
<td>HEAT_TRANS</td>
<td>A menu for Heat Transfer analysis.</td>
</tr>
<tr>
<td>FLUID_MECH</td>
<td>A menu for Fluid Mechanics analysis.</td>
</tr>
<tr>
<td>ELEC_MAGNET</td>
<td>A menu for Low-Frequency Electromagnetic analysis.</td>
</tr>
<tr>
<td>HF_EMAG</td>
<td>A menu for High-Frequency Electromagnetics analysis.</td>
</tr>
</tbody>
</table>

**OUTPUT_OPS Submenu**

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>PRINT_OPS</td>
<td>Controls writing of results to the output file.</td>
</tr>
<tr>
<td>PRINT_NDSET</td>
<td>Defines groups of nodes for results evaluation.</td>
</tr>
<tr>
<td>PRINT_ELSET</td>
<td>Defines groups of elements for stress evaluation.</td>
</tr>
<tr>
<td>STRAIN_OUT</td>
<td>Controls writing of strain components in the output file.</td>
</tr>
</tbody>
</table>
**STATIC Submenu**

- **LCSET** Activates load cases for static analysis.
- **LCLIST** Lists defined load cases.
- **ADAPTIVE** Selects (H, P, or HP) methods for static analysis.
- **PORERLABS** Sets a flag to plot the P-order.
- **A_STATIC** Specifies analysis options for the linear static module.
- **A_FFESTATIC** Specifies analysis options for the FFE static module.
- **AASYM** Specifies asymmetric loading and mode options.
- **A_STRESS** Specifies analysis options for the stress module.
- **STRESS** Activates stress calculations in the static module.
- **SUBMODEL** Defines a submodel in chosen areas.
- **REACTION** Flag for calculations of reaction forces and moments.
- **R_STATIC** Performs linear static analysis.
- **R_STRESS** Performs stress analysis for linear static problems.

The STATIC menu also includes the following submenus:

- **SUBSTRUCTUR** A menu for substructuring.
- **CRACK** A menu for crack definition.
- **ASME_CODE** A menu for the ASME code check.

**SUBSTRUCTUR Sub-submenu**

- **SPSTR** Creates super elements and defines type of analysis.
- **SPELE** Defines connectivity for a super element.
- **SPROT** Rotates a super element in the global Cartesian system.
- **SPLIST** Lists super elements and their super nodes.
- **SPDEL** Deletes super elements from the database.

**CRACK Sub-submenu**

- **CRACK** Defines a crack set and specifies the corresponding nodes.
- **CRACKLIST** Lists a pattern of crack sets and the associated nodes.
- **CRACKDEL** Deletes a pattern of crack sets.

**ASME_CODE Sub-submenu**

- **ASMESECDEF** Defines a section for use with ASME_CODE check.
- **ASMESECLIST** Lists defined ASME_CODE sections.
- **ASMESECDEL** Deletes ASME_CODE sections.

**FREQ/BUCK Submenu**

- **A_FREQUENCY** Specifies analysis options for frequency calculation.
- **A_BUCKLING** Specifies analysis options for buckling load calculation.
- **R_FREQUENCY** Performs natural frequency and mode shape analysis.
- **R_BUCKLING** Performs buckling analysis.

**POST_DYN Submenu**

- **PD_ATYPE** Specifies the type of post-dynamic analysis.
- **PD_ALIST** Lists information defined for post-dynamic analysis.
R_DYNAMIC Performs post-dynamic analysis.
PD_PREPARE Prepares plotting files.

The POST_DYN menu also includes the following submenus:

PD_DAMP/GAP A menu for damping and gap elements.
PD_CURVES A menu for post-dynamic curves.
PD_BEXCIT A menu for uniform and multiple base motion.
PD_OUTPUT A menu for output of post-dynamic analysis.

PD_DAMP/GAP Sub-submenu

PD_RDAMP Defines the coefficients of Rayleigh damping.
PD_MDAMP Defines modal damping.
PD_DAMPREAD Reads calculated material damping.
PD_DAMPLIST Lists the available damping information.
PD_CDAMP Defines concentrated dampers.
PD_CDDEL Deletes a pattern of concentrated dampers.
PD_CDLIST Lists a pattern of concentrated dampers.
PD_GAP Defines gap elements.
PD_GAPDEL Deletes a pattern of gap elements.

PD_CURVES Sub-submenu

PD_CURTYP Defines the curve type for Post-Dynamic analysis.
PD_CURDEF Defines a curve for Post-Dynamic analysis.
PD_CURDEL Deletes a pattern of curves in Post-Dynamic analysis.
PD_CURLIST Lists Post-Dynamic curves and harmonic functions.

PD_BEXCIT Sub-submenu

PD_BASE Defines input data for base excitation.
PD_BSLIST Lists defined base excitations.
PD_SPPRT Defines a support excitation level.
PD_SPPRTLIS Lists support excitation levels.
PD_SPPRTDEL Deletes support excitation levels.

PD_OUTPUT Sub-submenu

PD_PRINT Defines response printout options.
PD_PLOT Specifies postprocessing plot options.
PD_PLTINT Specifies plotting resolution.
PD_NRESP Specifies nodal response graph options.
PD_RELRESP Requests the calculation of relative nodal responses.
PD_SXYSET Requests element stresses, forces and moments for graph display.
PD_PLTLIST Lists all information related to Post-Dynamic plots.
PD_MAXMIN Finds nodes with extreme response in Post-Dynamic analysis.
PD_MAXLIST Lists the latest obtained extreme nodal responses.

NONLINEAR Submenu

NL_CONTROL Specifies the numerical procedure for nonlinear analysis.
NL_INTGR Specifies integration parameters for dynamic analysis.
NL_AUTOSTEP  Specifies automatic stepping option.
NL_BASE  Specifies base motion parameters.
NL_RDAMP  Specifies Raleigh damping coefficients.
NL_PRINT  Specifies print options.
NL_PLOT  Specifies postprocessing plot options.
NL_NRESP  Specifies nodal response graph options.
A_NONLINEAR  Specifies details for the nonlinear analysis.
R_NONLINEAR  Performs nonlinear static and dynamic structural analyses.

The NONLINEAR menu also includes the following submenus:

CONTACT  A menu for contact lines or surfaces.
WATERTABLE  A menu for water motion tables.
J_INTEGRAL  A menu for J integral options.

CONTACT Sub-submenu

NL_GS  Defines contact lines or surfaces for Nonlinear analysis.
NL_GSAUTO  Defines contact lines or surfaces from geometry.
NL_GSLIST  Lists contact lines and surfaces for Nonlinear analysis.
NL_GSPLLOT  Plots contact lines and surfaces and their orientation vectors.
NL_GSDEL  Deletes contact lines and surfaces for Nonlinear analysis.
NL_GSINTERF  Sets flag for initial interference processing of gaps.

WATERTABLE Sub-submenu

WMTDEF  Defines Water-Motion-Tables associated with IMPIPE elements.
WMTDEL  Lists a pattern of previously defined Water-Motion-Tables.
WMTLIST  Deletes a pattern of previously defined Water-Motion-Tables.

J_INTEGRAL Sub-submenu

J_INTDEF  Defines a J_integral path around a crack tip.
J_INTLIST  Lists J_integral paths for nonlinear analysis.
J_INTDEL  Deletes J_integral paths for nonlinear analysis.
J_INTPLOT  Plots J_integral paths for nonlinear analysis.
J_INTINF  Defines additional options for the J_integral.
J_INTELEM  Defines J_integral elements for axisymmetric and thermal analyses.

OPTM/SENS Submenu

This menu includes the following submenus:

DESIGN_VARS  Design variables menu for sensitivity and optimization.
OPTM_OBJ  Objective function menu for optimization.
OPTM_CON  Behavior constraints menu for optimization.
SENS_RESP  Response quantities menu for sensitivity analysis.
OPTM_LOOPS  A menu to control optimization loops.
SENS_RUNS  A menu to control sensitivity study.
DESIGN_VARS Sub-submenu

DVARDEF Defines design optimization variables.
DVARLIST Lists design optimization variables.
DVARDEL Deletes design optimization variables.
OP_DVMOVE Specifies design variable move limits.
SN_SETDEF Defines design variables sets for sensitivity.
SN_SETLIST Lists design variables sets for sensitivity.
SN_SETDEL Deletes design variables sets for sensitivity.

OPTM_OBJ Sub-submenu

OP_OBJDEF Defines objective function.
OP_OBJSET Defines objective function sets.
OP_OBJLIST Lists objective function.
OP_OBJDEL Deletes objective function sets.

OPTM_CON Sub-submenu

OP_CONDEF Defines behavior constraints.
OP_CONLIST Lists behavior constraints.
OP_CONDEL Deletes behavior constraints.
OP_CONTRIM Specifies truncation factors for constraints.

SENS_RESP Sub-submenu

SN_RESPDEF Defines response quantities for sensitivity.
SN_RESPLIST Lists response quantities for sensitivity.
SN_RESPDEL Deletes response quantities for sensitivity.

OPTM_LOOPS Sub-submenu

OP_CONTROL Specifies optimizer control parameters.
OP_RESTORE Restores a specified design set.
A_OPTIMIZE Specifies analysis options for optimization loops.
R_OPTIMIZE Performs design optimization.

SENS_RUNS Sub-submenu

A_SENSITIV Defines sensitivity study options.
R_SENSITIV Runs sensitivity analysis.

FATIGUE Submenu

FT_EVENT Specifies the number of cycles for a fatigue event.
FT_LOAD Defines fatigue loading.
FT_STREAD Defines stress conditions directly.
FT_CURDEF Defines fatigue property curves.
FT_LOC Defines a fatigue location by specifying a node label.
FT_SEC Defines a section for simplified Elastic-Plastic analysis.
A_FATIGUE Specifies SN curve type, element face and layer numbers.
R_FATIGUE Performs fatigue analysis.

The FATIGUE menu also includes the following submenus:
**FATIGUE_LIS**  A menu to list fatigue related specifications.

**FATIGUE_DEL**  A menu to delete fatigue related specifications.

**FATIGUE_LIS Sub-submenu**
- **FT_EVENTLIS**  Lists fatigue events and their specifications.
- **FT_STLIST**  Lists stress conditions for a pattern of fatigue locations.
- **FT_CURLIST**  Lists defined fatigue property curves.
- **FT_LOCLIST**  Lists information for a pattern of defined fatigue locations.
- **FT_SECLIST**  Lists information for a pattern of defined fatigue sections.

**FATIGUE_DEL Sub-submenu**
- **FT_EVENTDEL**  Deletes a pattern of fatigue events.
- **FT_LOADDEL**  Deletes a pattern of fatigue loadings.
- **FT_STDEL**  Deletes stresses associated with fatigue locations.
- **FT_CURDEL**  Deletes defined fatigue property curves.
- **FT_LOCDEL**  Deletes a pattern of fatigue locations.
- **FT_SECDEL**  Deletes a pattern of fatigue sections.

**HEAT_TRANS Submenu**
- **TOFFSET**  Specifies the temperature offset.
- **SB_CONST**  Specifies the Stefan Boltzmann constant.
- **AUTOSTEP**  Optimizes time step for phase change in thermal analysis.
- **RVFTYP**  Specifies the type of radiation source entity.
- **RVFDEF**  Specifies a source entity and a pattern of target entities.
- **RVFDEL**  Deletes view factors for a pattern of source entities.
- **RVFLIST**  Lists the requested view factors for source entities.
- **HT_SOLN**  Specifies thermal solution options.
- **HT_OUTPUT**  Specifies thermal output options.
- **A_THERMAL**  Specifies the analysis options for thermal analysis using HSTAR.
- **A_FFETHERMAL**  Specifies analysis options for the FFE thermal module.
- **R_THERMAL**  Performs thermal analysis.

**FLUID_Mech Submenu**
- **FL_MODEL**  Specifies the type of fluid (Newtonian or non-Newtonian).
- **FL_METH**  Specifies the type of the theoretical formulation.
- **FL_SOLN**  Specifies parameters related to the solution procedures.
- **FL_INTGR**  Specifies parameters for compressible fluid flow analysis.
- **FL_SCALES**  Specifies scaling factors for non-dimensionalization.
- **FL_OUTPUT**  Controls intervals at which FL output is printed and plotted.
- **A_FLOW**  Specifies the details of the fluid flow analysis.
- **R_FLOW**  Performs fluid flow analysis.

**ELEC_MAGNET Submenu (Low-Frequency Electromagnetics)**
- **EM_FREQRANGE**  Specifies the range of frequency for magnetodynamic analysis.
- **EM_OUTPUT**  Controls intervals at which EM output are printed and plotted.
- **EM_MODEL**  Specifies stationary or uniformly moving conductors.
A_MAGNETIC Specifies the details of the electromagnetic analysis.
R_MAGNETIC Performs electromagnetic analysis.

HF_EMAG Submenu (High-Frequency Electromagnetics)

A_HFRQEM Sets options for high-frequency analysis.
R_HFRQEM Runs high frequency electromagnetic analysis.

The HF_EMAG menu also includes the following submenus:

- CROSS_TALK A menu related to cross-talk analysis.
- INT_PATH A menu to define and list integration paths.
- TRANS_LINE A menu for transmission line.
- CAVITIES A menu for cavities.

CROSS_TALK Sub-submenu (Cross-Talk Electromagnetic Analysis)

- HF_XTKCONF Defines parameters for the X-talk time domain simulator.
- HF_XTKPULS Defines excitation pulses on the near-end of chosen lines.
- HF_XTKTERM Defines the terminations at ends of lines.
- HF_XTKLIST Lists x-talk information for a line.

INT_PATH Sub-submenu (Integration Paths for High-Frequency Electromagnetics)

- HF_PATH Defines integration paths for 2D field simulator and cavities.
- HF_PATHLIST Lists an integration path defined by HF_PATH.
- HF_PATHDEL Deletes a pattern of integration paths defined by HF_PATH.

TRANS_LINE Sub-submenu (Electromagnetic Transmission Line Analysis)

- HF_2DSOLN Sets solution options for the 2D field solver.
- HF_2DOUT Sets output options for the 2D field solver.

CAVITIES Sub-submenu (High-Frequency Electromagnetics Analysis of Cavities)

- HF_CAVSOLN Sets solution options for the cavity simulator.
- HF_CAVOUT Sets output options for the cavity simulator.

RESULTS Menu

- LCCOMB Combines the response of desired load cases.
- AVERAGE Specifies the procedure to calculate average nodal stresses.
- RESULTS? Lists available results.
- READ_PDRESP Reads Post-Dynamic response into secondary load case array.

The RESULTS menu also includes the following submenus:

- ACTIVATE A menu to activate the analysis type for postprocessing.
- PLOT A menu to plot the desired analysis results.
- LIST A menu to list analysis results.
- EXTREMES A menu to list extreme values.
**ACTIVATE Submenu**

- **ACTPOST** Specifies type of analysis for postprocessing.
- **ACTSTR** Loads the specified stress component into the plot buffer.
- **ACTSTN** Loads the specified strain component into the plot buffer.
- **ACTDIS** Loads specified displacement component into the plot buffer.
- **ACTTEMP** Loads thermal analysis information into the plot buffer.
- **ACTFLOW** Loads the specified flow component into the plot buffer.
- **ACTMAG** Loads the specified magnetic component into the plot buffer.
- **ACTFTG** Loads cumulative fatigue usage factors into the plot buffer.
- **ACTUSR PLOT** Activates a user plot for postprocessing.
- **SETPLOT** Sets plot type and specifies extreme values.
- **SETERASE** Provides the option to clear screen before new plots.

**PLOT Submenu**

- **IDRESULT** Displays location and value of the plotted quantity.
- **ANIMATE** Animates deformations, mode shapes and transient responses.
- **USRANIMATE** Animates images created by METAFILE or IMAGESAV commands.
- **DEF PLOT** Loads and plots deformations.
- **SM PLOT** Plots shear and moment diagrams for beam elements.
- **STRPLOT** Plots stress component previously loaded into plot buffer.
- **STNPLOT** Plots strain component previously loaded into plot buffer.
- **DISPLOT** Plots displacement component loaded into plot buffer.
- **TEMPPLOT** Plots the previously loaded temperature profile.
- **FLOWPLOT** Plots the flow component loaded into the plot buffer.
- **MAGPLOT** Plots electromagnetic component previously loaded.
- **FTGPLOT** Plots the previously loaded fatigue cumulative factor.
- **USR PLOT** Plots user defined quantities.
- **ISOPLOT** Generates isoplane plots of postprocessing quantities.
- **SECPLOT** Generates section plots of postprocessing quantities.
- **SETLSECPLOT** Sets parameters for path variation graphs.
- **LSECPLOT** Plots path variation graph of postprocessing quantity.

**LIST Submenu**

- **DISLIST** Lists displacement components.
- **ST RLIST** Lists stress components.
- **STNLIST** Lists strain components.
- **SMLIST** Lists shearing forces and bending moments for beam elements.
- **BEAMRESLIS** Lists nodal forces, moments and stresses for beam elements.
- **GAPRESLIS** Lists Gap element forces.
- **FREQLIST** Lists natural frequencies of the model.
- **TEMPLIST** Lists nodal temperatures.
- **FLOWLIST** Lists quantities related to a fluid-flow analysis.
- **FLOWPROP** Computes and lists Film coefficient and mass flow rate.
- **MAGLIS T** Lists quantities related to an electromagnetic analysis.
- **FTGLIST** Lists the cumulative fatigue usage factors.
### EXTREMES Submenu

<table>
<thead>
<tr>
<th>Command</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>DISMAX</td>
<td>Lists extreme values of displacements.</td>
</tr>
<tr>
<td>STRMAX</td>
<td>Lists extreme values of stresses.</td>
</tr>
<tr>
<td>STNMAX</td>
<td>Lists extreme values of strains.</td>
</tr>
<tr>
<td>SMMAX</td>
<td>Lists extreme forces (over full length) for beam elements.</td>
</tr>
<tr>
<td>BEAMRESMAX</td>
<td>Lists extreme nodal forces and stresses for beam elements.</td>
</tr>
<tr>
<td>TEMPMAX</td>
<td>Lists extreme values of temperature profiles.</td>
</tr>
<tr>
<td>FLOWMAX</td>
<td>Lists extreme values of flow components.</td>
</tr>
<tr>
<td>MAGMAX</td>
<td>Lists extreme values of electromagnetic components.</td>
</tr>
</tbody>
</table>