Press Brake Tooling

Folded Part Simulation
The first thing I did was I made an assembly of your tooling, this way you can use the same tooling assembly for different sheet metal parts. I used 2 mates to do this, on mate of both right planes and one mate of both side faces, this allows the punch to move up and down as in real life.
I then opened your sheet metal part and made a flat pattern configuration using the un-fold tool, then one by one I created a new configuration for each fold.
Flat Pattern Configuration
Fold 1 configuration
Fold 2 configuration
Fold 3 configuration
Fold 4 configuration
Fold 5 configuration, you will now see 1 Unfold and 5 separate fold icons in the feature manager three, one fold icon for each configuration each surpressed or unsurpresed as required
I then made a new assembly using the press brake tooling assembly and the folded part, I then selected the assembly in the feature manager three and selected properties.
Under the properties window I selected flexible so as I could now move the punch up and down.
I now created the configuration fold 1 in the assembly, I selected the folded part and used it’s configuration fold 1. I then mated the part with the bottom die using 2 coincident mates between the flat faces of the folded part and the top inner edges of the die, I then used a tangent mate to mate the punch in place, you might have to play around with the tangent mate to get it to work correctly. The folded part will not be fully fixed in place you will be able to sway it left to right you can apply more mates if you want but I did not as I did not see the need
I then selected the mates for fold 1 and inserted them into a new folder and called it fold 1 for easy management, you can see fold 2 to fold 5 are suppressed.
I then made a configuration in the assemble and called it fold 2, I then selected the folded part and changed the reference configuration to fold 2 under its properties window. I then mated the curved section of the fold to the top inner edges of the die using 2 mates and again a tangent mate between the punch in inside of the bend.
I did the same again for fold 3, fold 4 and fold 5 each time creating a configuration in the assembly and changing the reference configuration in the folded part properties.
This way you end up with a configuration for each fold
You can then bring a drawing view of the assembly into your drawing using the different configurations for each time you want to show the next bend in the bend process.