CUSTOM FORM SHAPES

The Custom Form IntelliShapes allows you to add a user-defined form shape to a sheet metal part. As part of the default sheet metal catalog, there are many standard forms that can be added and customized by variables in the form properties. However, there may be additional forms that are not included that are necessary for your design.

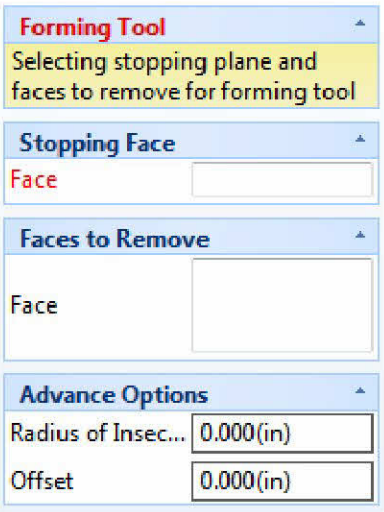
Using the Custom Form Utility, user can define a Form from standard solid geometry in the scene. The concept is that you create the solid shape that would be used to push (form) into a Sheet Metal part and the result is applied on the Sheet Metal shape.

Notes:

- The Forming Tool creates a definition on a standard part. Once the definition is created, drag and drop the part into a catalog to create the Custom Form.
- Custom Forms dropped on sheet metal cannot be editing in its shape. All modifications have to be performed on the Forming Tool and re-dropped on the sheet metal part.
- Design Variations are supported on the feature level of the Forming Tool Definition. When dropping the Custom Form on a sheet metal part, it will pop-up the Design Variation to allow selection of the feature. After accepting the Design Variation, it cannot be changed again unless you re-drop the Custom Form.

To Create A Custom Form

1. Create geometry using standard solid modeling tools in the 3D Scene. Note: You can also leverage imported data and use Design Variations to define the feature.
2. Select the Forming Tool command from the Sheet Metal Fluent Ribbon Bar. The Forming Tool Property Command Browser will appear.

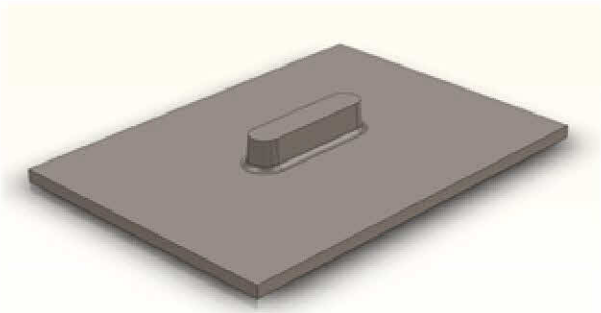


3. Select the Stopping Face. This is the face that represents the face on the Sheet Metal part where the form will be applied in a Negative Normal Direction.
4. Select the Faces to Remove. This is optional but you can select faces to be open to create necessary open areas for the form.

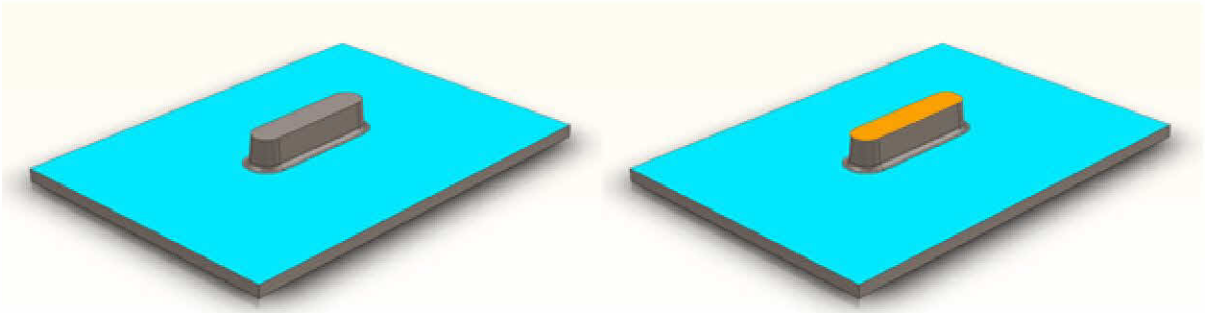
5. Set any advance options such as a radius at the stopping and form edge (only needed if a blend is not already applied). Offset allows the ability to offset the form to change the depth of the form into the sheet metal part.
6. Select Ok to complete the operation.

At this stage, a new indicator will be applied to the part in the scene browser denoting that it has a forming tool definition. Right-click on this indicator to edit the properties of the forming tool definition.

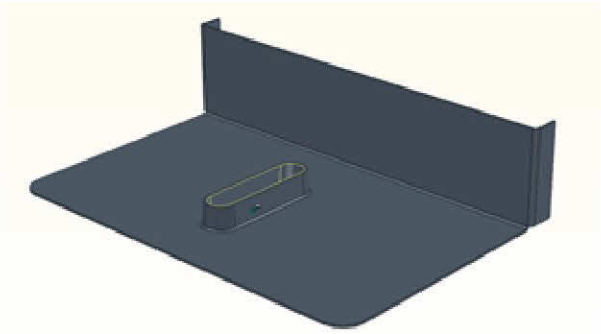
Next, drag & drop the part into a catalog to create the forming tool. When needed, simply drag & drop the shape onto a sheet metal part to apply the forming tool.



Solid Part Base Example for a Form Tool



Forming Tool Selections (Blue: Stopping Face, Orange: Open Face)



Forming Tool Result Added to Sheet Metal Part