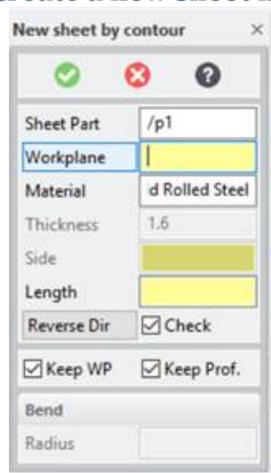
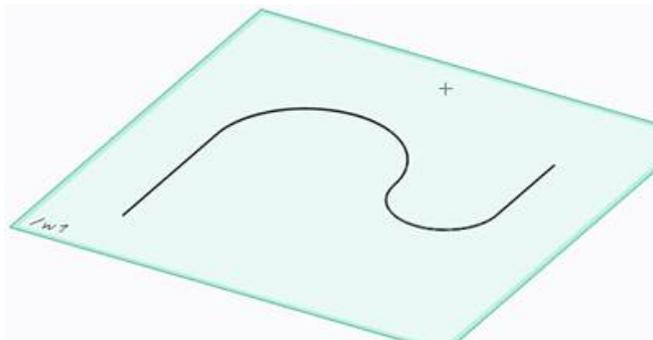


Create a new sheet metal part with profile contour



The command `sha_new_by_contour` is similar to the original command `sha_new_by_polyline`. The primary difference is the new contour command allows curved elements to be included in the open profile. The curved elements can be lines, arcs or splines. The primary requirement is that the contour must begin and end with a straight segment.



If the elements in the profile are tangentially connected, then no bend radius will be inserted between adjacent elements. If two adjacent elements are not tangentially connected, a bend radius of the specified size will be automatically entered with the new sheet metal part is created.

- Sheet Part – a new or empty part. Cannot be an existing part. Use the command `sha_add_by_contour` if it is desired to add this contour to an existing part.
- Workplane – a workplane containing an open profile.
- Material – select a material from the available material database list
- Thickness – derived from the selected material
- Side – select side of open profile that material will be produced
- Length – Distance to extrude profile to create the new part
- Reverse Dir – Toggle to select whether to extrude into the workplane or out of the workplane
- Bend – Only available if adjacent elements are not tangentially connected. Otherwise enter the default radius that will be applied to all non-tangentially connected elements.

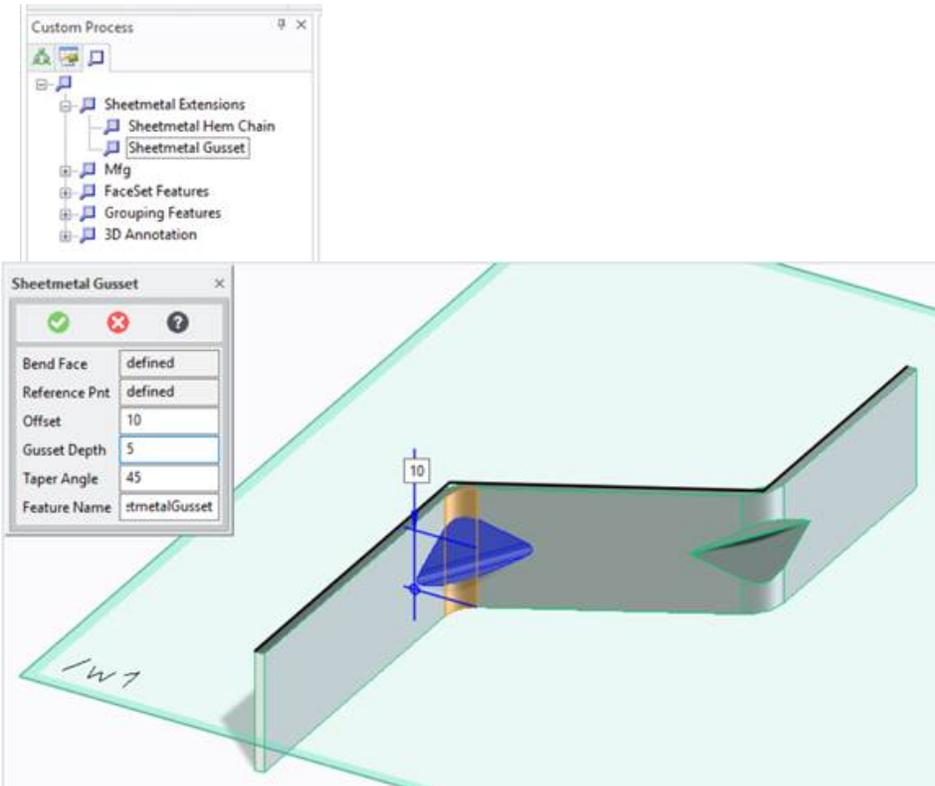
Add to part with profile contour

This command is used to add material to an existing sheet metal part using an open contour. This command works the same as the command above to create a new part with an open contour. It is nearly identical to the above command other than the following two items:

- Sheet Part must be an existing sheet metal part.
- You can specify a bend relief where the new material will be added to the existing sheet metal part. See the command `sha_add_sheet_by_polyline` for more details on bend reliefs.

Create/Modify Gusset

This command allows the creation of a gusset typically used to strengthen a particular bend on a sheet metal part. It is also possible to modify an existing gusset to change its size or location. Shown below is a gusset from both the front and back side of a sheet metal part. To create a gusset, open up the custom feature browser and select the option SheetMetal Gusset.



- Bend Face – Select a bend face on an existing sheet metal part where the gusset will be placed. It is allowed to have multiple gussets on the same bend.
- Reference Pnt – The reference location from where the offset is measured. By default, this point is the closes side face to where you picked on the bend face to place the gusset. You may modify this reference point to any other location on the model. This new reference point will then be used where the offset will be calculated from.
- Offset – set by default to be the location where you selected the bend face to determine where the mid-point of the gusset is to be located. You can modify this value as required. The distance is measured from the closes edge of the bend face. The distance must be greater than 0 and less than the length of the bend (i.e. it cannot exist outside of the bend face.)
- Gusset Depth – The depth of the gusset measured from the inside of the bend face to the top edge of the gusset. Enter the required depth for your application.
- Taper Angle – Is the angle of the gusset side faces measured from the mid plane of the gusset.
- Feature Name – the name to be assigned to the gusset.

Gussets may be placed on any bend face regardless of the bend angle. They can be applied to right angle bends or any other bend angle. Once created, gussets may also be modified, suppressed or resumed. To modify an existing gusset, there are two available methods:

- Select the gusset from the structure browser, right click on the object and select the modify option.
- Use the Modify Custom feature command and select the desired gusset in the viewport or in the structure browser.

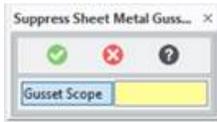
Both methods will display the gusset UI as described above and the user may modify any of the desired parameters.

Delete Gusset

To delete a gusset, select the gusset from the structure browser, right click on the object and select the delete option. A gusset can also be destroyed by using the delete custom feature command. In this case, the user can select the gusset from the viewport or select it within the structure browser. When a gusset is destroyed, it cannot be recovered/resumed at a later date (unless using the undo system).

Suppress Gusset

It is possible to suppress an existing gusset. Suppressing a gusset remove its geometrical representation from the model, but does not destroy its definition. This is useful to allow a sheet metal part to be flattened (since sheet metal parts with gussets cannot otherwise be flattened).



You can specify which gussets to suppress either by selecting either on a bend face or selecting a sheet metal part. If a bend face is selected, all gussets on that bend face will be suppressed. If you select the sheet metal part, then all gussets on the part will be suppressed.

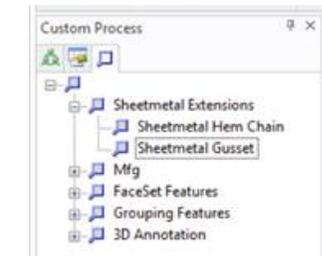
Resume Gusset

After a gusset has been suppressed, it may be recovered using the resume gusset command. To resume a gusset, you may either select on the bend face or the sheet metal part. If a bend face is selected, all gussets which existed on that bend face will be resumed. If the sheet metal part is selected, all gussets on the part will be resumed. Once a gusset has been suppressed, it can be resumed at any later point in time. Unlike the undo system, resume can be used even in different sessions of Modeling. There are two primary restrictions to allow suppressed gussets to be resumed:

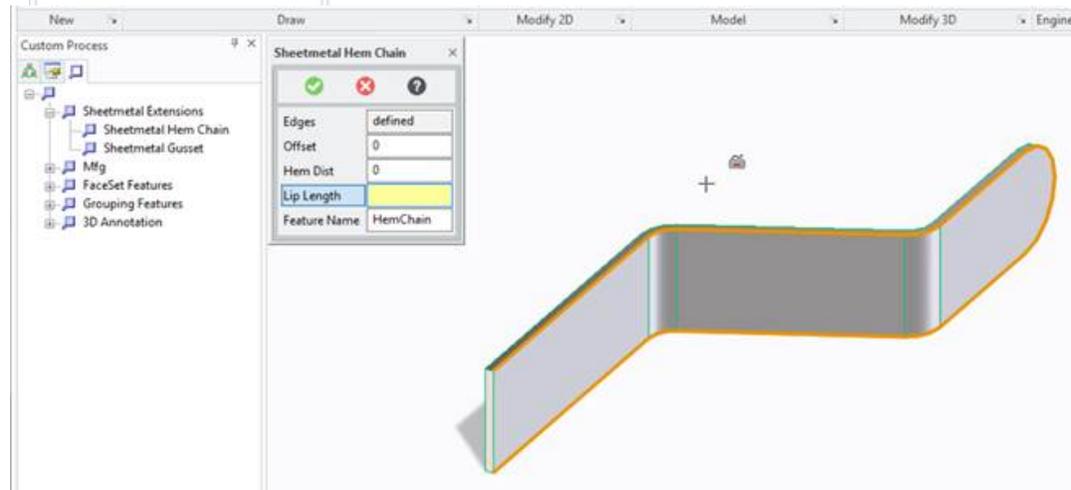
1. The bend face must still exist for a gusset to be resumed.
2. The reference point of the gusset must exist on the model when using the resume command.

If either of these conditions is not met, a suppressed gusset will no longer be recoverable.

Hem Chain

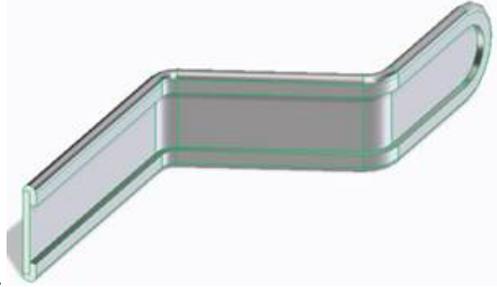


The Sheetmetal Hem Chain command can be used to create more complex hems not allowed by the standard Sheet Advisor Hem command. The Sheetmetal Hem Chain command is invoked via the custom feature browser as this command generates a feature to represent the hem.



- Edges – select one more edges to create a hem upon. If more than one edge is selected, they must be tangentially connected. In the above example, the chain selection method was used to all of the above highlighted edges with a single pick.
- Offset – the distance to offset the side face before creating the hem. This option basically allows the existing sheetmetal part to be enlarged by the offset distance before producing the hem.
- Hem Dist – the distance between the inside face of the hem and the sheet metal face being hemmed. Can be 0 for a closed hem.

- Lip length – length of the hem. This is the distance the hem will overlap over the original sheet metal part. The lip length must be smaller than the smallest radius of any edge included in the hem, otherwise an overlapping hem



would be produced.

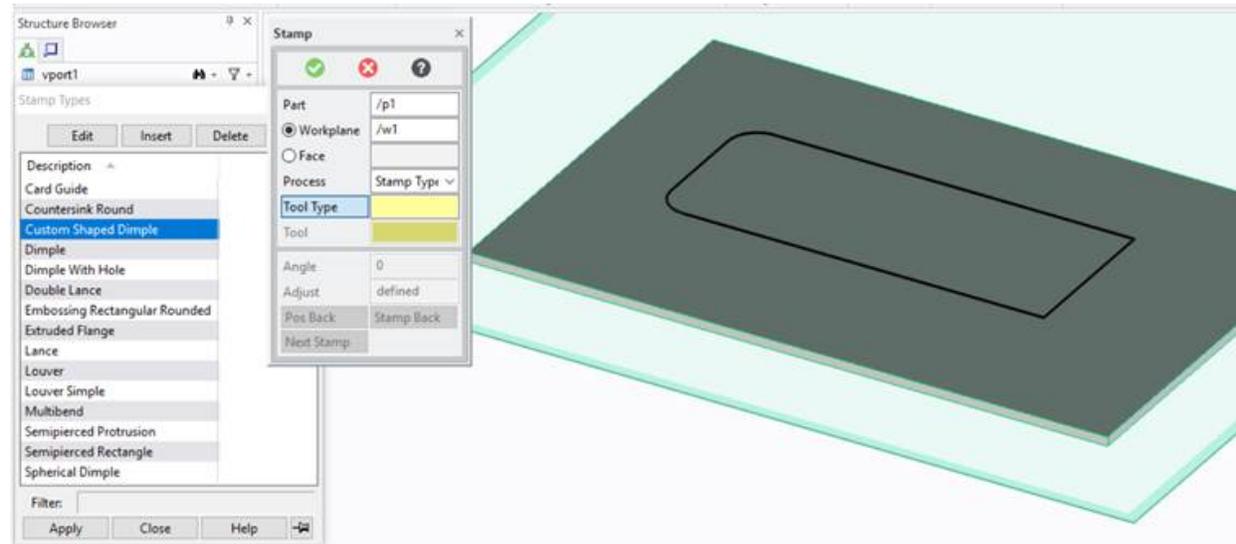
Once a hemmed feature has been created, it can be later deleted using the delete custom feature command.

Limitations

- This command does not have any options to produce bend reliefs
- Once created, a hemmed feature can not yet be modified. A possible future extension may make this possible.
- A hemmed feature can not be suppressed or resumed. A possible future extension may make this possible.
- Some complex hems may interfere with the ability of Sheet Advisor to flatten a sheet metal part.

Custom Shaped Dimple

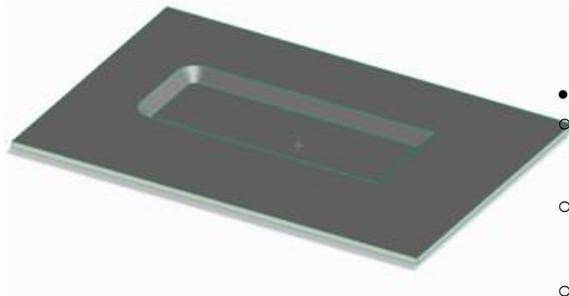
This custom shaped dimple Stamp option allows a custom shape to be used to define the dimple. Unlike other stamp operations, this stamp utilizes the profile on the current workplane to determine the shape of the dimple.



To create a custom shaped dimple, first draw a valid closed outline of the dimple on the current workplane. Then initiate the stamp operation. Select the Custom Shaped Dimple for the tool type. Once selected, a list of predefined depths, angle and directions will be displayed. Choose one of these values or add your own value to the tables.

- Depth – distance measured from the sheet metal face to the top of the dimple.
- Angle – The taper angle of the dimple
- Direction – Either UP or DOWN. Determines the direction of the stamp.

Once the above information is specified, a construction point is placed on the profile to allow accurate placement of the tool when it is desired for it to coincide with the location of the profile itself. Simply pick the construction point, and the custom dimple will be created exactly at the same location where the profile is located. Of course, any alternate location can also be selected. Once the position of the custom dimple has been selected, it is also possible to select additional locations. For



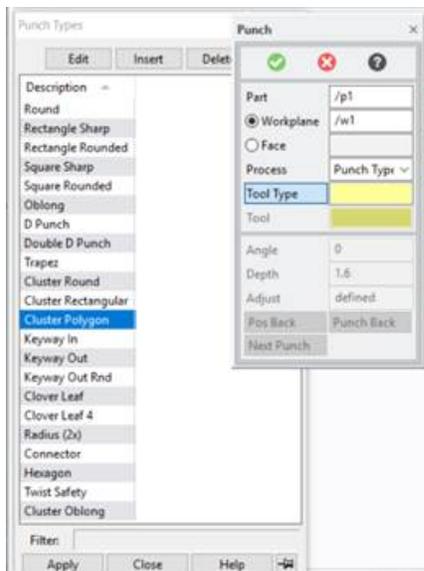
each location specified, a custom shaped dimple will be created at each location.

- **Limitations**
 - The specified angle and depth combination must not create a condition where the dimple would become self-intersecting or an edge of the dimple would become negative in length.
 - For example, if the profile of the custom shaped dimple contains a radius of 5, for a taper angle of 45 degrees, the maximum depth of the dimple cannot be less than 5 units.
 - If the maximum depth is exceeded, the system will highlight

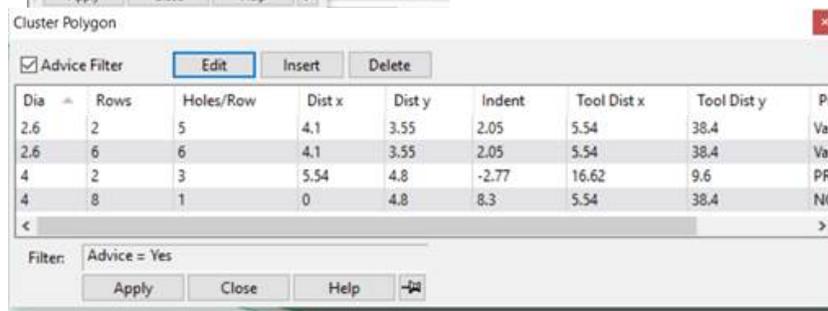
the edges on the profile that are problematic and display an error message indicating what the issue is. To fix the problem, change the depth, angle or modify the 2d profile of the custom shaped dimple.

Cluster Polygon Punch

A new stamp type called Cluster Polygon has been added to the list of available punches. Use this tool to create a cluster of polygon shaped holes.

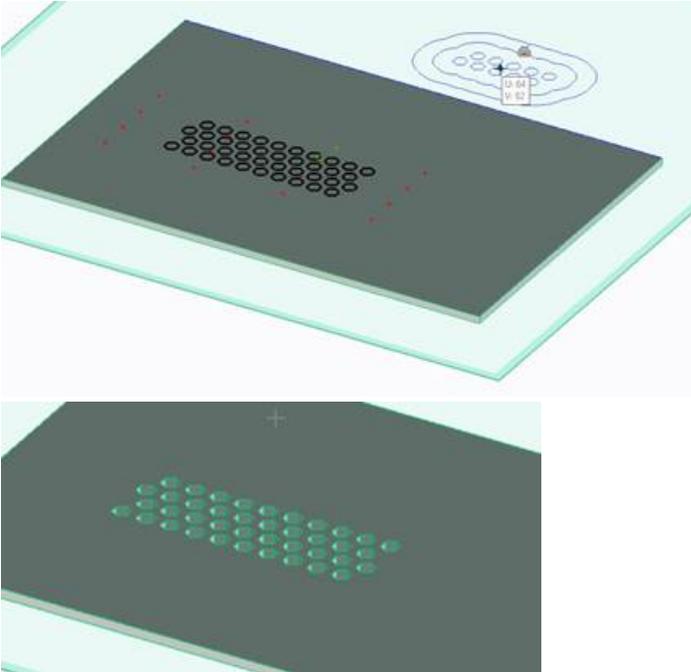


To use, initiate the punch command. From the list of tool types, select the Cluster Polygon option.



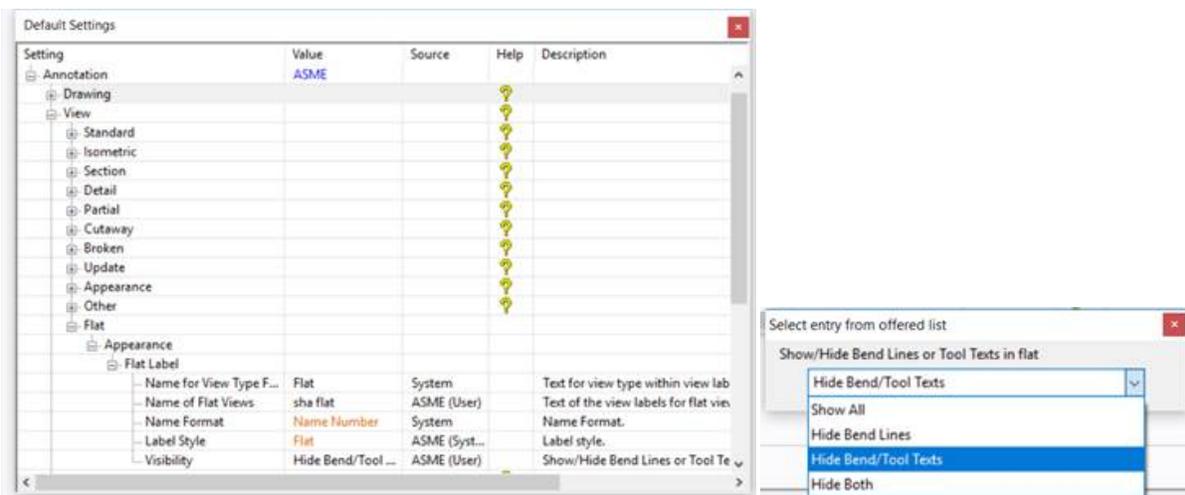
Select one of the available sizes from the table, edit an existing row to change one or more parameter(s), or use the insert option to create a new cluster polygon.

Lastly, select the location where the cluster should be placed. A feedback model is created along with 9 construction points to allow accurate placement of the cluster. If multiple clusters are desired, select multiple positions. Use the construction points to create clusters which are aligned with each other.



Bend Line and Tool Text Visibility

In the annotation module, a new setting has been introduced to control the visibility of bend lines and tool texts in a flat view. By default, both Tool texts and Bend lines are visible in a flat view. To modify this behavior, set the setting via the annotation setting browser.



The setting, Annotation/View/Flat/Appearance/Flat Label/Visibility controls this new behavior. There are four possible options:

- Show All – the default. Show both bend lines and tool texts in the flat view.
- Hide Bend Lines – Don't display the bend lines in the flat view
- Hide Bend/Tool Texts – hide the text labels for bend lines and tool texts in the flat view
- Hide Both – Don't show bend lines or tools texts.

NOTES:

- Any flat view created after this setting has been set, will adhere to mode set.
- The setting is persistent. This means that when Modeling/Annotator is started, the last used value will become the default value.

Controlling the display of Pseudo Folders

By default, the display name of pseudo folders in the structure browser are derived from the first child node of the pseudo folder. Should that child have a Model Name, that will be used for the pseudo folder. If the child has no Model Name, then the part name is used instead. For some cases, this may not be a desired outcome. To overcome this issue, a lisp function can be created to provide an alternate name to the pseudo folder which contain parts or assemblies. Some integration kit knowledge is required to implement this alternate naming option.

The first step, is to implement a function called pseudo-folder-alt-name in the :valve lisp package. The signature of this function is as follows:

```
(defun valve::pseudo-folder-alt-name(bnode children)
```

```
...
```

```
)
```

Description:

The function used to return an alternate name for a pseudo folder which contains shared parts/assemblies

Parameters:

bnode {BrowserNode-struct} This is the browser node to a pseudo folder containing shared parts/assemblies.

Children {List of BrowserNode-structs} A list of browsernode structs for all children contained within the browser node.

Return Value:

(values {STRING} {Bnode}) – both the string value of the alternate name to display, and the BrowserNode of the child used to derive the name from.

Nil – use the default name for this pseudo folder

Example:

```
(defun valve::pseudo-folder-alt-name(bnode children)
  (let (child (first (last children)))
    (values (browsernode-objpname child) child)
  )
)
```

In the above example, we first select the browsernode that we use to derive the name.

- we select the last item in the list of children (children are passed to this function in reverse order of what is shown in the structure browser).
- We then return the name of the child and the node from which the name was derived. The returned node is used to display the correct ModelManager status data.
- This example duplicates the default behavior. Although a much simpler implementation would have been to simply return nil.