Generating offset surfaces from a complicated surface body

Generally, when you model a sheet metal body with finite thickness, you create one side and using offset command you can create the other side.

This modeling method is used to create a sheet metal body with thickness even when you have only its upper side surface. Then you can check the interference between inner side of the surface body and another part. However, if you have a surface body with some complex shape, it is not easy to create it using above mentioned method.

XOR helps you to easily convert a surface data to mesh in order to complete the inner shape from the converted mesh using offset operation. You can quickly create a surface body on the mesh using Auto Surfacing command and complete the whole shape connecting the inner and the outer surface body on the boundary of model.

Through this document, you will understand how to create a complex inner side of body from the existing design data using the Convert Body, Auto Surfacing commands.

Method

First of all, you need to prepare a mesh from the existing design data in order to apply the offset operation.

1. Go to Mesh mode and select Tools > Mesh Tools > Convert body in the menu. Click the Ok button.

The design data is converted to a mesh and the copied mesh data will be registered in the Model Tree as shown in the image below.
Now extract a curve along the boundary of upper body using Convert Entities command. Then extract a curve along the boundary of the converted mesh using Boundary command as well.

1. Go to the Mesh mode, select Tools > Mesh Tools > Offset in the menu.

2. Select the Surface Method as offset method and input a thickness value in the Distance box.

3. Choose offset direction using Flip button so that it is offset to the inside of the part.

   Note
   You can choose offset direction by just clicking the (Flip) button. If you want to turn over the current direction to the opposite direction, click the Flip button and the copied mesh will be offset along the selected direction. Also if you click this button again, the offset direction will be reversed again.

4. Go to the 3D Mesh Sketch mode, or select Tools > 3D Sketch Entities > Boundary in the menu.
5. Select the option Interactively-Split Curve Segments From Single Boundary and select an edge as Boundary. Then pick nodes to create partial boundary curves as shown in the image below.

6. Go to the 3D Sketch mode, or select Tools > 3D Sketch Tools > Convert Entities in the menu.

7. Extract a curve along the boundary of upper body using Convert Entities command as shown in the image below.

8. Try to merge between the split splines using Merge command.

Finally, connect the curves that are extracted from the upper side of body and the inner side of mesh using the Surface Loft command. You can also easily and quickly create an inner side of body from the mesh using Auto Surfacing command.

1. Try to execute the Surface Loft command. Click the Surface Loft button in the main toolbar.

2. Select two sketch chains as Profiles and click the OK button.
3. You can also create other lofting surfaces by the same way as shown in the image below.

4. Try to apply Auto Surfacing command to the offset mesh. Click the Auto Surfacing button in the main toolbar.

5. Select the mesh and set the value for No. Of Surfaces as shown in the image below.
A surface body will be created on the offset mesh as shown in the image below.

6. Try to execute the Sew command. Click the Sew button in the main toolbar.

7. Select all surface bodies and click the next button.

8. Click the OK button. The split boundaries of the surface bodies are sewed. Then you
will get a solid model which has thickness as shown in the image below.

<Upper Surface>                          <inner Surface>