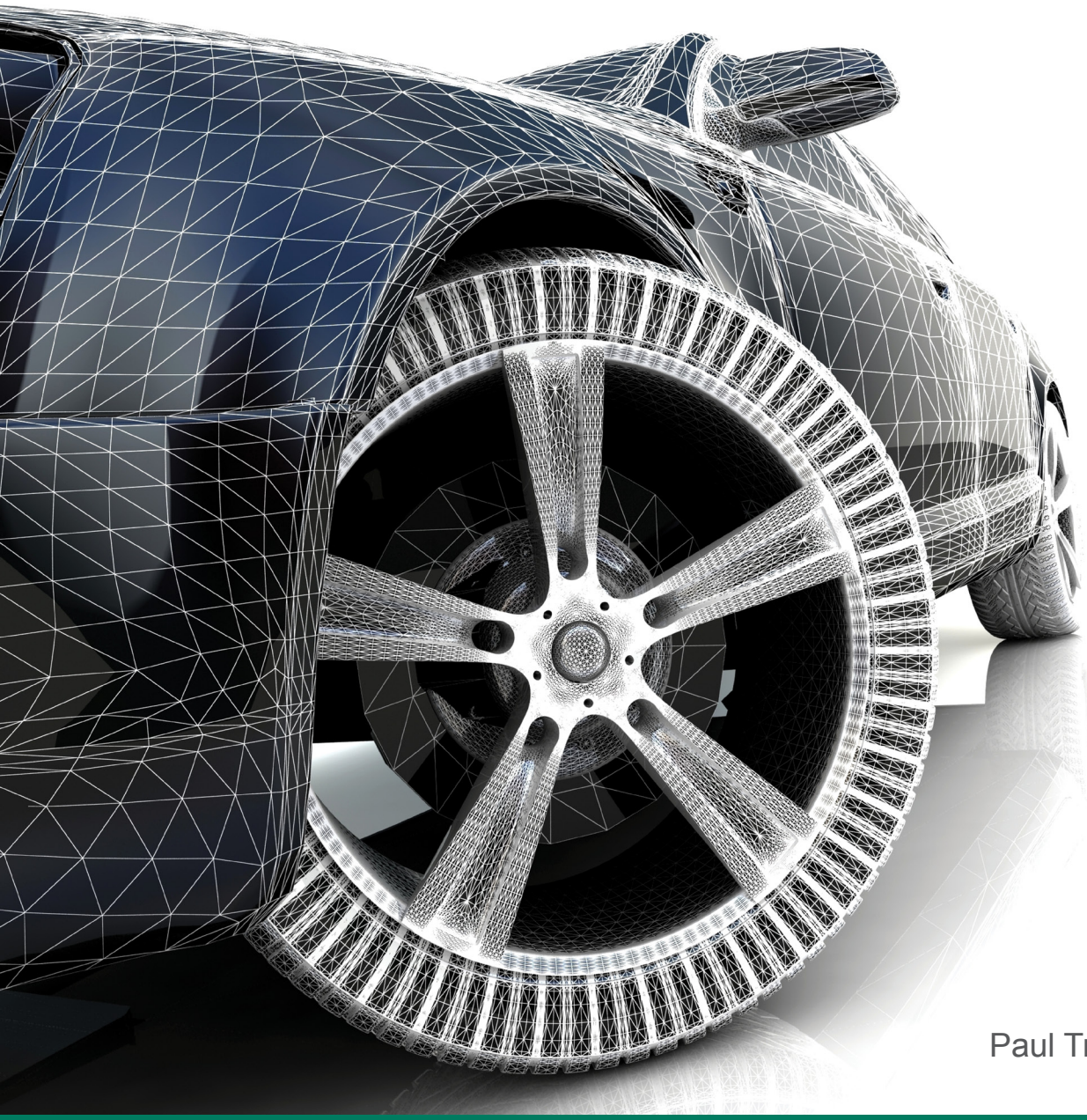
CSWE - Certified SolidWorks® Expert Preparation Materials

SolidWorks 2010 - 2013



Paul Tran, CSWE, CSWI

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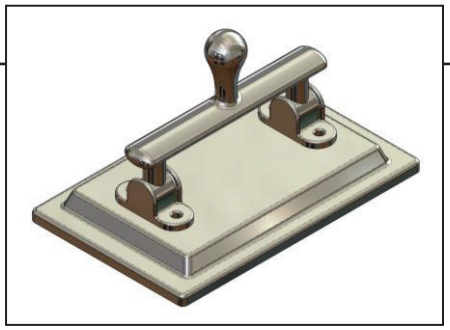
CHAPTER 1

Surface Modifications

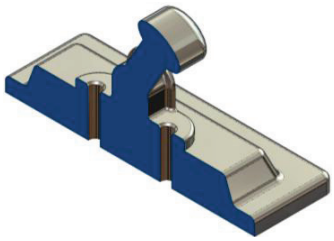


CSWE Exam Preparation

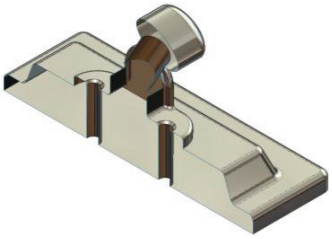
Surface Modifications Summary

When working with imported parts, whether the imported model is a solid model or a surface model, we will have to deal with one major disadvantage, and that is not having the features history available for modifications.

The SolidWorks software offers a few tools for manipulating the geometry of the imported model such as surface move, rotate, copy, delete, etc. When used together with the surfaces commands, much more can be achieved.

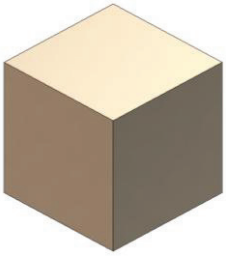


Solid Model

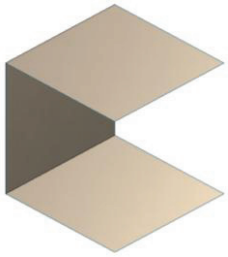


Surface Model

When a surface model has a number of faces that consists of a closed volume, it can be converted into a solid model. However if the surface model is opened, it can only be made into a thin walled part.



Closed Volume = Solid Model



Opened = Surface Model

This lesson focuses on the techniques that are used in the CSWE exam, where the solid model will be converted into a surface model, and the handle will be rotated into the upright position. The deleted faces will be patched up and knitted to the entire model and then become one closed volume. The volume will then get converted back to a solid model.

CSWE Exam Preparation

Surface Modifications

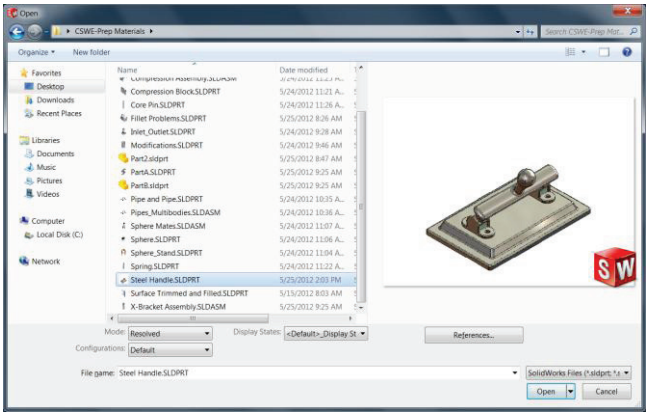
Dimensioning Standards: **ANSI**

Units: **INCHES** – 3 Decimals

Tools Needed:

Temporary Axis	Move / Copy	Delete Surface
Loft Surface	Trim Surface	Knit Surface

1. Opening a part document:

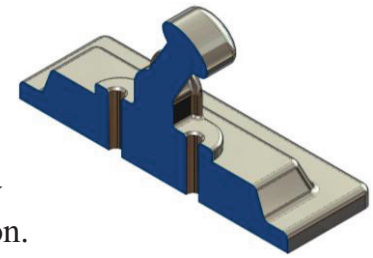


- Click **File / Open**.

- Browse to the Training CD locate and open the part named: **Steel Handle**.

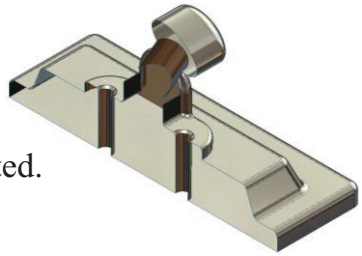
- This model was created as a solid, but the approach that we are about to take will turn it into a surface model.

- The sample images below show the cross sections of the model, before and after the Delete Face operation.



2. Deleting the faces of the model:

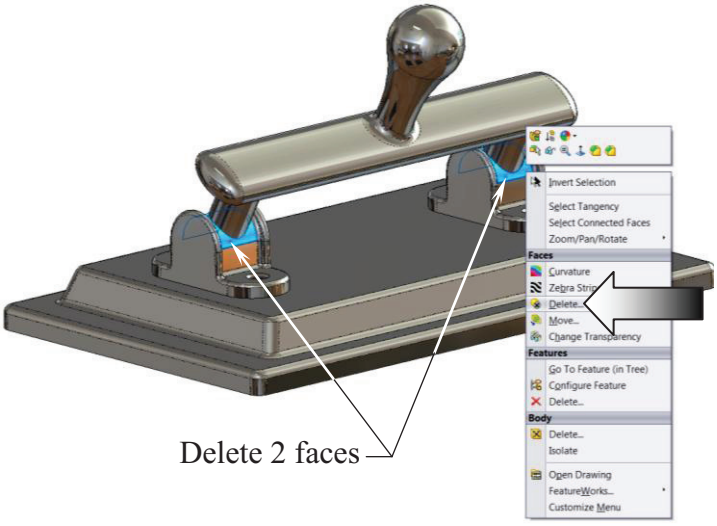
- Right click on one of the 2 faces as indicated below and select the **Delete** option below the Face selection.

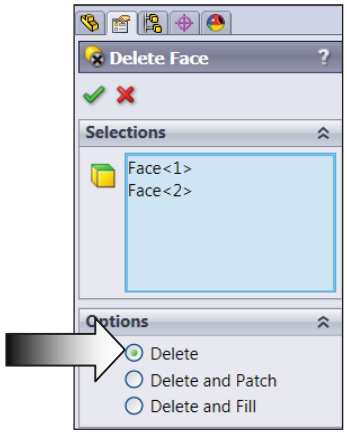


- The Delete Face options appear on the Properties tree.

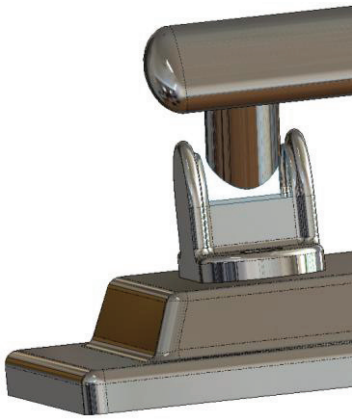
- Select the **Delete** option and select the 2 faces as noted.

- Click **OK**. The 2 selected faces are removed and the Solid model changes to a Surface model.

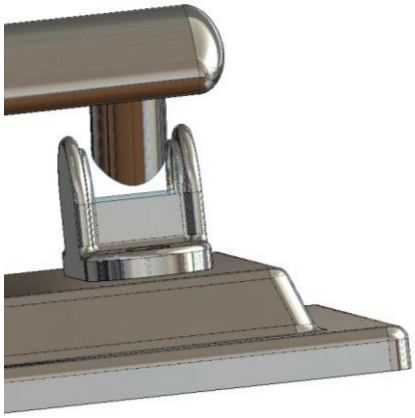




- Rotate and zoom closer to examine the result.



Left end



Right end

- Because the solid model has changed to a surface model, there is no mass in it and the Mass Properties command is grayed out at the moment.

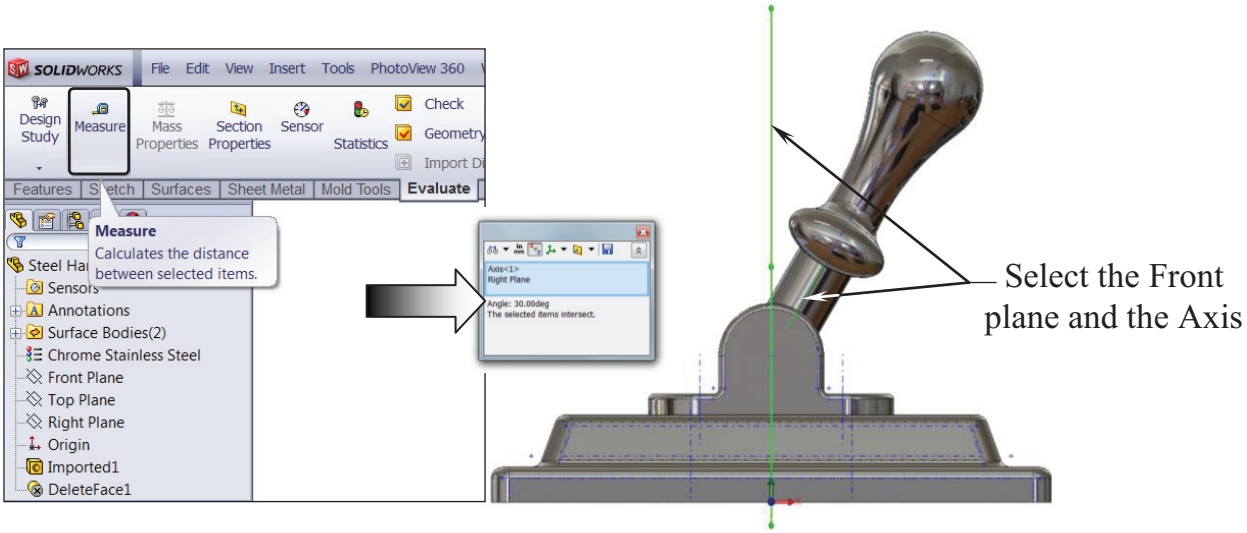
- The intention is to separate the handle from the base, so that it can be rotated to the upright position. The 2 deleted faces will get recreated and knitted with the surface model to become one closed volume and then thicken into a solid.

3. Measuring the current angle:

- Change to the Right orientation (Control + 4), and enable the **Temporary-Axis** from the **View** pull down menu.

- Switch to the **Evaluate** tool tab and select the **Measure** command.

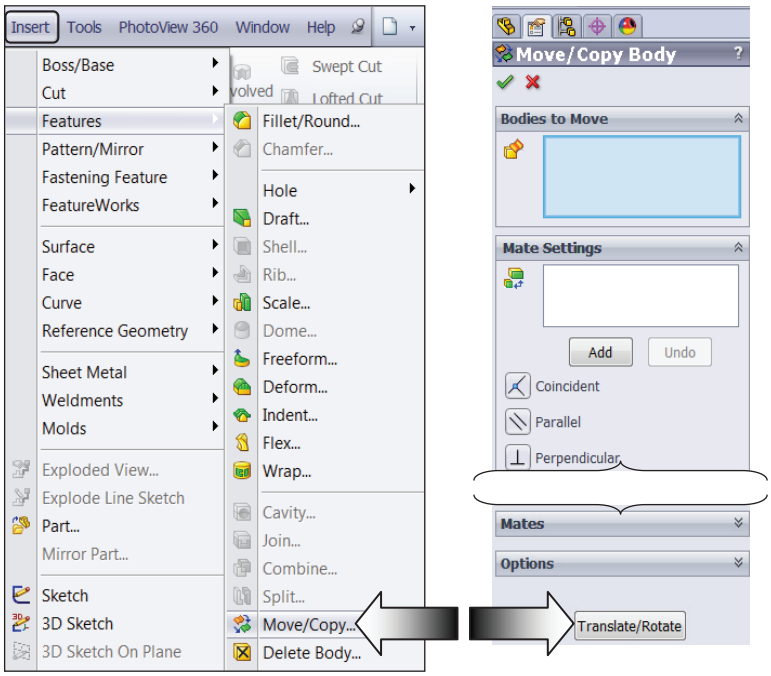
- Measure the angle between the **Front** plane and the center **axis** of the handle.



- Exit the Measure command.

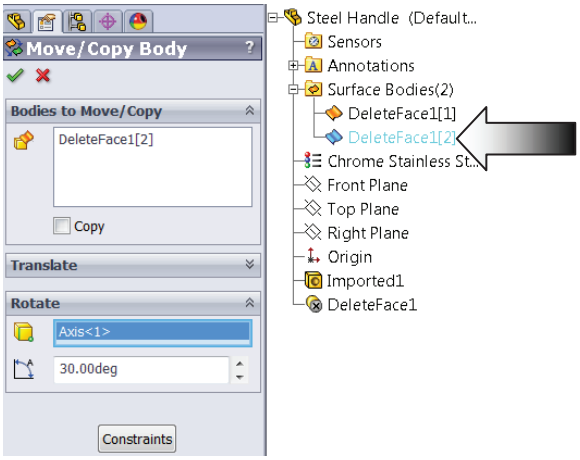
4. Rotating the handle:

- Select the **Move/Copy** command (arrow) from the View pull down menu.



- At the bottom of the Properties tree, click the **Translate/Rotate** button (arrow).

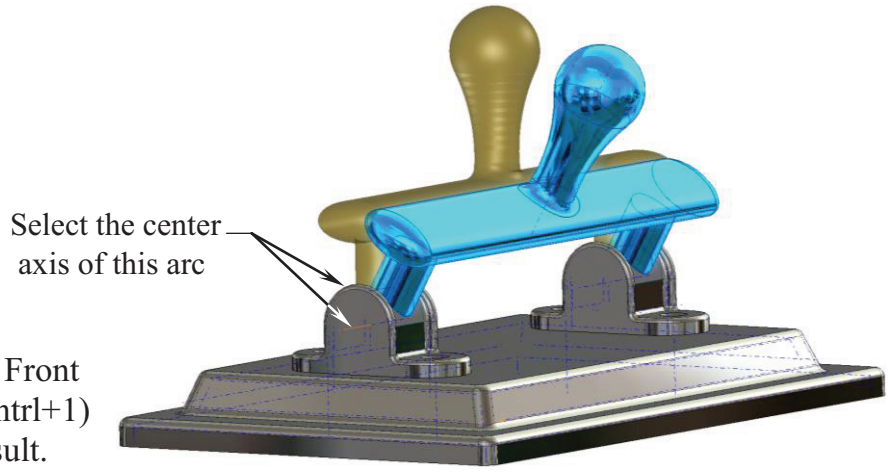
- Under the Bodies to Move / Copy selection box, select the Handle from the Surface Bodies folder (arrow).



- Expand the Rotate selection box and select the Axis as indicated below.

- Enter **30deg** for angle.

- Click **OK**.

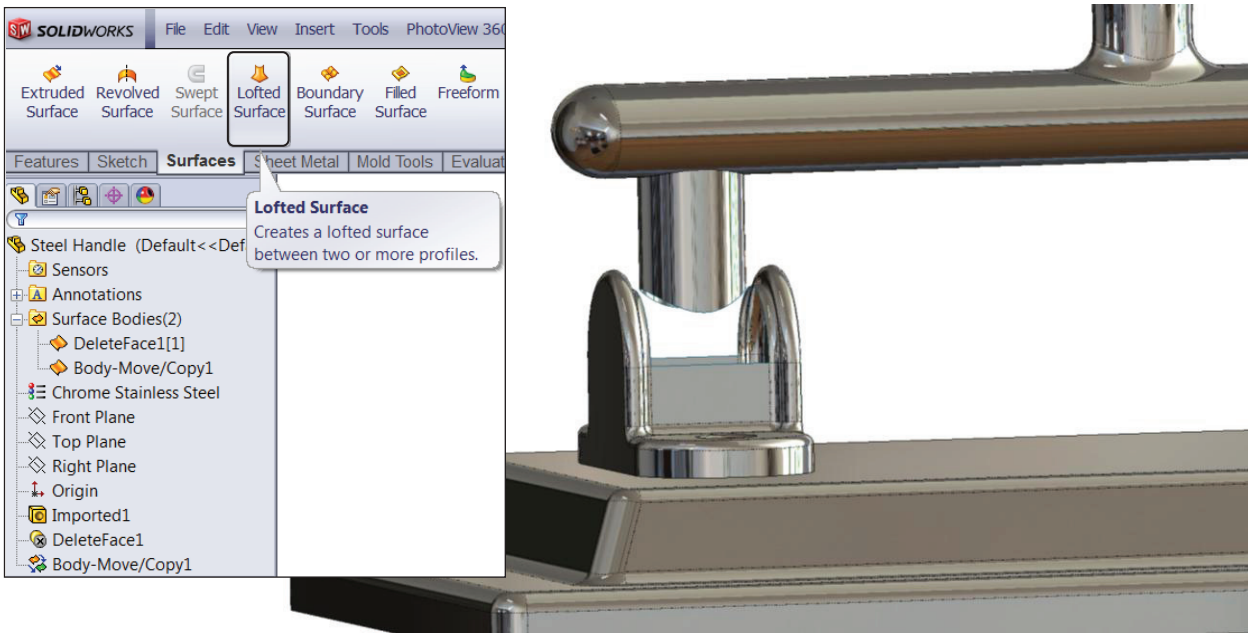


- Change to the Front orientation (Cntrl+1) to view the result.

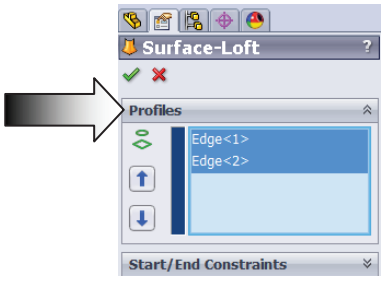
5. Recreating the deleted surfaces:

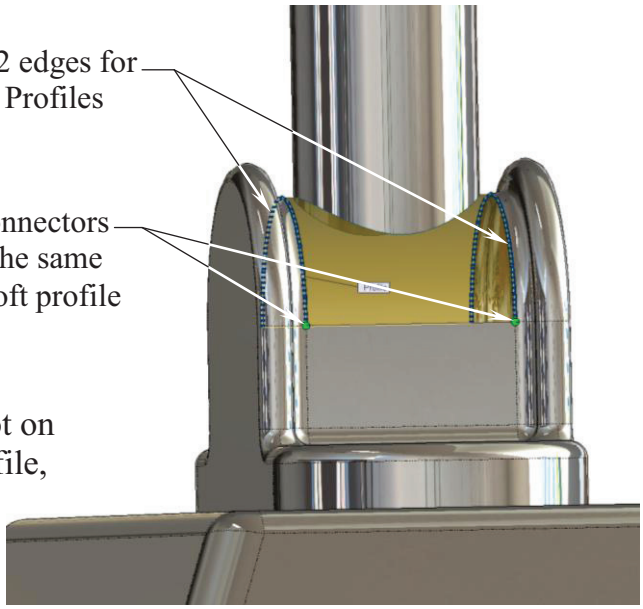
- Now that the handle has rotated to the upright position, we can fill in the gaps between the handle and the base. One of the ways to do that is Surface Loft.

- Switch to the Surfaces tool tab and select the **Lofted Surface** command.



- Under the **Profile** selection box, select the <u>2 curve edges</u> as indicated below (more settings on next page).

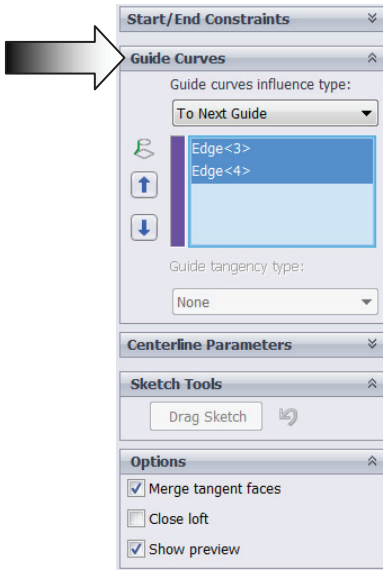


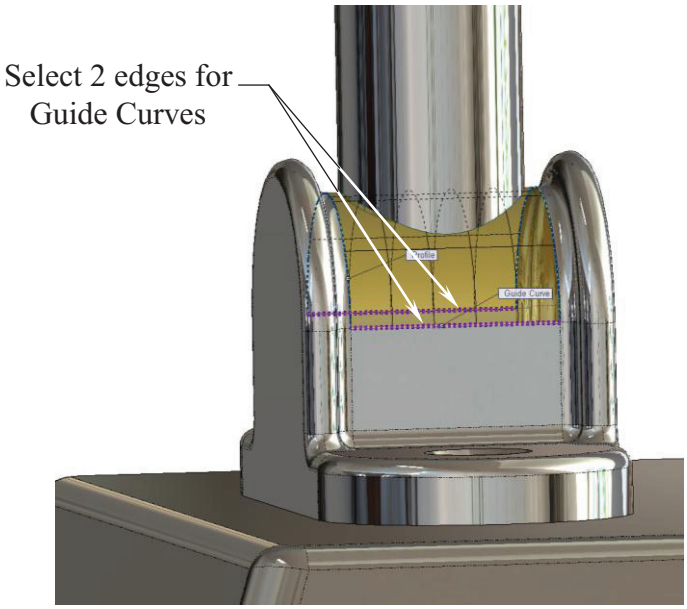


- If the green connectors are not on the same size of each loft profile, the loft will be twisted. Drag the connectors if needed to correct them.

- Click inside the **Guide Curves** selection box to activate this option.

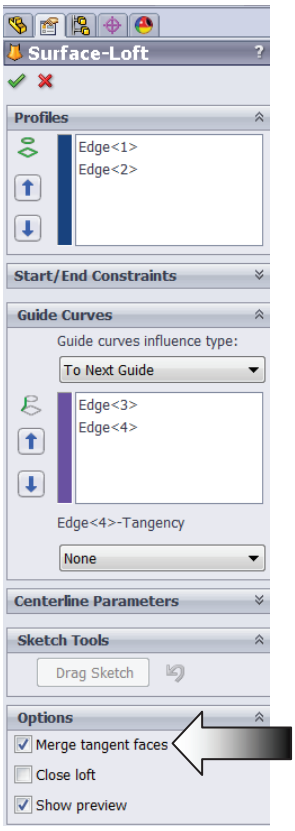
- Select the 2 linear edges as noted to use as guide curves.

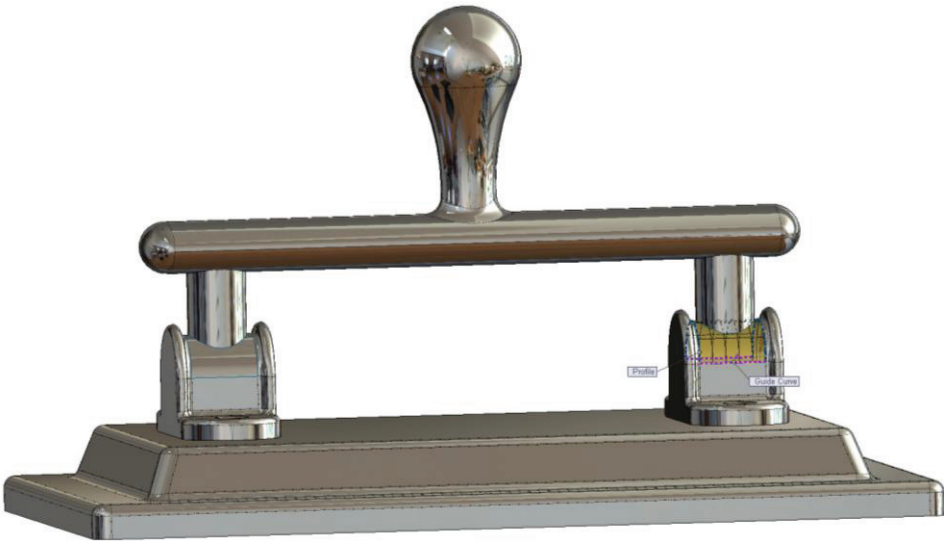




- Under the **Options** selection box, enable the checkbox **Merge Tangent Faces**.

- Also enable the **Show Preview** checkbox.

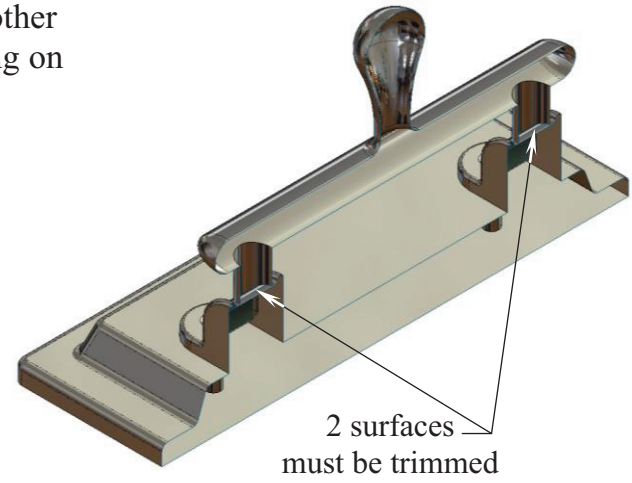




- The yellow preview indicates that the surface loft is being created.

- Click **OK** to accept and exit the Surface Loft command.

- Repeat the step 5 and create another Lofted-Surface to fill the opening on the other end of the handle.



- Create a section view using the Right plane. Verify that all gaps have been filled at this point.

- There are 2 surfaces under the handle that need to be trimmed, and it is quite difficult to see clearly where they are. So we are going to hide the base, and work with the handle only.

6. Hiding a surface body:

- One of the major advantages in working with multibody parts is the ability to hide or show any of the bodies at any time, and a new feature can be added to the selected body without affecting the others.

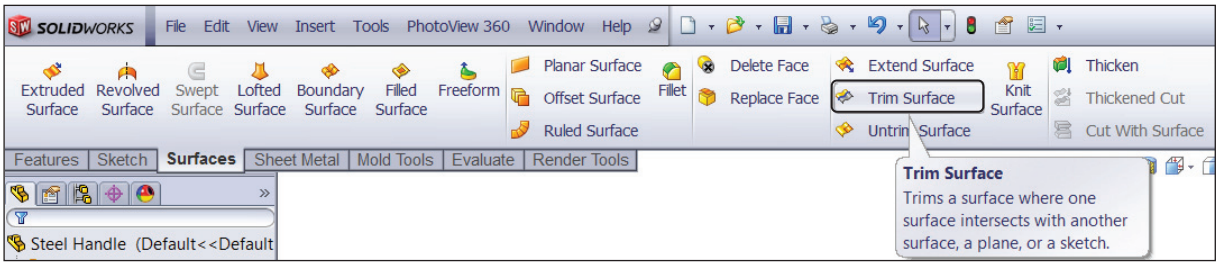
- The bodies can be selected directly from the graphics area or from the Surface Bodies folder, listed on the FeatureManager tree.

- Click anywhere on the base and select the **Hide** command (arrow).

7. Trimming surfaces:

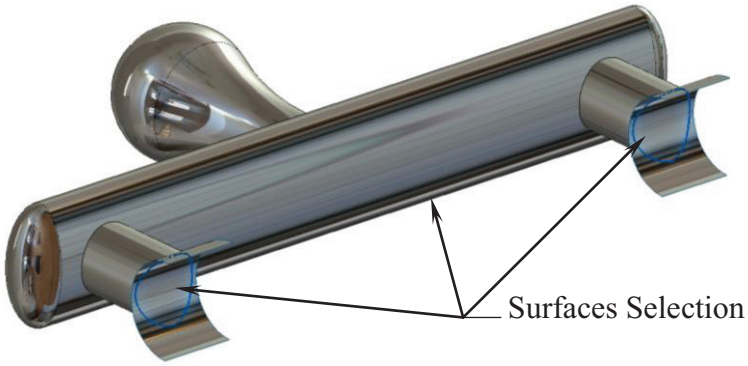
- Rotate the handle and locate the 2 surfaces under the handle as noted.

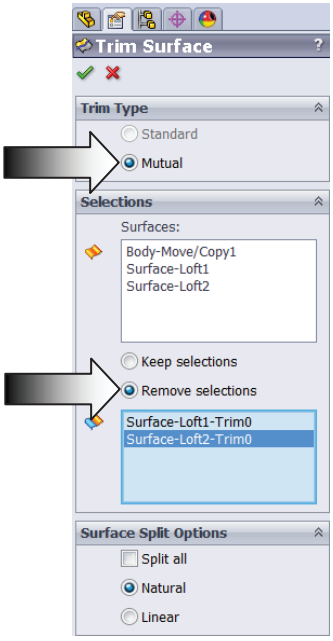
- Click the **Trim Surface** command from the Surfaces tool tab.

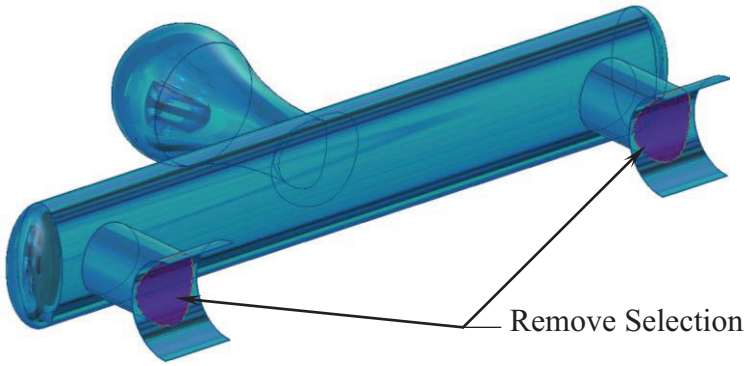


- Under the **Trim Type** selection box, click the **Mutual** option.

- Under **Trimming-Surfaces**, select the 2 trim surfaces and the Handle.







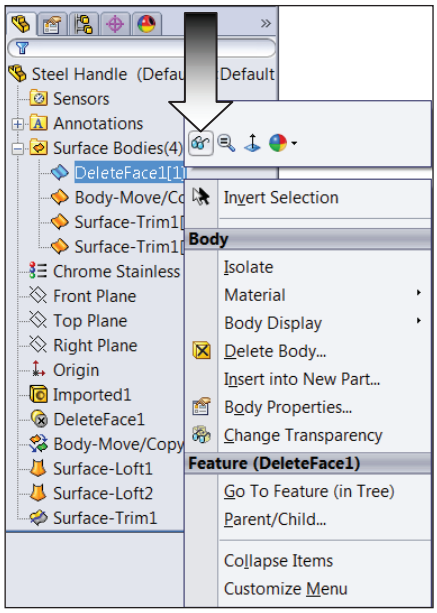
- Click the **Remove-Selection** button and select the 2 faces again.

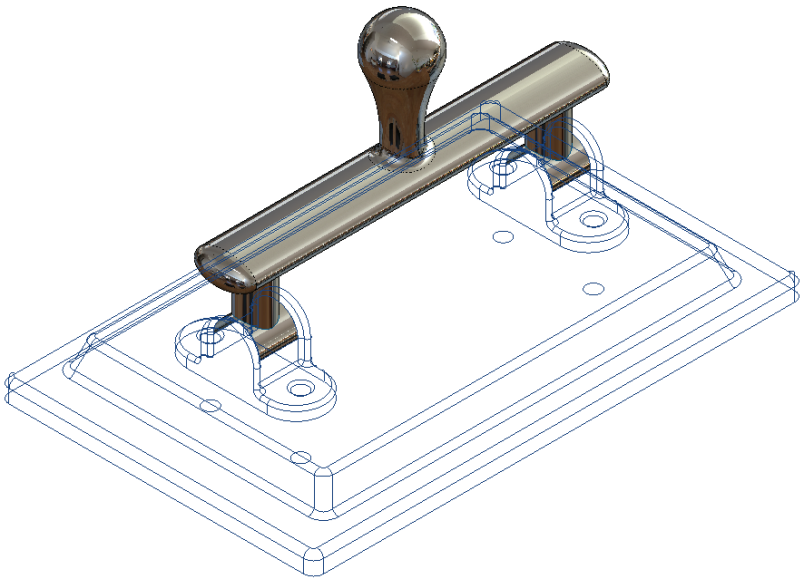
- Click **OK** to exit the Trim Surface.

8. Showing a surface body:

- After the trim we can now make the base body visible for the next step.

- Right click the Base body (DeleteFace1) and select **Show** .

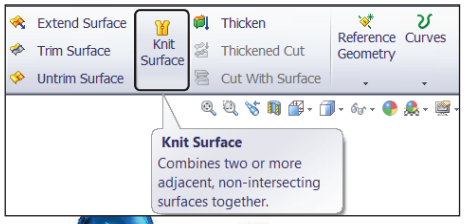




- The base body appears. We can now knit the base body to the other surface bodies, so that they can become 1 closed volume.

9. Knitting all surface bodies:

- Click the **Knit Surface** command from the Surfaces tool tab.

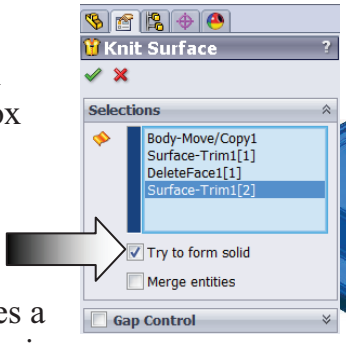


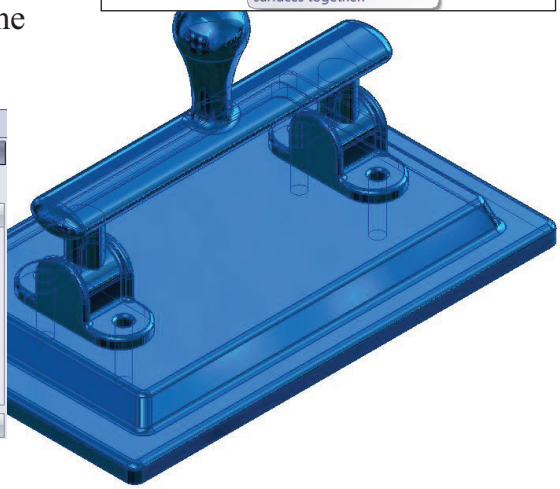
- Select <u>all surface bodies</u> either from the graphics area or from the Feature tree.

- Enable the **Try To Form Solid** checkbox

- Click **OK**.

- The surface model becomes a solid model again.

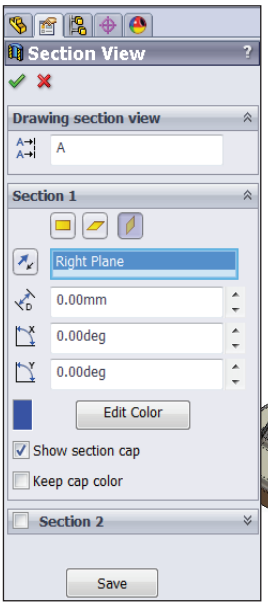




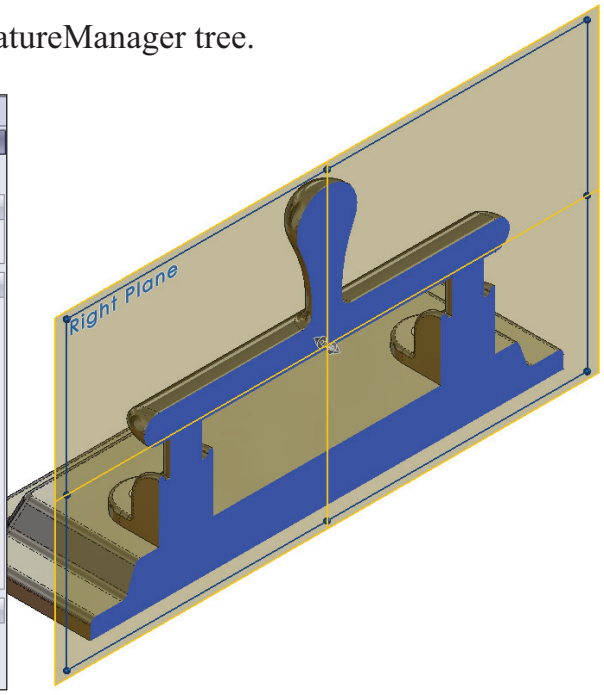
10. Creating a section view:

- Select the **Right** plane from the FeatureManager tree.

- Click the **Section View** command on the View Heads Up toolbar.



- Verify that the part is now a solid model.



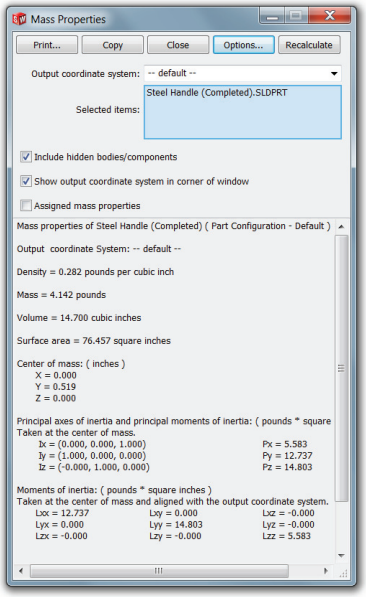
- Click **OK** to exit the section view command.

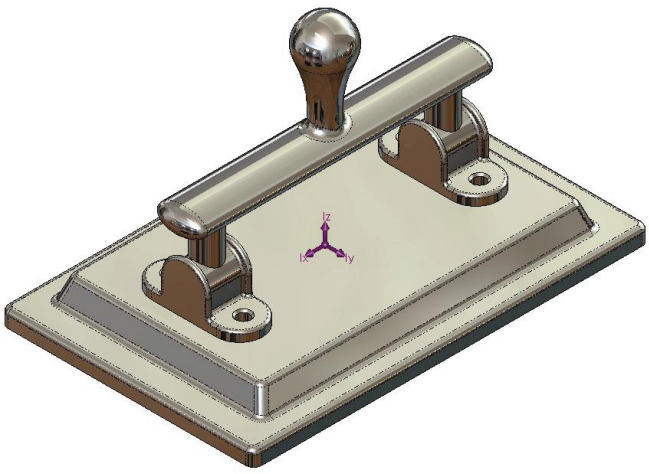
11. Measuring the mass:

- Change the material to **Chrome Stainless Steel**.

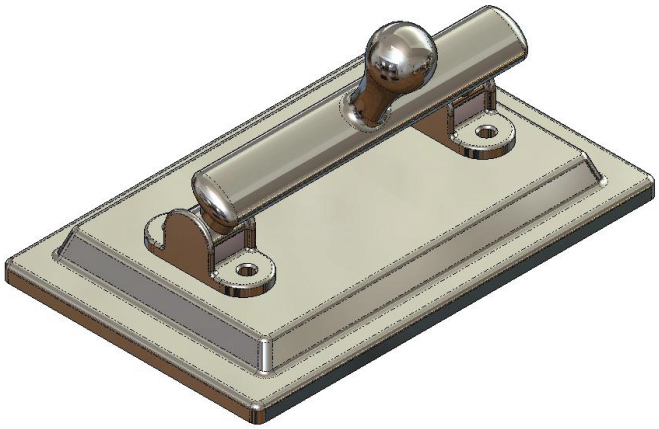
- Click the **Mass Properties** command from the Evaluate tool tab.

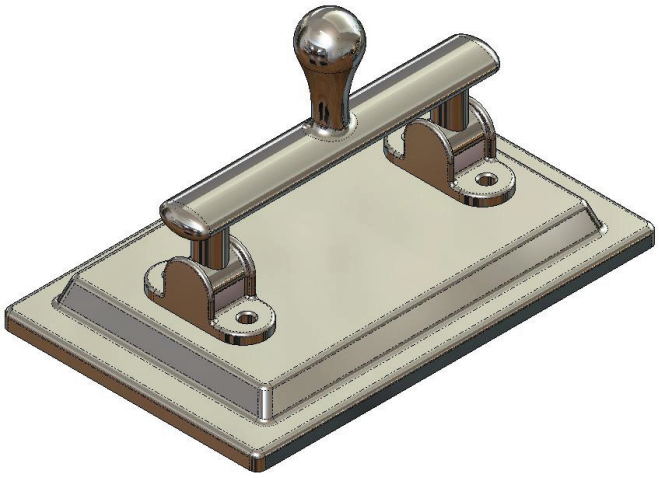
- Using 3 decimal places, enter the **final mass** of the model here: _____.

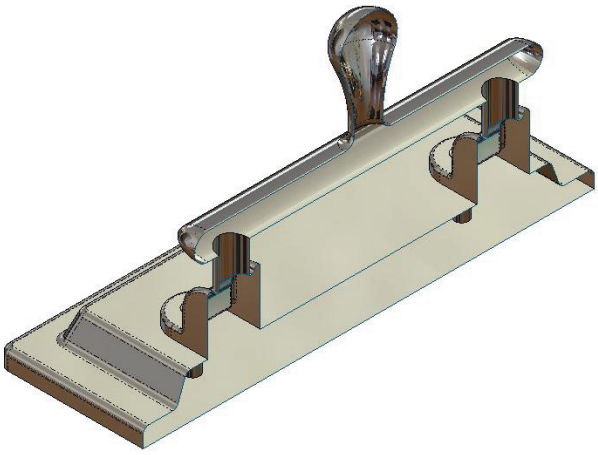


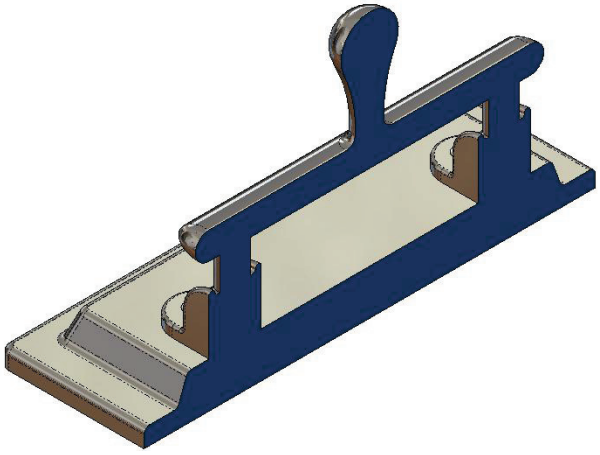


- Save the document as **Surface Modifications Completed** and close all documents.



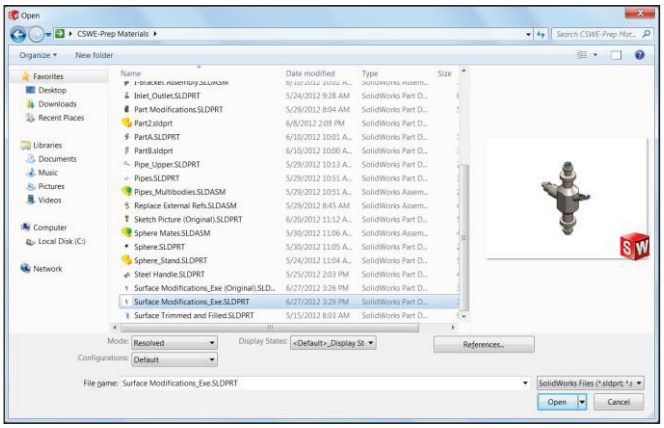






Exercise: Surface Modifications

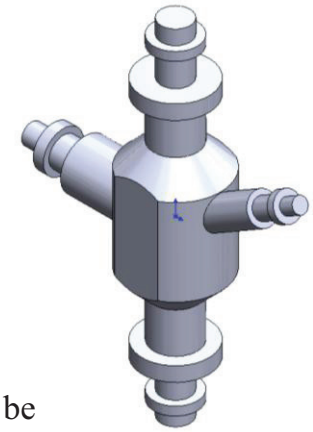
1. Opening a part document:



- Click **File / Open**.

- Browse to the Training CD locate and open the part named: **Surface Modifications_Exe**.

- This part was created as a solid model, and to make it more similar to the ones used in the CSWP-Surface and in the CSWE exams, it was saved under a format other than SolidWorks, therefore no feature history will be available for editing.

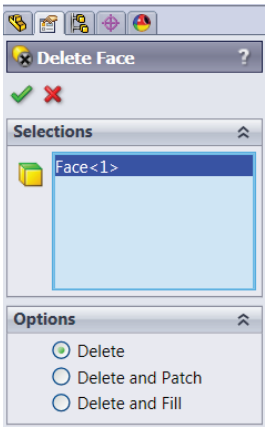


- An advanced user should know one or more different methods for working with these types of models.

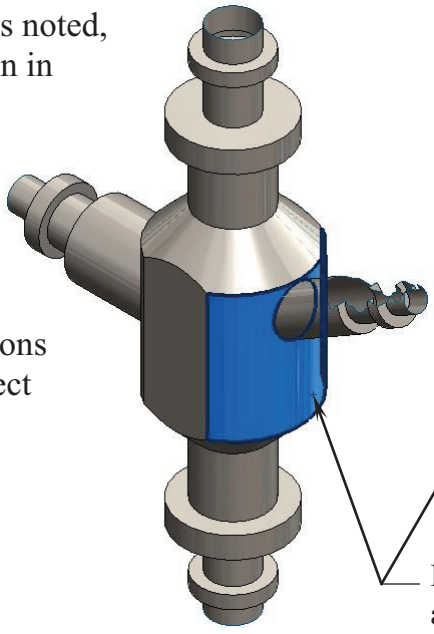
2. Converting to surface model:

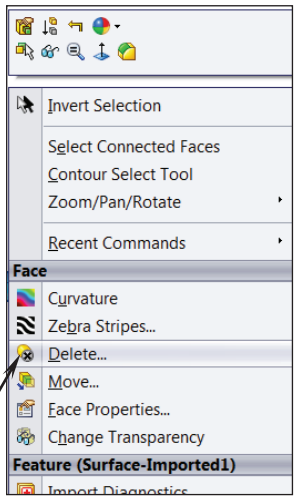
- By removing one of the faces of the solid model, it will be transformed into a surface model instantly.

- Right click the faces as noted, select the **Delete** option in the Face section.



- In the Options section select the **Delete** option.

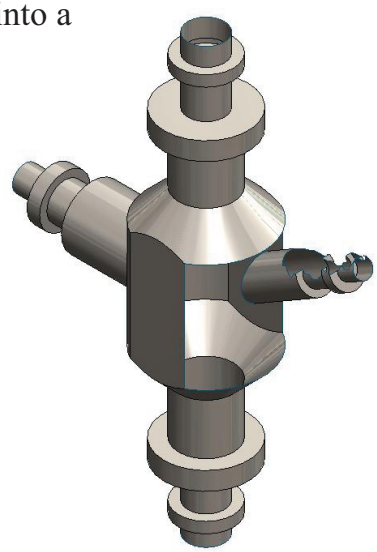




Right click this face and choose Delete

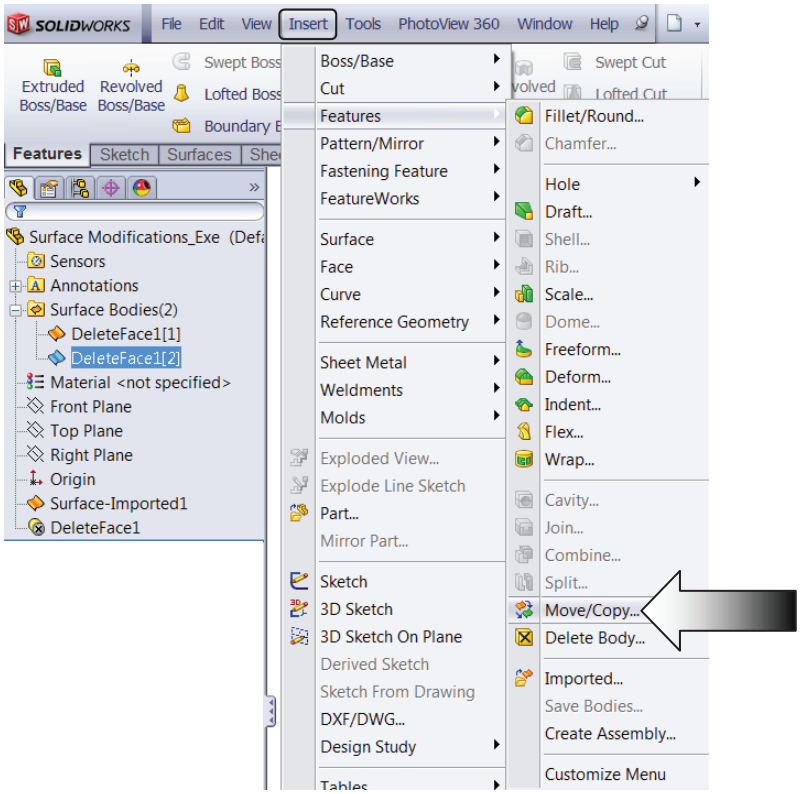
- Click **OK**.

- The selected face is removed and the part changes into a surface model. This model has 2 surface bodies.



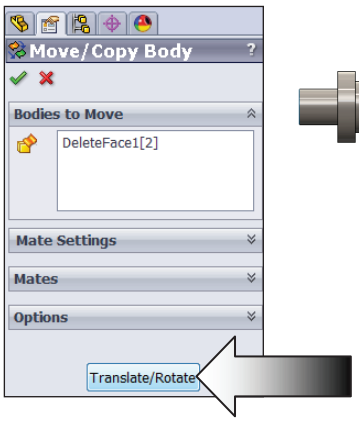
3. Rotating a surface body:

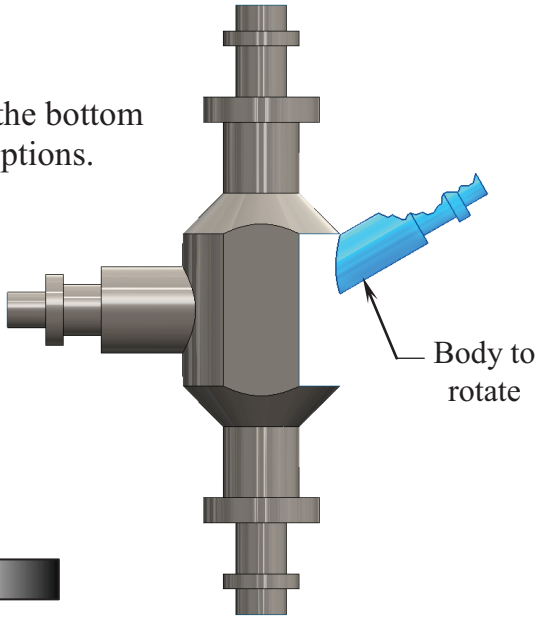
- Select **Insert / Features / Move/Copy**.



- Click the **Translate / Rotate** button at the bottom of the tree to change the move / rotate options.

- Click in the Bodies to Move box to activate it and select the surface body on the right as indicated.





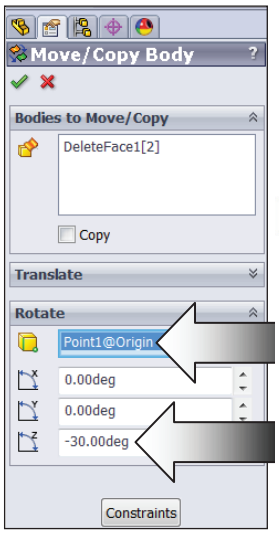
- More setting on the next page...

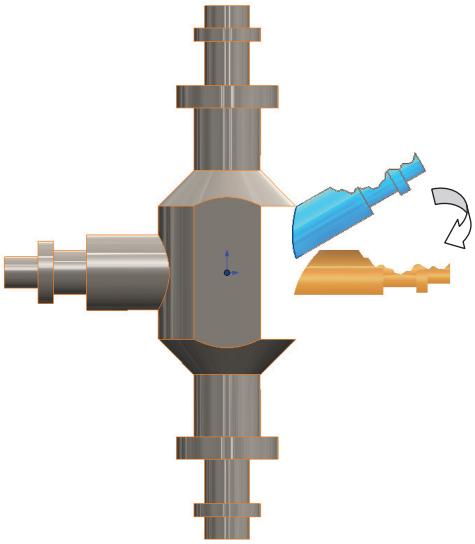
- Expand the **Rotate** section and select the Origin as the center of rotation.

- In the Z direction, enter **-30.00deg** for rotate angle (arrow).

- The preview shows the selected surface body is being rotated downwards.

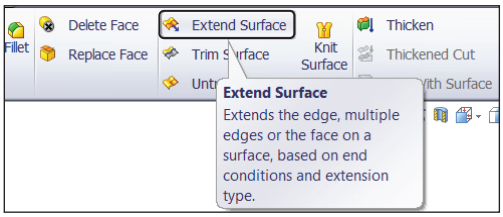
- Click **OK** to exit the Move/Rotate command.

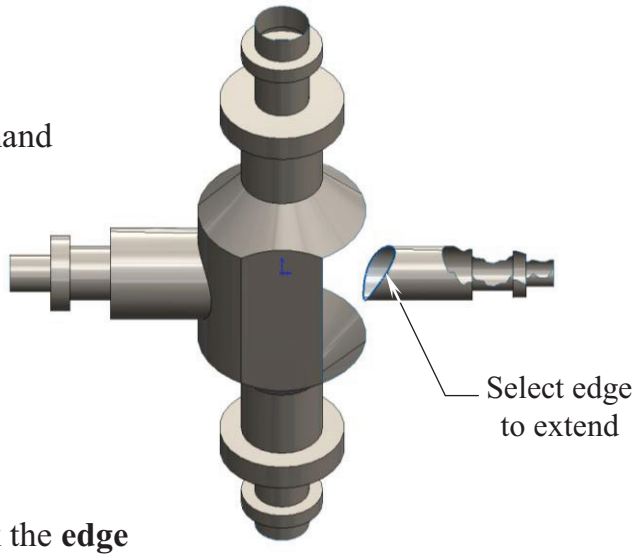




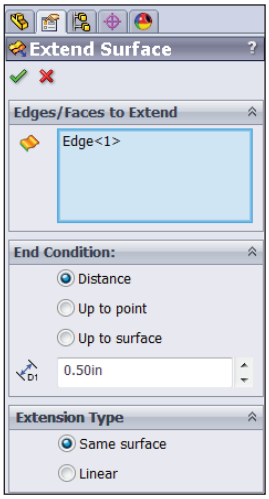
4. Extending a surface body:

- Click the **Extend Surface** command from the Surfaces toolbar.





- For Edges/Faces to Extend, click the **edge** of the surface body on the right as noted.

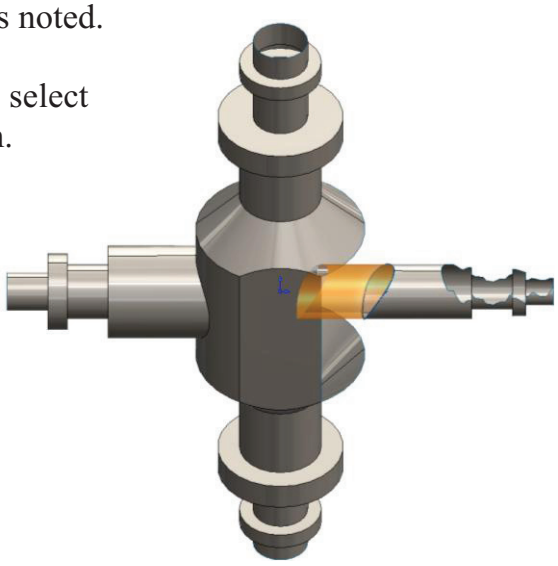


- For End Condition, select the **Distance** option.

- Enter **.500in** for distance.

- For Extension Type, select the **Same Surface** option.

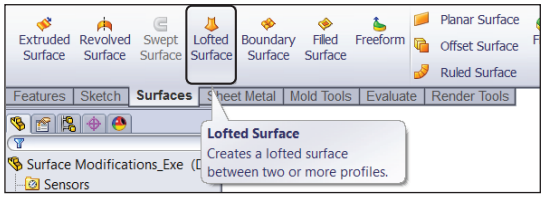
- Click **OK**.

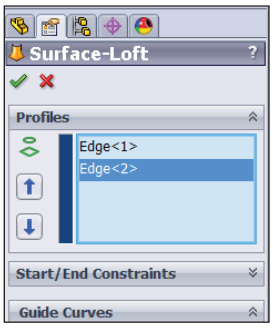


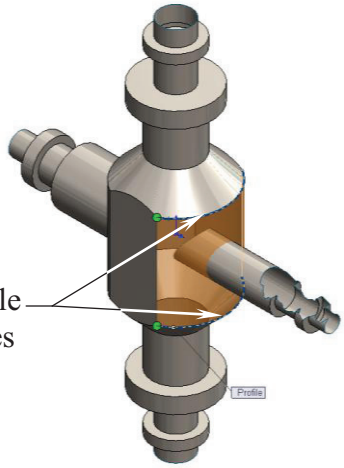
5. Creating a lofted surface:

- The surface that was deleted earlier will be recreated, using its own existing geometry.

- Click **Lofted Surface** on the Surfaces tool bar.







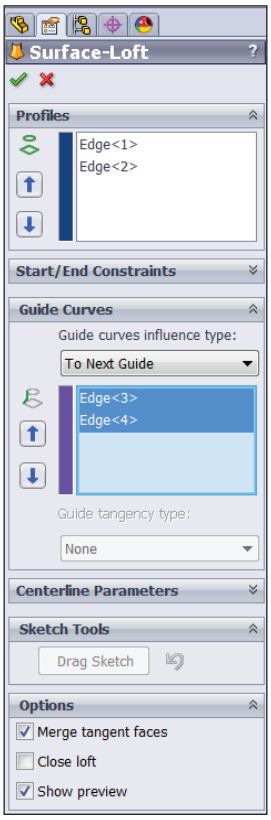
- For Loft Profiles select the 2 curved edges as noted.

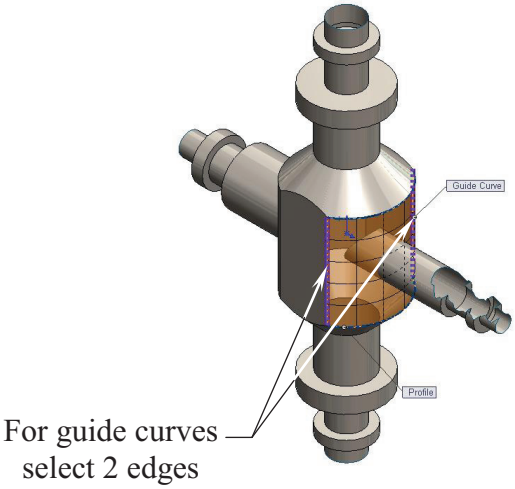
- Activate the Guide Curve section and select the 2 vertical edges as noted.

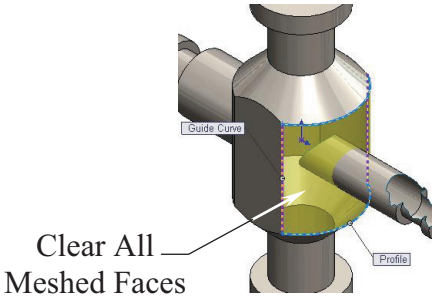
- The preview shows a new surface is being created.

- The grid lines on the preview surface is called the Mesh. To remove them, right click on this surface, go to Mesh Preview and select: **Clear All Meshed Faces**.

- Click **OK** to accept and exit the Lofted Surface command.



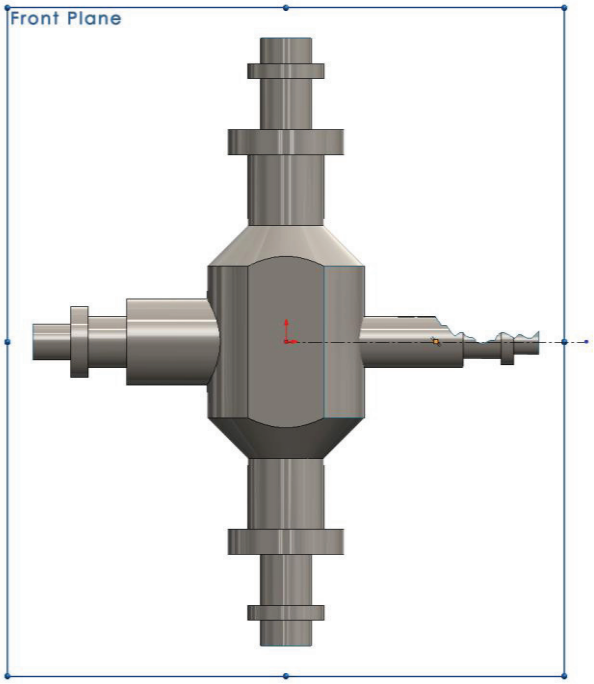




6. Repairing the broken surface body:

- The broken area can be repaired with a revolved surface. This repair involves a few steps: Convert the broken edges into a new sketch, remove its relations, revolve the sketch as a new surface, delete the broken area, and then knit the revolved surface to the surface part.

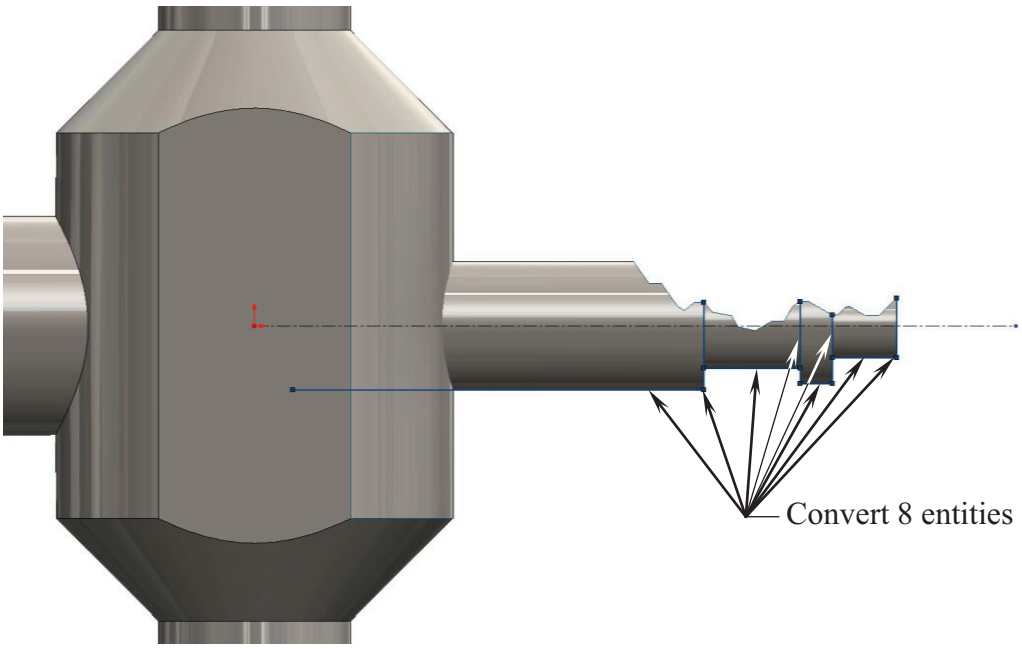
- Select the Front plane and open a new sketch.



- Sketch a horizontal centerline that will be used as the revolve line.

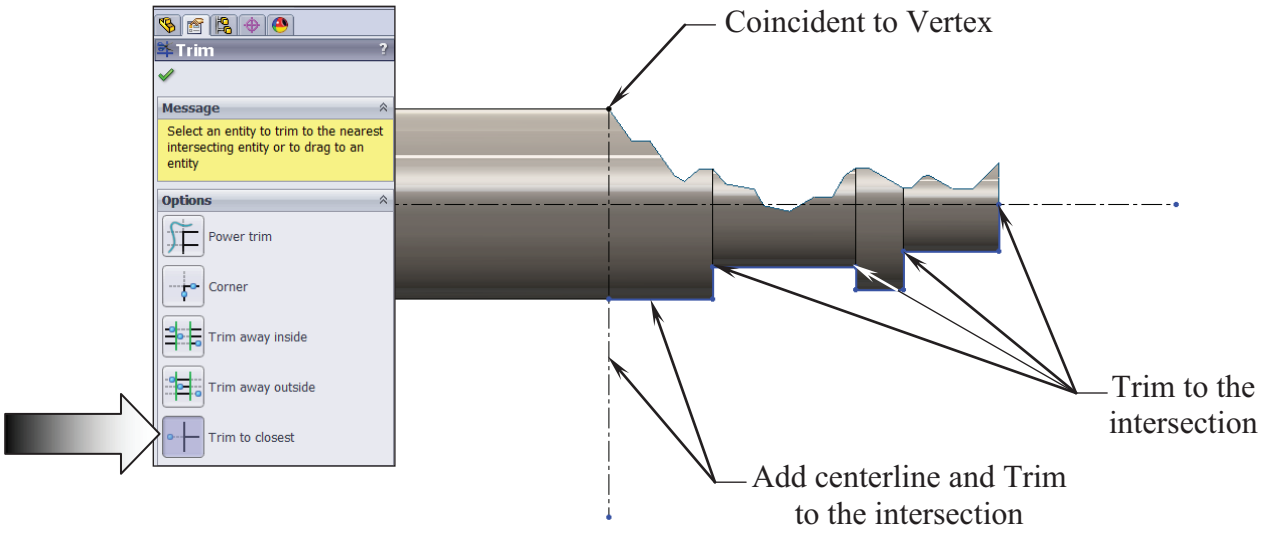
- Hold the Control key and select the **8 edges** of the broken area as indicated below.

- Some of the converted lines are crossing the centerline, this will cause an overlapping error when the revolving the sketch. Those lines need to trimmed in the next step.



- Select the **Trim Entities** command from the Sketch toolbar.

- Select the Trim To Closet option (arrow).

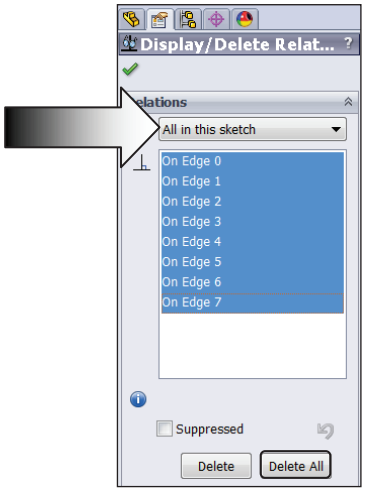


- Add a vertical centerline and use it to trim one of the lines.

- Click on the portions of the lines that need to be deleted. When completed, all of the lines must be connected continuously.

- When an entity is converted from a model edge, SolidWorks creates a relation called On-Edge to reference it. When the model edge is changed, the entity updates.

- There are 8 on edge relations that need to be removed, so that when the broken area is deleted, it will not cause the dangling errors.



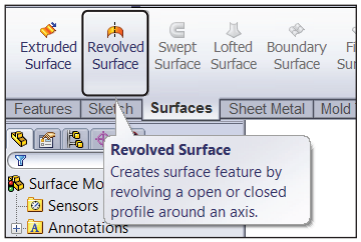
- Click the **Display/Delete Relations** command from the Sketch toolbar.

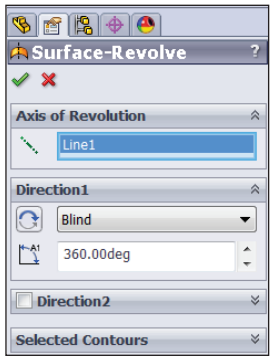
- Under the relation section, select the option **All In This Sketch** (arrow).

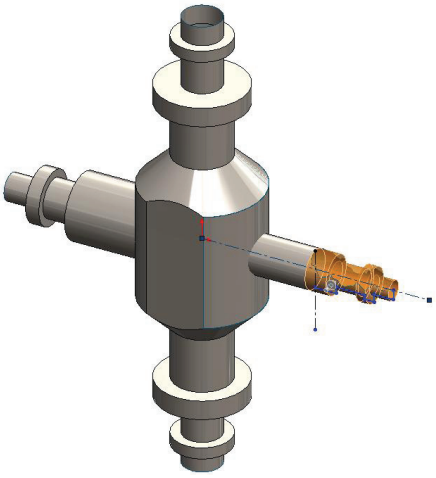
- Highlight all On-Edge relations and click **Delete All**.

- Click **OK** to exit the Display/Delete Relation command.

- Switch to the Surfaces toolbar and select the **Revolved Surface** command.







- Select the **horizontal centerline** as the Axis of Revolution.

- Use the default **360deg** revolve angle.

- Click **OK**.

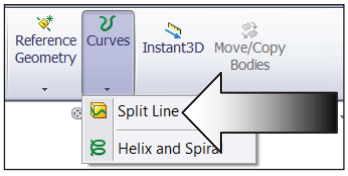
7. Creating a split line:

- The Split Line divides the surface body into 2 bodies, and the broken area can be deleted without affecting the other half .

- Select the Front plane and open a new sketch.

- Sketch a **vertical line** starting at the vertex shown.

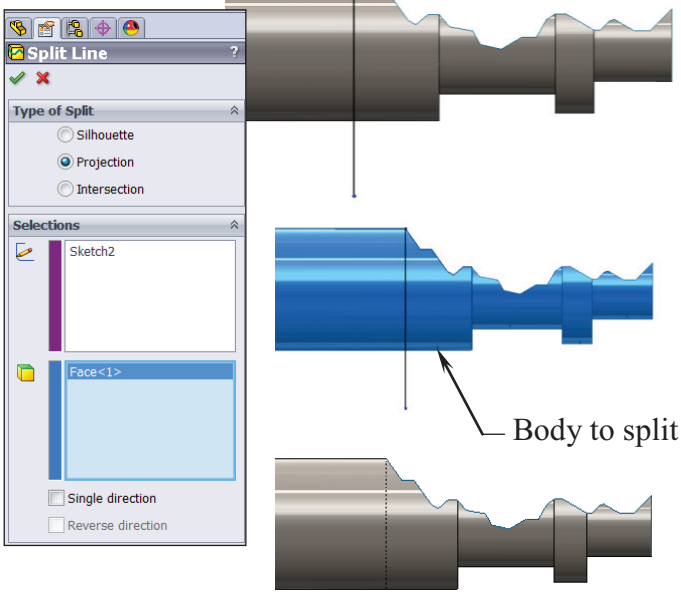
- Switch to the **Feature** toolbar and select: **Curves / Split Line**.



- Use the default **Projection** option

- Click the surface **body to split** as noted.

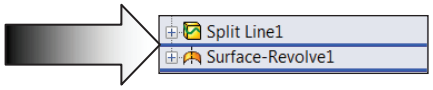
- Click **OK** to complete and exit the split command.

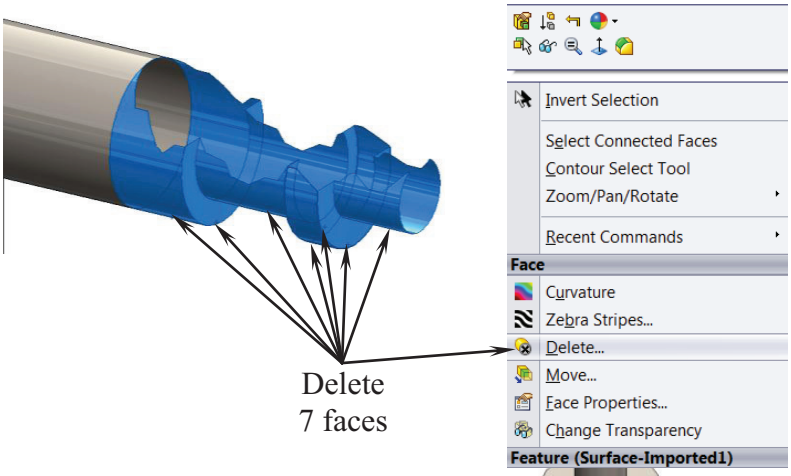


8. Deleting surfaces:

- The broken area can now be deleted.

- Drag the **Roll-Back** line up one step to temporarily hide the Revolved Surface.

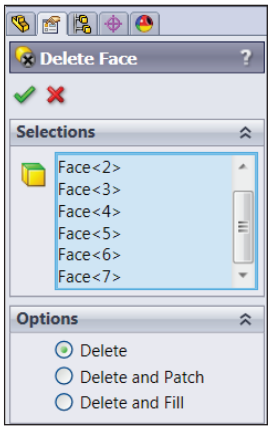


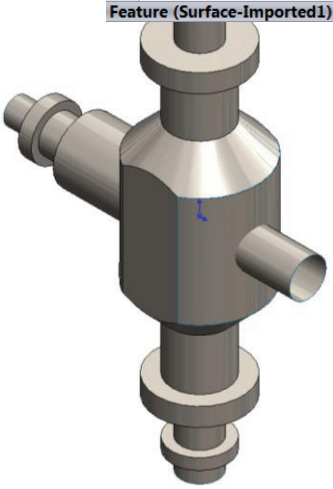


- Right click over one of the surfaces in the broken area and select the **Delete** option below the Face section.

- Select the **7 faces** to delete as indicated.

- Under the Options section click the **Delete** button.



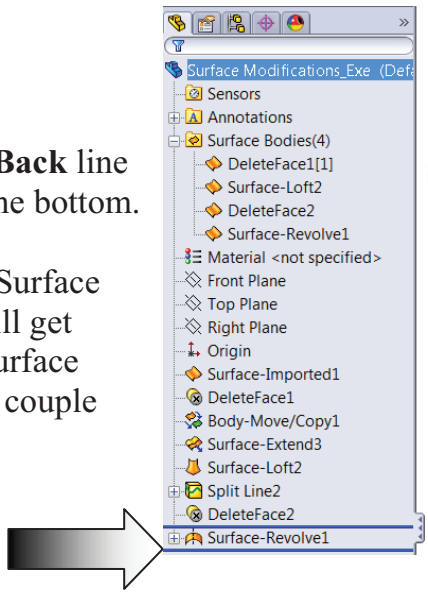


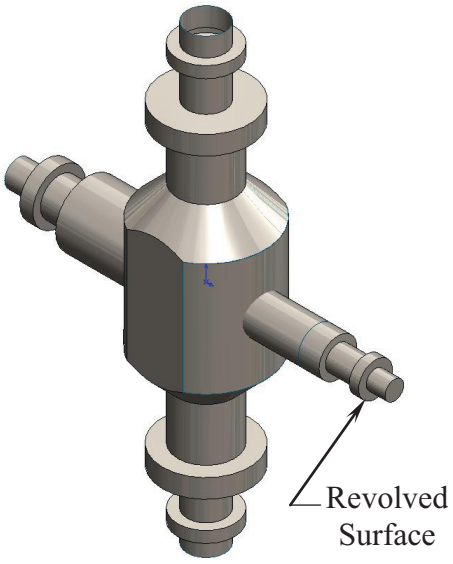
- Click **OK** to accept and exit the Delete Face command.

- The selected surfaces are deleted leaving a sharp clean edge that matches the edge of the Revolved Surface.

- Drag the **Roll-Back** line back down to the bottom.

- The Revolved Surface reappears. It will get knitted to the surface part in the next couple of steps.

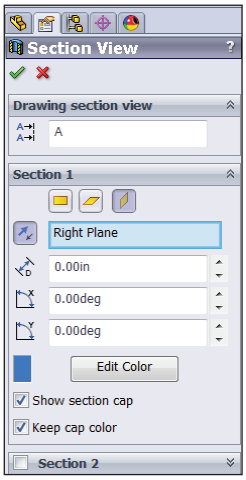


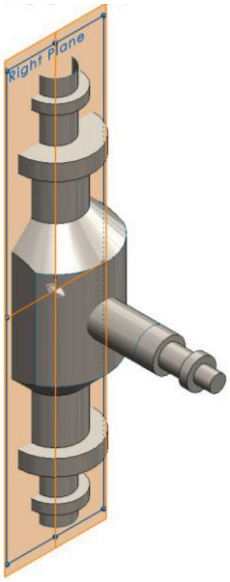


9. Creating a section view:

- There are some areas that still need trimming. The section view will help us see and work with them a little easier.

- Select the Right plane from the FeatureManager tree and click the **Section View** command on the View Heads Up toolbar.





- Rotate the view to see the left end of the Revolved Surface.

10. Trimming the surfaces:

- Change to the **Surfaces** tool tab.

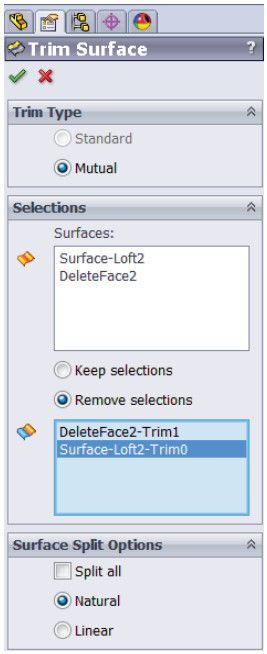
- Click the **Trim Surface** command.

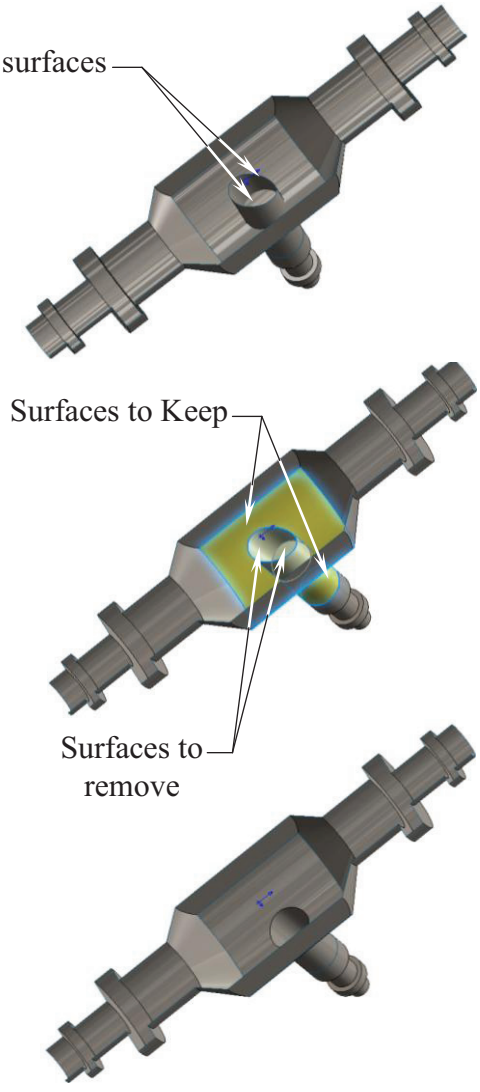
- Select **Mutual** under the Trim Type.

- For Trimming Surface select the **cylindrical body** and the **lofted surface** as indicated.

- Click the **Remove Selection** button and select the two surfaces as noted (the left end of the cylindrical body and the surface that intersects the lofted surface and the cylindrical body).

- Click **OK** and check your surface model against the image shown here.





11. Creating the planar surfaces:

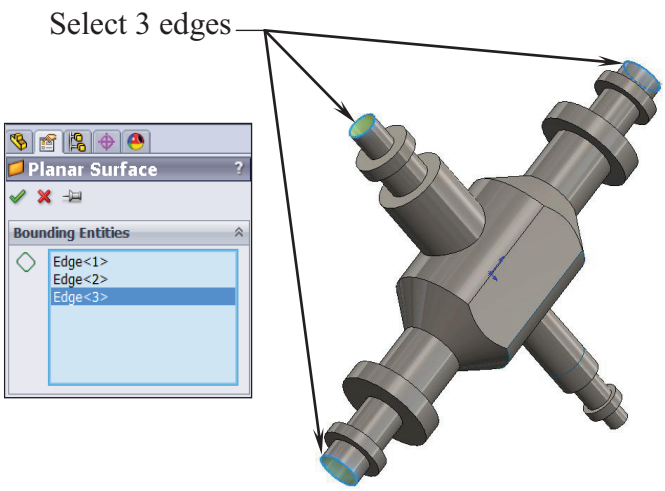
- Before the surface model can be turned back into a solid model, all openings should be closed off. One of the quick ways to do this is to use the Planar Surface command.

- Click the **Planar Surface** command from the Surfaces tool tab.

- Select the **3 circular edges** as indicated for Bounding Entities.

- The preview of the new surfaces appears, indicating that they are being created.

- Click **OK** to accept and exit the Planar Surface command.



12. Knitting all surfaces:

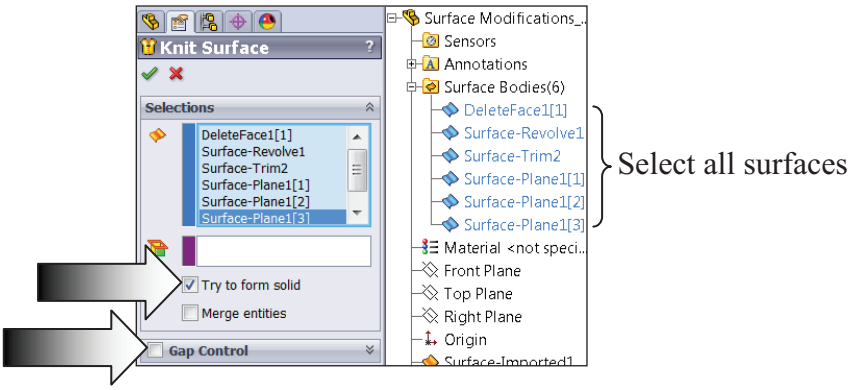
- After all surface bodies are knitted into a single body, it can be made into a solid model.

- Click the **Knit Surface** command from the Surfaces tool tab.

- Expand the Surface Bodies folder on the FeatureManager tree.

- Select all surfaces inside of this folder, there should be a total of 6 surfaces.

- Enable the **Try To Form Solid** checkbox and uncheck the Gap Control option.

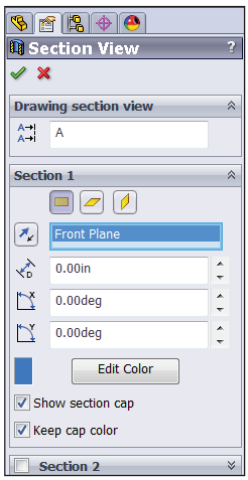


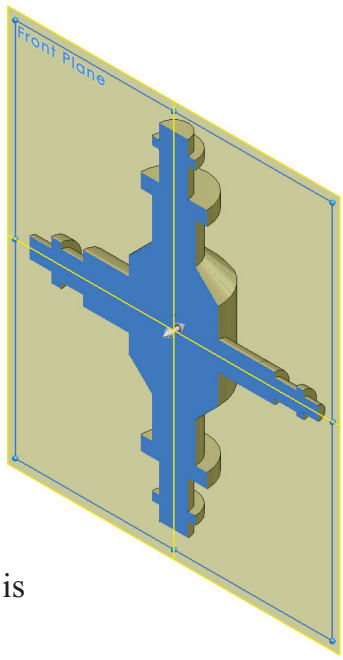
- Click **OK** to exit the Knit Surface command.

13. Verifying the solid model:

- Create a section view to see if the surface model has turned into a solid model.

- Select the Front plane from the FeatureManager tree and click the **Section-View** command.



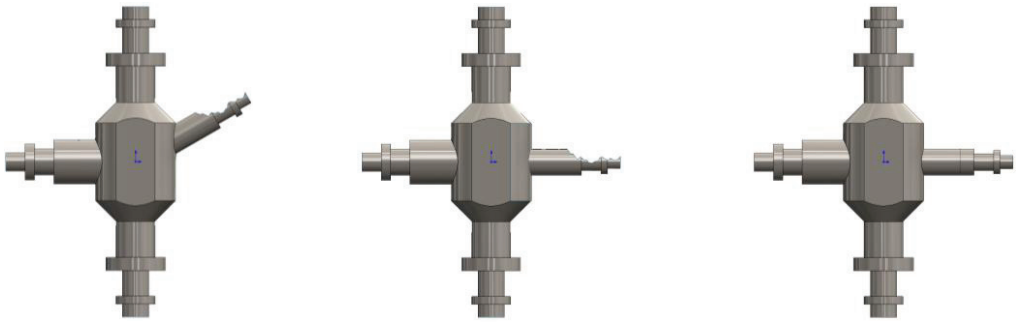
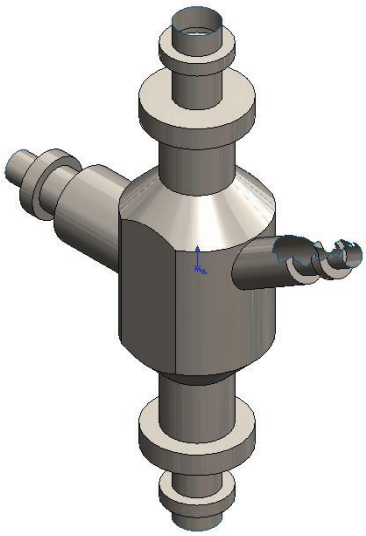
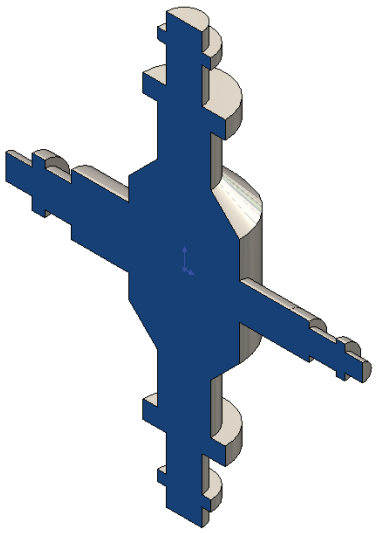
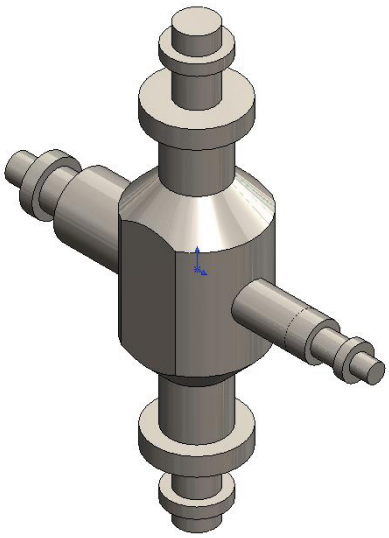


- The preview should show the model is a solid part at this point.

- Click **Cancel** to exit the Section View command.

14. Saving your work:

- Click **File / Save As**.

- Enter **Surface Modification_Exe** for the name of the file.

- Click **Save** and replace the existing file when prompted.