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SCHOOL OF MECHANICAL ENGINEERING

BMEE306P- CADFEA Lab



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Tutorial	Topics
2	Surface modelling, mutli solid bodies modelling, Rendering of product and repair of 3D models

Objectives:

- Usage of surface modelling to model plastic components
- Introduction to mold design using solid bodies
- Create drafting for the 3D models according to the manufacturing process

Tutorial :1 : Modelling of Nozzle surface

Surfaces Overview

Surfaces are a type of geometry with zero thickness. To create surfaces, you use many of the same methods used to create solids, such as extrudes, revolves, and sweeps. Surfaces also use other functions or features such as trim, untrim, extend, and knit. Surfaces have advantages over solids. They are more flexible than solids because you do not have to define the boundaries between the surfaces until the final steps of the design. This flexibility helps product designers work with smooth, extended curves such as those used in automobile fenders or telephone housings.

In this lesson, you start with an existing sketch composed of lines, arcs, splines, and sketch points. Then you apply the following surface features to create a nozzle:

- Loft
- Planar
- Trim
- Sweep
- Revolve
- Extend
- Knit
- Move/Copy
- Untrim
- Fill
- Thicken



Existing sketch Finished nozzle


Lofted Surface - Creating the Base

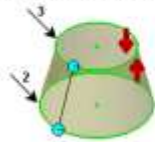
First, create the base for the nozzle using a surface loft between two arcs. Surface lofts include the same options as solid lofts. You can specify Start/End Tangency types, use Guide Curves, and so on.


1. [Click here](#)  to open Nozzle.sldprt, or browse to drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\examples\tutorial\surfaces\nozzle.sldp



For clarity, many images display only the sketches relevant to that procedure.

2. Click **File > Save As** and save the model as nozzle_01.sldpart.
3. Click **Lofted Surface**  on the Surfaces toolbar.
4. Select **Sketch2** and **Sketch3** for **Profiles**  in the PropertyManager.







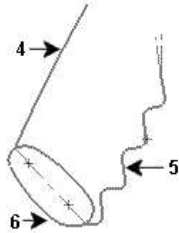
5. Under **Start/End Constraints**:
 - a. Select **Normal to Profile** in **Start constraint** and **End constraint**.
 - b. Set **Start Tangent Length** and **End Tangent Length** to 0.50.
6. Click **OK** .



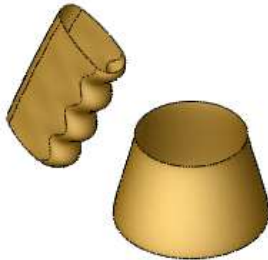
Swept Surface - Creating the Handle

With the **Swept Surface** tool, create the nozzle grip. To define the finger hold of the grip, include a guide curve in the surface sweep.

1. Click **Swept Surface**  on the Surfaces toolbar.
2. Select **Sketch6** for **Profile**  in the PropertyManager.
3. Select **Sketch4** for **Path** .
4. Under **Guide Curves**:
 - a. Select **Sketch5** for **Guide Curves** .
 - b. Select **Merge smooth faces**.



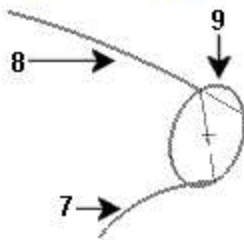
5. Under **Options**, clear **Merge tangent faces**.
6. Click **OK** .




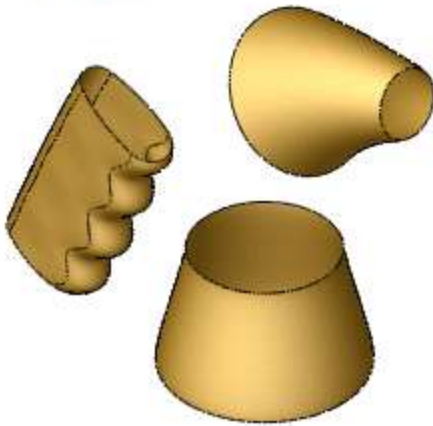
Swept Surface - Creating the Exit Nozzle

Create another swept surface for the exit nozzle.

1. Click **Swept Surface**  on the Surfaces toolbar.
2. Select **Sketch9** for **Profile**  in the PropertyManager.
3. Select **Sketch7** for **Path** .
4. Under **Guide Curves**:
 - a. Select **Sketch8** for **Guide Curves** .
 - b. Select **Merge smooth faces**.






5. Click **OK** .

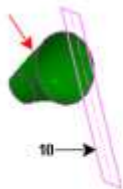



Split Lines - Dividing the Exit Nozzle

The **Split Line** tool divides a face into multiple faces. This allows you to connect the base, the grip, and the exit nozzle with surface lofts. First, split the exit nozzle.

1. Click **Split Line**  on the Mold Tools toolbar.
2. In the PropertyManager, under **Type of Split**, select **Projection**.
3. Under **Selections**:
 - a. Select **Sketch 10** for **Sketch to Project** .
 - b. Select the face of the exit nozzle for **Faces to Split** .

If necessary, rotate the exit nozzle to select the face.






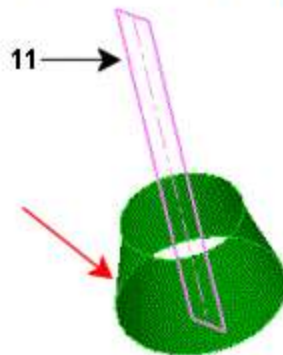
4. Click **OK** .




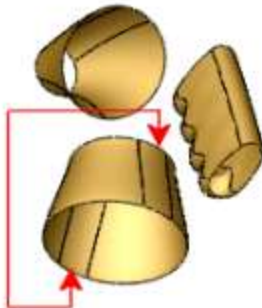
Split Lines - Dividing the Base

Next, split the base.

1. Click **Split Line**  on the Mold Tools toolbar.
2. In the PropertyManager, under **Type of Split**, select **Projection**.
3. Under **Selections**:
 - a. Select **Sketch 11** for **Sketch to Project** .
 - b. Click in **Faces to Split** , and select the face of the base.



4. Click **OK** .



Lofted Surface - Surface Bodies

Connect the three surface bodies using surface lofts. First, connect the exit nozzle to the grip.


1. Click **Lofted Surface**  on the Surfaces toolbar.
2. Select the top segment edge of the exit nozzle (created by the split feature), and the center segment edge of the grip for **Profile**  in the PropertyManager.



If the profile for the loft is twisted, adjust the connector.

See [Loft Synchronization](#)





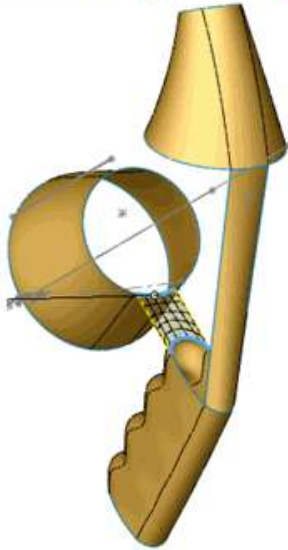
3. Under **Start/End Constraints**, select **Tangency To Face** for **Start constraint** and **End constraint**.
4. Under **Options**, select **Merge tangent faces**.
5. Click **OK** .



Connecting the Base to the Grip

Next, connect the base to the grip.

1. Click **Lofted Surface**  on the Surfaces toolbar.
2. Select the edges on the base and the grip for **Profile**  in the PropertyManager.




Check the preview. If the profile is twisted, adjust the connector.

See [Loft Synchronization](#)

3. Under **Start/End Constraints**:
 - a. Select **Tangency To Face** for **Start constraint** and **End constraint**.
 - b. Set **Start Tangent Length** to 3 and **End Tangent Length** to 7.

Switch the start and end lengths as required, to apply the value of 7 to the side near the grip.



4. Under **Options**, select **Merge tangent faces**.

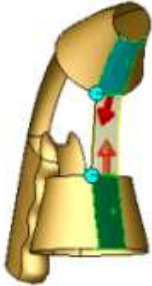
5. Click **OK** .




Connecting the Base to the Nozzle

Finally, connect the base to the exit nozzle.

1. Click **Lofted Surface**  on the Surfaces toolbar.
2. Select the edges on the base and the exit nozzle for **Profiles**  in the PropertyManager.



If the profile is twisted, adjust the connector.
See [Loft Synchronization](#)





3. Under **Start/End Constraints**, select **Tangency To Face** for **Start constraint** and **End constraint**.
4. Under **Options**, select **Merge tangent faces**.
5. Click **OK** .



6. Save the model.

Knit Surface - Joining the Base Entities

Join the surfaces you created with lofts and sweeps using the knit command. Knitting surfaces combines two or more adjacent surface bodies into one.

1. Click **Knit Surface**  on the Surfaces toolbar.
2. Expand **Surface Bodies**  in the FeatureManager design tree.
3. Select all the surface bodies in the folder for **Surfaces and Faces to Knit** .
4. Click **OK** .

The **Surface Bodies**  folder now holds a single surface body.

Knit Surface does not change the appearance of the model.

Filled Surface - Enclosing an Open Area

Fill each side of the area enclosed between the base, the grip, and the exit nozzle using the **Filled Surface** tool. To manipulate the curvature of the surface, use a sketch point to constrain the curve. **Constraint Curves** allow you to add slope control to the path.

1. Click **Back**  to display the model in profile.
2. Click **Filled Surface**  on the Surfaces toolbar.
3. Select an edge, right-click and choose **Select Open Loop for Patch Boundaries**  in the PropertyManager.



Select Open Loop finds all the edges in a closed loop, creating the surface fill.






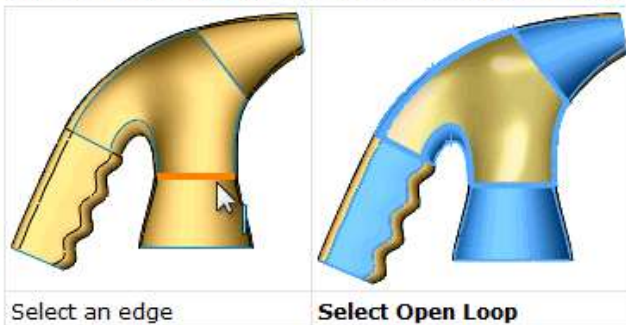
4. Under **Edge settings**:
 - a. Select **Tangent** in **Curvature Control**.
 - b. Select **Apply to all edges**.

About Patch Boundaries
5. Under **Curvature Display**, clear **Mesh preview**.
6. Click in **Constraint Curves** , and select **Pull Point1**  in the Rysut FeatureManager design tree.
7. Under **Options**, select **Fix up boundary**.
8. Click .



Completing the Filled Surface

1. Click **Front**  to display the model in profile.
2. Click **Filled Surface**  on the Surfaces toolbar.
3. Select an edge, right-click and choose **Select Open Loop** for **Patch Boundaries**  in the PropertyManager.





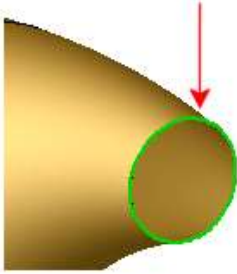
4. Under **Edge settings**:
 - a. Select **Tangent** in **Curvature Control**.
 - b. Select **Apply to all edges**.[About Patch Boundaries](#)
5. Click in **Constraint Curves** , and in the flyout FeatureManager design tree, select **Pull Point2** .
6. Under **Options**, select **Merge result**.
7. Click **OK** .





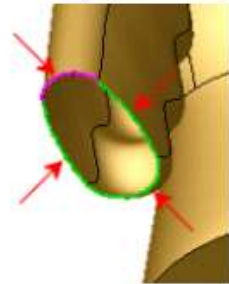
Planar Surface


Apply a planar surface to close the openings on the grip and the exit nozzle.

1. Click **Planar Surface**  (Surfaces toolbar).
2. Select the edges on the exit nozzle for **Bounding Entities**  in the PropertyManager.



3. Click **OK** .
4. Click **Planar Surface**  (Surfaces toolbar).
5. In the graphics area, right-click an edge on the grip and choose **Select Open Loop**.








The four sketch entities on the grip are selected for **Bounding Entities**  in the PropertyManager.

6. Click **OK** .

Knit Surface - Joining the New Entities

Knit all the surfaces into a single entity.

1. Click **Knit Surface**  on the Surfaces toolbar.
2. Expand **Surface Bodies**  in the FeatureManager design tree.
3. Select all the surface bodies in the folder for **Surfaces and Faces to Knit** .
4. Click **OK** .

The **Surface Bodies**  folder now holds a single surface body.

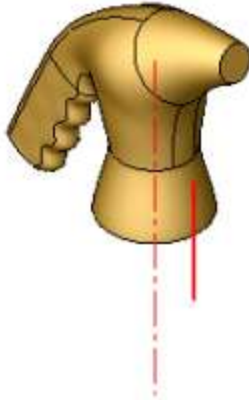


5. Save the model.

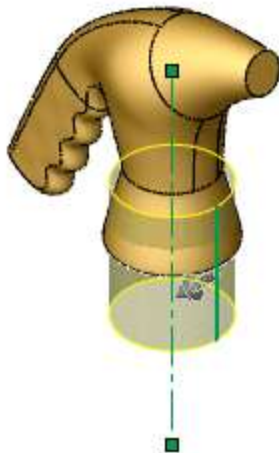
Revolved Surface


Use the **Revolved Surface** tool to create a surface that extends the nozzle base.

1. Select **Sketch13** in the FeatureManager design tree.



2. Click **Revolved Surface**  on the Surfaces toolbar.
3. In the PropertyManager, under **Direction 1**:
 - a. Select **Blind** in **Revolve Type**.
 - b. Set **Angle**  to 360.



4. Click **OK** .

Move/Copy Bodies - Moving a Surface

Move the revolved surface, and position it below the existing nozzle base with the **Move/Copy Bodies** tool. This tool moves, rotates, or copies bodies and surfaces, and places the bodies in any position using coordinates.


1. Click **Move/Copy Bodies**  on the Features toolbar.
2. Expand **Surface Bodies**  in the FeatureManager design tree.
3. Select **Surface-Revolve1** for **Solid and Surface or Graphic Bodies to Move/Copy**  in the PropertyManager.



4. Clear **Copy**.

To display the **Copy** option and the **Translate** properties, you might have to click **Translate/Rotate**.



5. Under **Translate**, set ΔY to -6.35 to move the surface body down.

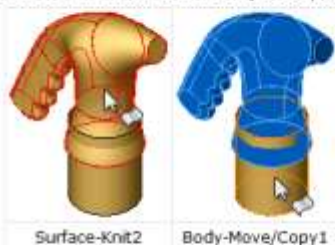
6. Click **OK** .




Trim Surface - Removing Surfaces

Use the **Mutual** option of the **Trim Surface** tool to remove extraneous faces. The **Mutual** option uses multiple surfaces as mutual trim tools.

1. Click **Trim Surface**  on the Surfaces toolbar.
2. In the PropertyManager, under **Trim Type**, select **Mutual**.
3. Select **Surface-Knit2** and **Body-Move/Copy1** in the graphics area for **Trimming Surfaces** .



4. Select **Remove** selections.
5. Click **Pieces to Remove**  and select the faces shown.




You can select the faces for **Pieces to Remove** in any order. The list that appears in **Pieces to Remove** is based on your selection order, not on the entity you select.

6. Click **OK** .

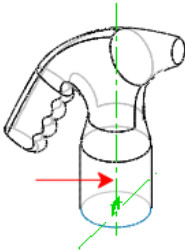


Extruded Surface - Creating a Trim Tool

With the **Extruded Surface** tool, create a trim tool at the base of the nozzle. Trimming the surface creates the first notch at the nozzle base.

For clarity, switch the display to **Hidden Lines Visible** .


1. Select **Sketch14**.

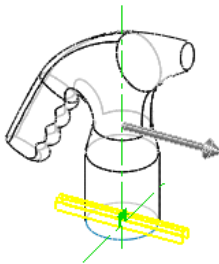



2. Click **Extruded Surface**  on the Surfaces toolbar.

3. Under **Direction 1**:

- a. Select **Mid Plane** in **End Condition**.







- b. Set **Depth**  to 140.



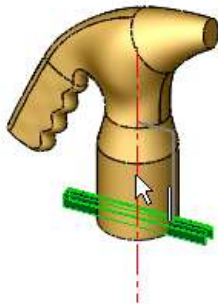
4. Click **OK** .

Move/Copy Bodies - Copying a Body


To create a second, intersecting trim tool, move and copy the surface extrude you created in the previous step.

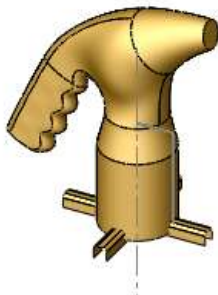
1. Click **Shaded with Edges**  (View toolbar).
2. Click **Move/Copy Bodies**  (Features toolbar).
3. In the FeatureManager design tree, expand **Surface Bodies** , and select **Surface-Extrude1**.
4. Under **Bodies to Move/Copy**:
 - a. Select **Copy**.
 - b. Set **Number of Copies**  to 1.
5. Expand **Surface-Revolve1** in the FeatureManager design tree, right-click **Sketch13**, and select **Show** .
6. Under **Rotate**, click **Rotation Reference** 
 - a. Select **Line1@Sketch13** in the graphics area for **Rotation Reference**.

Line1 is the axis used with Sketch13 to create the surface revolve.





- b. Set **Angle**  to 90.

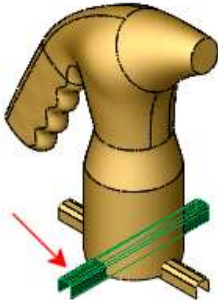
7. Click **OK** .



Trim Surface - Creating Cuts - Set 1

Create the first of two cuts at the base of the nozzle with the **Trim Surface** tool.

1. Click **Trim Surface**  on the Surfaces toolbar.
2. In the PropertyManager, under **Trim Type**, select **Standard**.
3. Under **Selections**:
 - a. Select **Body-Move/Copy2** in the graphics area for **Trimming Surface, Plane, or Sketch** .



- b. Select **Keep selections**.
- c. Select the trimmed surface **Surface-Trim1-Trim1** in the graphics area for **Pieces to Keep** .






4. Click **OK** .
5. Under **Surface-Bodies**  in the FeatureManager design tree, click **Body-Move/Copy2**, and select **Hide** .



Trim Surface - Creating Cuts - Set 2

Create the second of two cuts at the base of the nozzle with the **Trim Surface** tool.

1. Click **Trim Surface**  on the Surfaces toolbar.
2. In the PropertyManager, under **Trim Type**, select **Standard**.
3. Under **Selections**:
 - a. Select **Surface-Extrude1** in the graphics area for **Trimming Surface, Plane, or Sketch** .
 - b. Select **Keep selections**.
 - c. Select the main surface body in the graphics area for **Pieces to Keep** .



4. Click **OK** .
5. Under **Surface-Bodies**  in the FeatureManager design tree, click **Surface-Extrude1**, and select **Hide** .




Delete/Keep Body - Deleting Trim Tools

Delete the surface extrude and the surface body created with the move copy tool. These entities were used to trim the model and need to be removed for the final thicken surface operation.

1. Click **Delete/Keep Body**  on the Features toolbar.
2. Under **Type**, select **Delete Bodies**.
3. Expand **Surface-Bodies**  in the FeatureManager design tree.
4. Select **Surface-Extrude1** and **Body-Move/Copy2** in the folder for **Solid/Surface Bodies to Delete/Keep** .

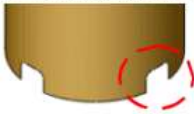




5. Click **OK** .



Untrim Surface - Patching a Surface


To strengthen the base of the model, use the **Untrim Surface** tool to patch one of the surface cuts.

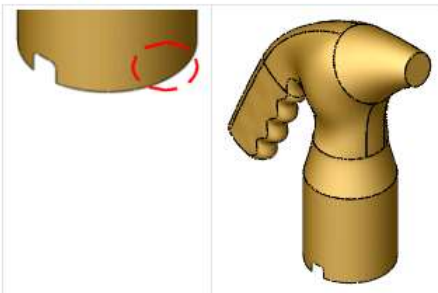


1. Click **Untrim Surface**  on the Surfaces toolbar.
2. Select **Edge1** in the graphics area for **Selected Face/Edges**  in the PropertyManager.








The **Untrim Surface** tool extends an existing surface along its natural boundaries, so you can select any edge from **Surface-Trim3**.

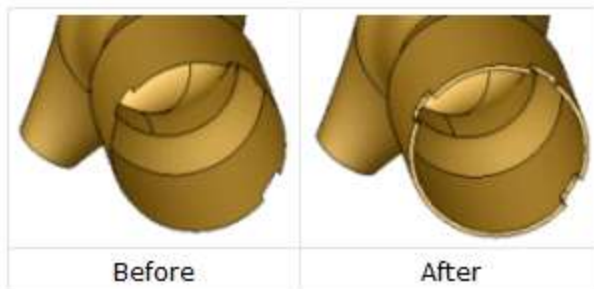
3. Under **Options**:
 - a. Select **Extend edges**.
 - b. Select **Merge with original**.
4. Click **OK** .



Thicken Surface - Creating a Solid

Thicken the surface model to create a solid model.

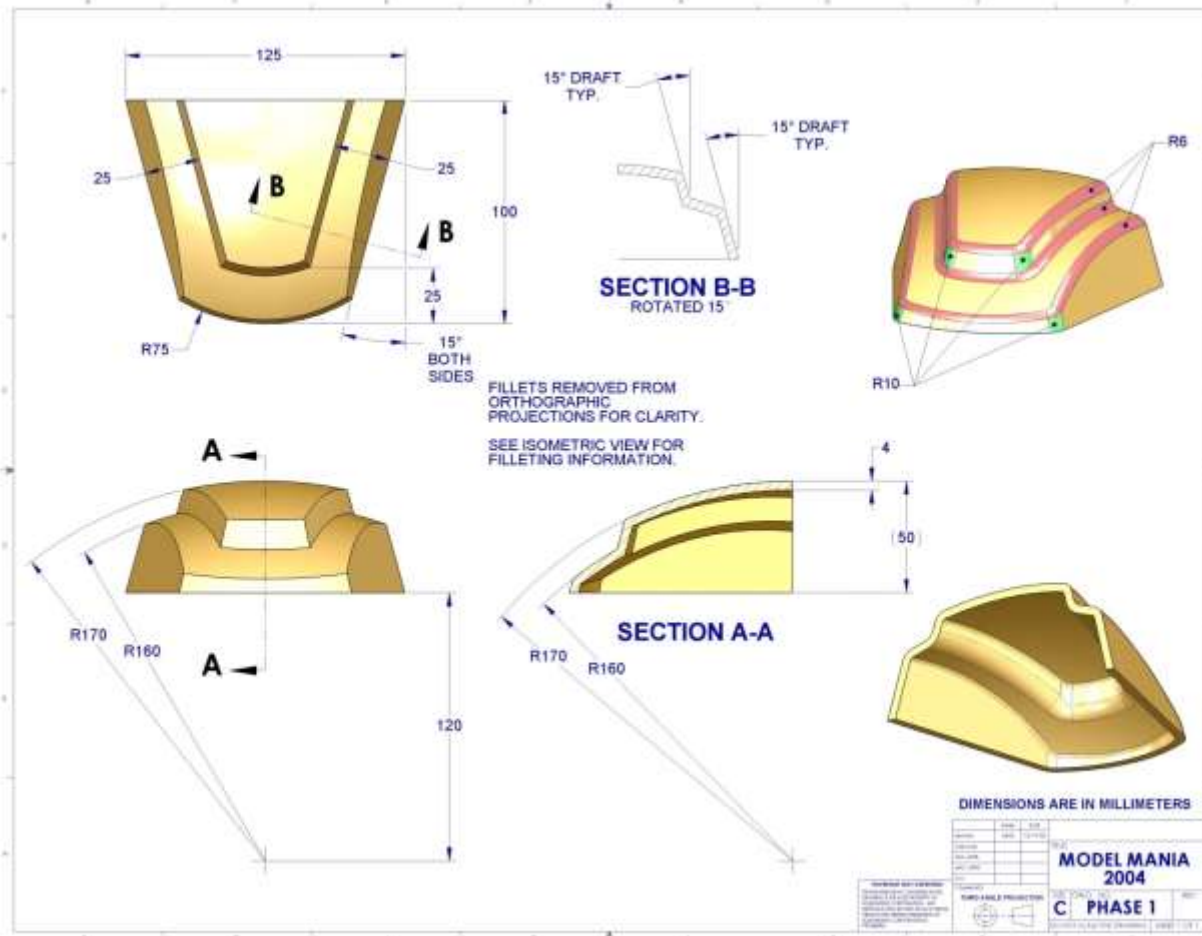
1. Click **Thicken**  on the Features toolbar.
2. In the PropertyManager, under **Thicken Parameters**:
 - a. Select **Surface-Untrim1** for **Surface to Thicken** .
 - b. Click **Thicken Side 1** .
 - c. Set **Thickness**  to 0.5.
3. Click **OK** .



4. Save the model.

Expected Output : The drafting should have views of the surface model that could be useful for downstream application in CFD or FEA or CAM.

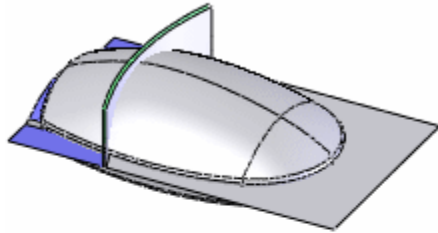
Sample drafting images:



Tutorial – 2 Introduction to design molded product design using solid bodies

Molded Product Design - Advanced

This advanced tutorial describes recommended techniques for designing stylized molded products with multiple components.



You might want to design molded products but do not know the best way to break up the master part into its various molded housings. Maybe it is unclear what the correct tools and methods are in SOLIDWORKS for deriving parts to their individual housings and for maintaining associativity.

By using the techniques recommended in this tutorial, your molded parts retain associativity between each other and should experience fewer downstream problems.

This tutorial teaches you how to build an assembly of a computer mouse from a master part, using three different techniques:

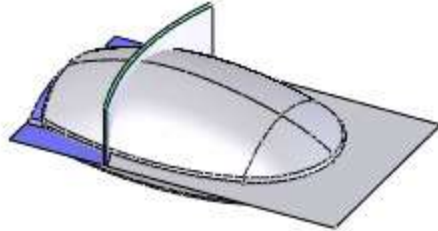
- Base part - Use the **Insert Part** command to derive parts
- Split part - Use the **Split** command to split out parts
- Multibody - Use the **Save Bodies** command to derive parts

These techniques are not a replacement for using assemblies. Because the master part is a part document, limitations exist such as no mates between components. However, you can still rotate or move the bodies using the **Move/Copy** command.

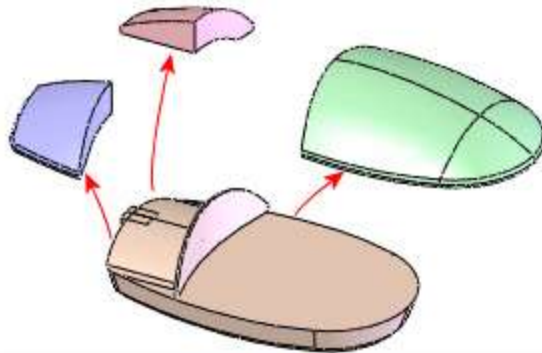
Design Workflow

The design workflow is similar for the three techniques.

Create a master part



Break out individual parts or bodies from the master part. In this tutorial, you break out four parts: right and left mouse buttons plus an upper and lower housing.

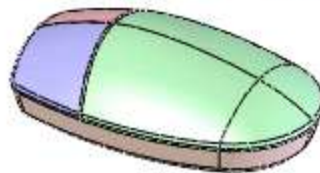


Add features to the individual parts (Not included in this tutorial)

For example, you could add a shell feature to the lower housing



Create an assembly from the individual parts



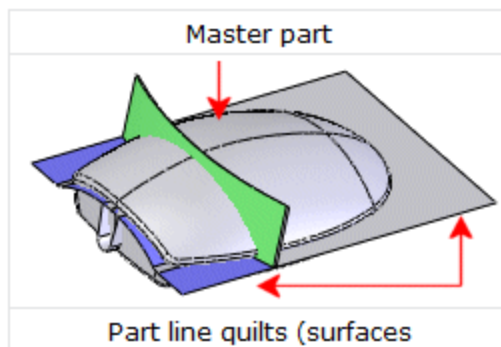
Creating the Master Part

You need to create the master part before you split out the individual components with these techniques. Create the master part with the design intent for the final product.

Follow these rules:

- The master part must contain all items that share a common reference
- Include only design details in the master part for parts that share a common reference

For the base part and split part techniques, you add surfaces to the model that serve as part line quilts (surfaces) to divide the master part into individual parts. One of the master parts used in these lessons, *MouseBase.sldprt*, is shown below.



The intent of this tutorial is to demonstrate techniques to break up the master model into its various molded components without breaking associativity. The actual building of the master part is not included in this tutorial.

What Technique Do I Use?

SOLIDWORKS recommends using these techniques as follows:

Design Team	Recommended Technique
<p>Single designer</p> <p>When a single designer works on a model, use the multibody technique because all the references for the bodies are contained within a single part file with no need to derive or reference additional files.</p>	<p>Multibody Technique</p>
<p>Multiple designers</p> <p>For large design groups where work needs to be divided among multiple designers, the best solution is to use the base part or split part technique.</p> <p>The lead designer or project manager is responsible for:</p> <ul style="list-style-type: none"> • Splitting out parts so that different designers can work on individual parts. • Bringing the individual parts back into the master part or an assembly. 	<p>Base Part Technique. The entire solid, and surfaces if you choose, are brought into the derived parts. The author of the master part must inform the designers of the derived parts about what references they must use.</p> <p>Split Part Technique. The author of the master part: Controls segmentation of the master part into individual parts. Controls the references that the designers of the individual parts see. The designers of the derived parts do not need to worry about what references they use.</p>

SOLIDWORKS strongly recommends these techniques instead of an in-context assembly for these reasons:

- Performance.

With an in-context assembly, it would take 10 times longer or more to rebuild because every change made in every part requires a complete rebuild of every part, whether it has a change at the part level or not.

- Design Intent and Best Practice for Design.

The three techniques follow a clearly-established master part methodology in which the master part captures the design intent of the shape and propagates this intent to its injection molded components. These techniques work well because they use a one-way driven process where the master part always drives the derived parts.





Base Part Technique

Steps in this lesson:

1. Use the **Insert Part** command combined with part line quilts and other cut techniques to derive individual parts from the master part.
2. Insert the individual parts into an assembly.
3. Check for the part associativity by changing a reference in the master part.

We recommend you save your work in folders named `Base`, `Split`, and `Multibody` to avoid confusion between techniques or the overwriting of documents.


Creating the Upper Housing

1. Click [here](#) : to open MouseBase.sldprt (or browse to drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\MoldedProductDesignAdvanced\Mouse Base.sldprt).
2. Click **File > Save As**, and save the part in a new folder named Base.
3. Close the part.
4. Click **New**  (Standard toolbar) and open a new part.
5. Click **Insert Part**  (Features toolbar).
6. In the dialog box, select MouseBase.sldprt (in folder Base) and click **Open**.
7. In the PropertyManager, under **Transfer**, select **Solid bodies** and **Surface bodies**, clear all other options, then click .


If you are asked to change the unit of measure of the derived part, click **Yes**.

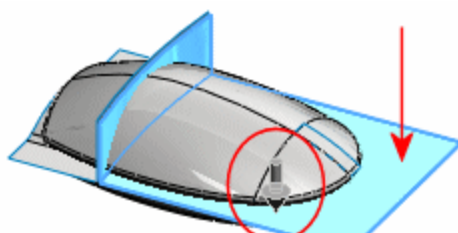
You select **Surface bodies** because you need the inserted part to contain the part line quilt surface. The **Insert Part** command inserts a part as a feature composed of solid bodies. The command inserts only axes, planes, cosmetic threads, or surfaces if you select those items under **Transfer** in the Insert Part PropertyManager.


To edit the entities transferred into an inserted part, in the FeatureManager design tree, right-click the inserted part feature and select **Edit Feature**.


8. Expand the **Surface Bodies** folder in the FeatureManager design tree, right-click **Surface-Extrude1** and select **Hide** .

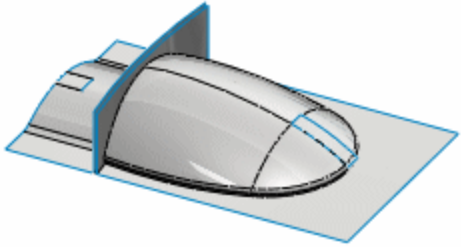
This surface body is not necessary for this lesson.

9. Click **Cut With Surface**  (Features toolbar).
10. In the PropertyManager:
 - a. Select the surface shown for **Selected surface for cut**.







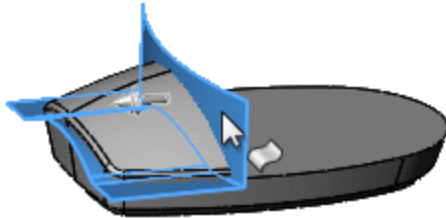
Make sure the arrow in the graphics area points downward so you cut away the bottom of the master part. If not, click **Flip Cut**  to reverse the arrow's direction.


- b.** Click  to cut away the buttons and the lower housing, while leaving the upper housing.
- 1.** Save the part as `UpperHousing.sldprt`.

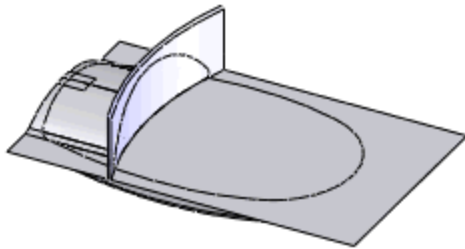


Creating the Lower Housing

1. In the FeatureManager design tree, right-click **SurfaceCut1** and select **Edit Feature** .
2. In the PropertyManager, click **Flip Cut**  so the arrow in the graphics area points upward.
3. Click .
4. Click **Cut With Surface**  (Features toolbar).
5. In the PropertyManager:
 - a. Select the surface shown for **Selected surface for cut**.

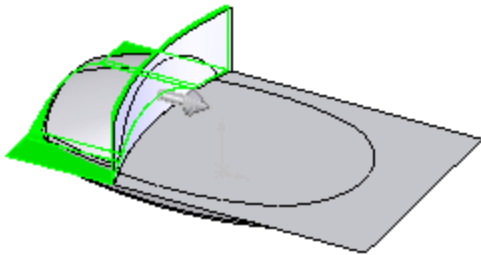


- b. Make sure the arrow in the graphics area points to the left of the cutting surface so you cut away the buttons
 - c. Click .
6. Save the part as `LowerHousing.sldprt`.

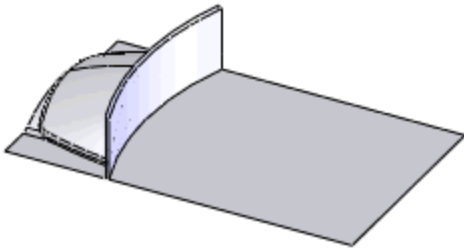






Creating the Left Button

1. In the FeatureManager design tree, right-click **SurfaceCut2** and select **Edit Feature** .
2. In the PropertyManager, click **Flip Cut**  so the arrow in the graphics area points to the right, away from the buttons.



3. Click .






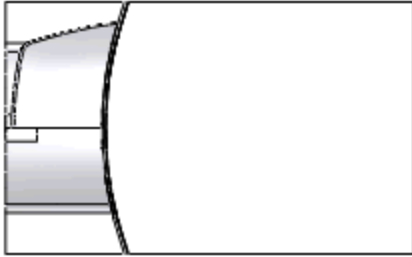
4. Click **Top**  (Standard Views toolbar).
5. Click **Cut With Surface**  (Features toolbar).
6. In the PropertyManager, select **Front**  in the FeatureManager design tree for **Selected surface for cut**.
7. Make sure the arrow in the graphics area points upwards to cut away the right button.
8. Click .



9. Save the part as **LeftButton.sldprt**.

Creating the Right Button

1. In the FeatureManager design tree, right-click **SurfaceCut3** and select **Edit Feature** .
2. Click **Flip Cut**  so the arrow in the graphics area points downward to cut away the left button.
3. Click .



4. Save the part as `RightButton.sldprt`.

Creating the Assembly


Drag the parts to create the assembly.

1. Create a new assembly document.

Close the Begin Assembly PropertyManager if it appears.

2. Open these three parts you just created: the left button plus the upper and lower housings.

The right button is already open.

3. Click **Tile Horizontally**  (Standard toolbar) so that all five documents are visible.

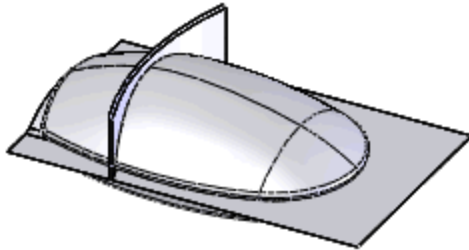
4. In the LeftButton document, drag the part name, **LeftButton**, from the top of the FeatureManager design tree and drop it onto the **Assem<n>** name at the top of the assembly document's FeatureManager design tree.

Dragging this way aligns all the documents properly to the new assembly document origin.

5. Repeat the process for the right button, upper housing, and lower housing parts.

6. Save the assembly document as `AssemblyBase.sldasm`.



Click **Yes** if you are prompted to rebuild and save referenced models.

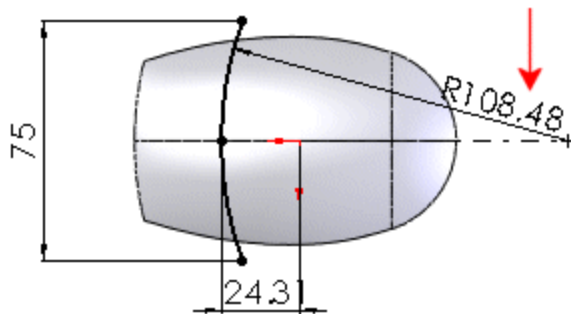


The assembly displays all the surface part line quilts. To hide the quilts, in the assembly's FeatureManager design tree, hide all the bodies in the **Surface Bodies** folder for all four components.

Changing the Master Part

Changing the master part changes the derived parts and the assembly because they all reference the master part. In this example, change the size of the mouse buttons.

1. Open `MouseBase.sldprt` (in folder `Base`).
2. Tile the windows horizontally. There should be six files open.
3. Display all the files in **Top**  view.
Tile horizontally again to resize the windows.
4. In `MouseBase.sldprt`, in the FeatureManager design tree, right-click the sketch for **Surface-partline-quilt-buttons** and select **Edit Sketch** .
5. Change the radius dimension shown from 108.48 to 250.



6. Exit the sketch and save the part.

Completing the Master Part

1. Click the title bar of the **UpperHousing.sldprt** file.

 [Video: Clicking the Part Title Bar](#)

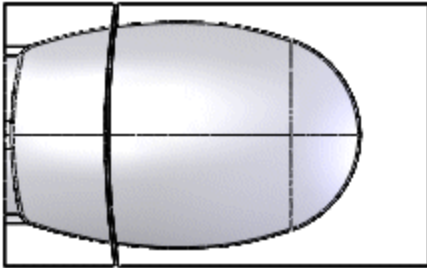
The part rebuilds. The surface part line changes shape, changing the size of the mouse buttons and the upper housing.

2. Click the title bar for the right button, left button, and the lower housing to update them.

3. Click the title bar for **AssemblyBase.sldasm**.

Click **Yes** if prompted to rebuild the assembly.

The assembly composed of the individual derived parts updates because of the associativity between all the parts and the assembly.



4. Save the assembly.

In the Save Modified Documents dialog box, click **Save All**.

5. Close all the files.

Congratulations! You have completed this lesson.

[Return to the tutorials overview page.](#)

Multibody Technique

The multibody technique is similar to the split parts technique. You create the multibody part using the **Split** command. However, you keep the bodies in the multibody part instead of splitting them off as separate parts.

To create the individual parts, you then use the **Save Bodies** command, which offers several advantages:



- Bodies are assigned IDs
- Body names are transferred automatically from the master part to the individual parts
- Individual parts are derived automatically
- You create the assembly directly from the PropertyManager

You use a version of the computer mouse that has already been split into a multibody part.

We recommend you save your work in folders named **Base**, **Split**, and **Multibody** to avoid confusion between techniques or the overwriting of documents.

Setting Up the Multiple Bodies

When you create the multibody part, give the solid bodies descriptive names because these names carry over into the individual parts you create.

1. Click [here](#) : to open MouseBase.sldprt (or browse to drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\MoldedProductDesignAdvanced\MouseMultibody.sldprt).
2. Click **File > Save As**, and save the part in a new folder named **Multibody**.
3. In the FeatureManager design tree, expand the **Solid Bodies**  folder.

The **Cord-Restraint** body is hidden because this tutorial deals with splitting off only the buttons and housings.

Note the descriptive body names:

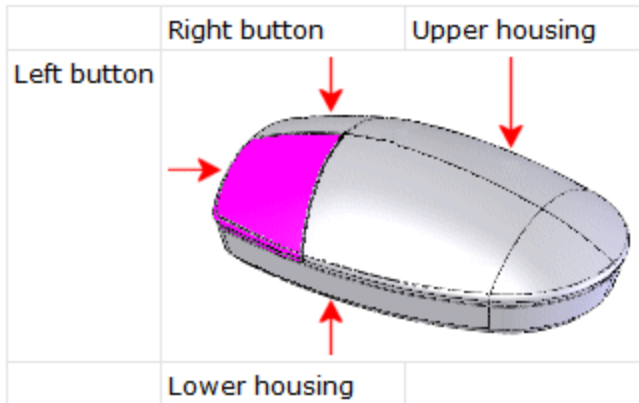
- Lower-Housing
- Upper-Housing
- Right-Button
- Left-Button



Creating Parts with the Save Bodies Command

1. Click **Insert > Features > Save Bodies**.

2. In the graphics area, select the left button on the model.

In the PropertyManager, the name `Left-Button.sldprt` appears for the selected body under **File**. The body name from the solid bodies folder is used as the individual file name.



You can also select the check boxes under  in the PropertyManager to select bodies, or you can click  to automatically name all the bodies.

3. Select the remaining three bodies.
Do not exit the **Save bodies** feature yet.

Creating the Assembly

Create the assembly directly from the Save Bodies PropertyManager.

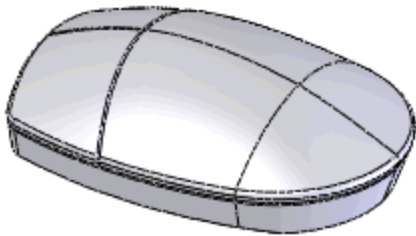
1. Under **Create Assembly**, click **Browse** and save the assembly as `AssemblyMultibody.sldasm`.

The **Save as type** box indicates this is a **SplitAssembly** file type.

The full path to the assembly appears in the PropertyManager.

2. Click .

Click **Rebuild and save the model** if prompted.





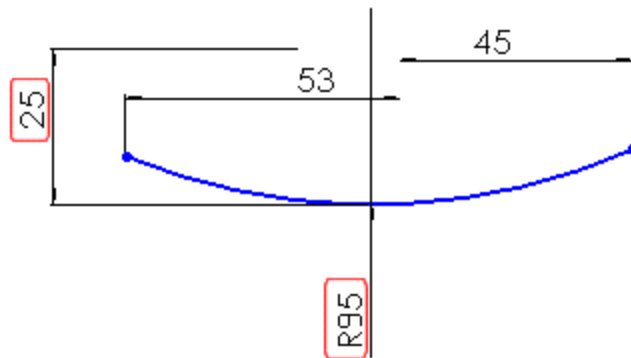
3. Click **Window** and select **AssemblyMultibody.sldasm**.


The components use the same names as the bodies you saved from the multibody part.

Changing the Master Part

Changing the master part changes the derived parts and the assembly because they all reference the master part. In this example, you change the width of the mouse.


1. Tile the windows and display a **Top**  view of `MouseMultibody.sldprt` and `AssemblyMultibody.sldasm`.
2. In the master part (`MouseMultibody.sldprt`), in the FeatureManager design tree, expand **Sweep1**, right-click **Sketch1** and select **Edit Sketch** .
3. Change these dimensions:
 - a. Change 30 to 25.
 - b. Change 125 to 95.



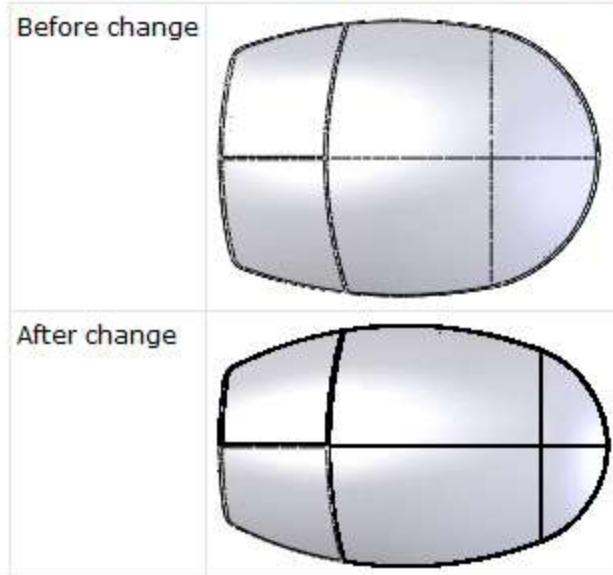
4. Exit the sketch.
5. If the part fails to rebuild and you get error messages, click **Continue (Ignore Error)** in the error message and **Close** in the What's Wrong dialog box, and then click **Rebuild** .
The master part, `MouseMultibody.sldprt`, rebuilds with the changed dimensions.
6. Save the part.

Completing the Change

Update the assembly to view the changes made to the master part.

1. Click the title bar for **AssemblyMultibody.sldprt**.
2. Click **Yes** when prompted to rebuild, or click **Rebuild**  in the assembly file.

The assembly adjusts size, as do all related components.



3. Save the assembly.
In the Save Modified Documents dialog box, click **Save All**.
4. Close all the files.

Expected Output : The drafting should have views of the multibody product if we are to submit for filing of an patent

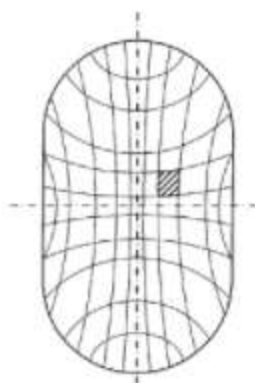


FIG. 12A

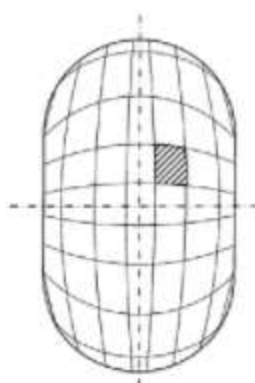


FIG. 12B

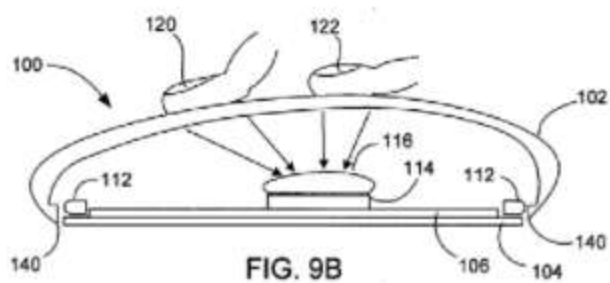


FIG. 9B

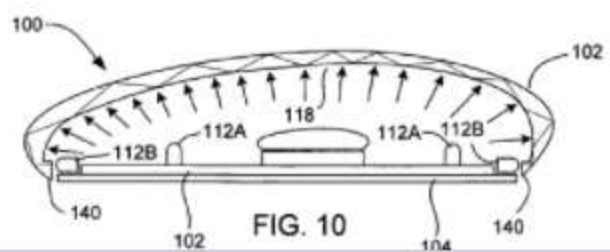


FIG. 10

Tutorial : 3: Rendering of product

PhotoView 360 and Appearances Overview

You can use appearances and decals to make your model appear more realistic. With PhotoView 360, you can create photorealistic renderings of your model to use for promotional or other purposes.

There are two lessons in this tutorial:

- [Appearances and Decals](#)



- [PhotoView 360](#)



Appearances and Decals

In this lesson, you learn how to work with appearances and decals.

You will learn to:


- Use the DisplayManager
- Add appearances to a parts
- Add appearances to features
- Edit appearances
- Apply a decal




Learning About the DisplayManager




In this section you learn about the DisplayManager and how to use it effectively to manage the visual aspects of your model.

To start working with model appearances:


1. Click [here](#)  or open drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\appearances\USB_flash_drive1.sldasm.

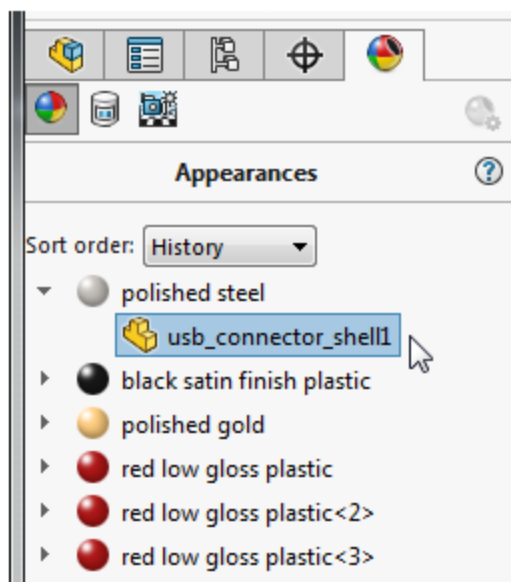


2. In the Manager pane, click the DisplayManager tab .

The DisplayManager catalogs the appearances ; decals ; and scenes, lights, and cameras  applied to the model. You can create, edit, or delete each item in the DisplayManager.


[A closer look at appearances](#)

3. Click **View Appearances** .
4. In **Sort order**, select **History**.
The model history displays with several different appearances, including three instances of **red low gloss plastic**.
5. Expand the first **polished steel** appearance to see it is applied at the part level to **usb_connector_shell1**.




Adding an Appearance to a Part

Next you add an appearance at the part level.

1. On the Heads-up View toolbar, under **View Settings** , select **RealView Graphics** to enable RealView Graphics.

If your hardware does not support RealView, you might not be able to see all the visual effects in this tutorial.

2. In the graphics area, right-click the main portion of the flash drive and click **Open Part**  to open the part USB_case1 in its own window.

[Why did I open the part in its own window?](#)

3. In the Task Pane, on the Appearances, Scenes, and Decals  tab, click **Appearances > Plastic > High Gloss**.
4. In the bottom pane, double-click **blue high gloss plastic** to apply it to the entire part.




Adding Appearances to Features

In this section, you apply appearances at the feature level.

1. In the Task Pane, on the Appearances, Scenes, and Decals  tab, click **Appearances > Plastic > Medium Gloss**.
2. In the bottom pane, drag **cream medium gloss plastic** onto `Cover_boss` in the FeatureManager design tree to apply it to the feature.





3. In the Task Pane, on the Appearances, Scenes, and Decals  tab, click **Appearances > Metal > Steel**.
4. In the bottom pane, drag **brushed steel** onto the large outer face of the part.

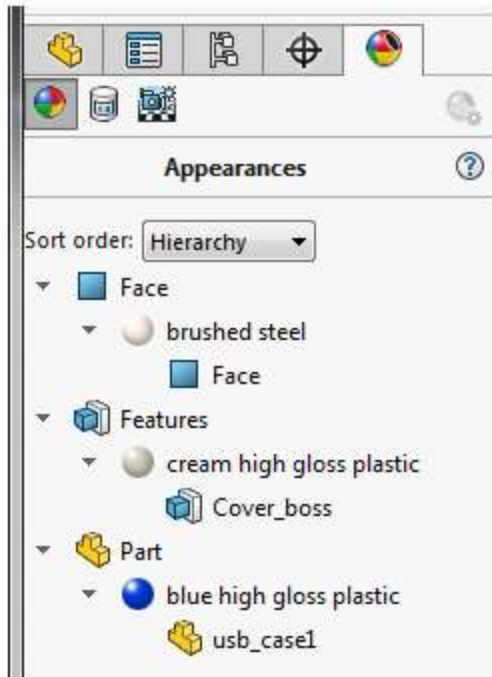
The Appearance Target palette appears in the graphics area. Choose the face option to finish applying the appearance.



Reviewing Your Work


Review your work:




1. Open the DisplayManager , click **View Appearances**  and sort by **Hierarchy**.
2. Expand **Face**, **Features**, and **Part** to see appearances applied at each level.
3. Expand appearances to see which model entity they are applied to.




Why is the appearance hierarchy sort order listed in this way?

Editing an Appearance

1. In the Task Pane, on the Appearances, Scenes, and Decals  tab, click **Plastic > High Gloss**. Drag **green high gloss plastic** onto the part USB_case1 in the FeatureManager design tree.


Because you applied the appearance at the part level, the face and feature level appearances do not change.
2. In the DisplayManager, under **Face**, right-click **brushed steel** and click **Edit Appearance** .
3. In the PropertyManager, click **Advanced**.
4. On the Mapping tab, under **Size/Orientation**, set the **Width**  and **Height**  to 8mm to reduce the scale of the brushed texture. The texture updates in the graphics area.

To automatically scale appearance textures to the model size, click **Tools > Options > Document Properties > Model Display** and select **Automatically scale appearance textures, surface finishes, and decals to the model size**.
5. Click .
6. Save and close the file. Return to USB_flash_drive1.SLDASM. If you are prompted to rebuild the model, click **Yes**.
7. Save the file USB_flash_drive1.SLDASM.

Adding a Decal

In this section, you add a decal to the part.


In SOLIDWORKS 2011 and later, you can add decals without loading any add-in applications.

1. In the FeatureManager design tree, right-click the part USB_case1 and click **Open Part**  to open the part in its own window.






If you are prompted to proceed with feature recognition, click **No**.

2. Press the space bar and click **Top**  in the View Orientation popup.



3. In the CommandManager, on the Render Tools tab, click **Edit Decal** .

If the Render Tools tab is not visible, right-click any tab in the CommandManager and click **Render Tools**.

4. In the Decals PropertyManager, click **Browse** and open drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\appearances\solidworks_logo.png.
 - a. On the Image tab, under **Mask Image**, select **Use decal image alpha channel**.
 - b. Select the main face of the part.
The image appears on the selected face in the graphics area.
 - c. On the Mapping tab, under **Size/Orientation**, set the **Rotation**  to 270 degrees.
 - d. Select **Fixed Aspect Ratio**. Set the **Width**  to 40mm.
Because **Fixed Aspect Ratio** is selected, the **Height**  adjusts automatically.
 - e. Under **Mapping**, set the **Along Axis**  to -3 mm.
 - f. Click .The image is applied as a decal.



If the decal disappears, click **View > Hide/Show > Decals** to show the decal.

5. Save and close the file.
6. In the USB_flash_drive1.SLDASM file, the decal appears in the assembly.
7. Save the file USB_flash_drive1.SLDASM.

Congratulations! You have completed this tutorial.

If you have PhotoView 360, you can continue to work with this model. [Click here](#) to go to the PhotoView 360 tutorial.

[Return to the tutorials overview page.](#)

PhotoView 360

In this lesson, you learn to work with cameras, scenes, and lighting in PhotoView 360.

The steps include:

- Using a camera view
- Adding appearances to components
- Changing and editing scenes and lighting
- Saving a final render

There are two sets of sample files for this tutorial. If you started with the appearances tutorial, you can continue to use the sample assembly USB_flash_drive1 and its components. If you are starting with the PhotoView 360 tutorial, use the sample assembly USB_flash_drive2 and its components.

For example, if the tutorial prompts you to open the part USB_cover n :


- $n = 1$ if you started with the Appearances and Decals tutorial
- $n = 2$ if you are starting with the PhotoView 360 tutorial



Opening the PhotoView 360 Preview Window

In this section, you use PhotoView 360 to preview your render.

Previewing your render makes it easy to adjust cameras, scenes, lighting, and shadows before you create your final image.

1. [Click here](#)  or browse to drive letter: \Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\appearances\USB_flash_drive2.sldasm to open the sample assembly.


You can open an unfinished version of the assembly to practice working with appearances and decals. [Click here to learn how.](#)

2. To enable PhotoView 360, click **Tools > Add-Ins** and select **PhotoView 360**.
3. Click **OK**.
4. On the Render Tools tab in the CommandManager, click **Preview Window**



5. In the dialog box, click **Continue without Camera or Perspective**.
Even though the dialog box recommends it, you do not need to turn on a camera yet. In the next topic, you turn on a camera that was previously created and saved with the model.
6. Arrange or resize the Preview window so that you can see the graphics area.

While it is open, the Preview window remains on top of the SOLIDWORKS window.

You can preview renders in the SOLIDWORKS graphics area, rather than the Preview window, by clicking **Integrated Preview**  on the Render Tools tab.

Working with Cameras

In this section you activate a camera view.

[A closer look at cameras](#)

1. In the DisplayManager , click **View Scenes, Lights, and Cameras** , and expand **Camera** .
2. Right-click **Camera1** and click **Camera View**.

The previously saved camera view is activated. The SOLIDWORKS graphics area and preview window update to the Camera1 view.

Camera1 view was previously created and saved with the model.



Adding an Appearance to Components

Next you add an appearance to the case and cover at the component level.


[A closer look at appearances](#)

1. In the FeatureManager design tree, press **Ctrl** and select `USB_casen` and `USB_covern`.
2. In the Task Pane, on the Appearances, Scenes, and Decals tab, click **Appearances > Plastic > High Gloss**.
3. In the bottom pane, double-click **white high gloss plastic** to apply the appearance to the preselected parts.

The render preview window updates in real time.




[Why did I apply the appearance at the component level?](#)

4. In the DisplayManager, click **View Appearances**  and sort by **Hierarchy**.
Components have been added to the appearance hierarchy sort order.
5. Expand **Components** and **white high glass plastic**.
`USB_covern-1@USB_flash_drive`, `USB_casen-1@USB_flash_drive` are listed.

If you edit the **white high gloss plastic** appearance, it will affect all the components with that appearance.

[Why is the appearance hierarchy sort order listed in this way?](#)

6. In the graphics area, right-click the USB case and click **Open Part**  to open the part in its own window.



The USB case retains the color applied at the part level because component level appearances override part, body, feature, and face level appearances in the assembly only.

7. Close the file and return to `USB_flash_driven.SLDASM`
8. In the Task Pane, on the Appearances, Scenes, and Decals tab, click **Appearances > Lights > Neon Tube**. Drag and drop the **Blue Neon Tube** appearance onto the `USB_LEDn` part in the FeatureManager design tree.

Working with Scenes


In this section, you make changes to the environment around the model.

You can dramatically change the lighting and reflections in your model by changing and editing the scene. If you don't like the way your model looks, we recommend that you change the scene first and then adjust the lights, reflections, and shadows as needed.


You can also use scenes with context backgrounds for promotional materials.

The current scene is **3 Point Faded**. To change and edit the scene:

1. In the Task Pane, on the Appearances, Scenes and Decals tab, click **Scenes > Studio Scenes**. Drag **Reflective Floor Black** into the SOLIDWORKS graphics area.

The background, lighting, and reflections change in the SOLIDWORKS graphics area and the Preview window. In the DisplayManager, in Scenes, Lights, and Cameras , the scene changes to **Reflective Floor Black**.



2. Right-click **Scene (Reflective Floor Black)** in the DisplayManager and click **Edit Scene**.
3. In the PropertyManager, under **Floor**, clear **Floor Reflections**.
The reflections disappear in the SOLIDWORKS graphics area and the preview window.
4. On the PhotoView 360 Lighting tab, under **Environment Rotation**, drag the slider to 180 degrees.
The reflections and lighting dynamically change in the preview window and the SOLIDWORKS graphics area. You can most easily see the changes by watching the USB cap in the graphics area.
5. Under **PhotoView 360 Lighting**, set the **Background Brightness** to 1.5 and the **Rendering Brightness** to 1.0.
Only the preview window changes because PhotoView 360 Lighting adjustments affect only the rendered image.
6. Click .

The best way to change the rendering brightness is to adjust the Primary PhotoView 360 Lighting because it is easier to control than other lights. It is also easier to make Primary PhotoView 360 Lighting appear realistic. We recommend adjusting the Primary PhotoView 360 Lighting first and then working with other lights as needed.




Working with Lighting

In this scene, you change the lighting and shadows.

When should I use lights?

Because they are visible in both SOLIDWORKS and PhotoView, each point, spot, and directional light appears in both **SOLIDWORKS Lights** and **PhotoView 360 Lights**. By default, lights are off in PhotoView because the lighting provided by scenes is usually sufficient to illuminate the model.

1. In the DisplayManager, expand **PhotoView 360 Lights**, right-click **Directional2** and click **On in PhotoView 360**. The Preview window updates.
2. Right-click **Directional2** and click **Edit Directional Light**.
 - a. In the PropertyManager, on the **PhotoView 360** tab, select **Shadows**.
 - b. Set the **Shadow Softness** to 2.00deg.
The Preview window updates as the settings are adjusted.
 - c. Click .





3. Save the file.

Performing a Final Render


The Final Render is the image you create after making all the necessary changes in your preview.

1. Click **PhotoView 360 > Options**.
2. In the PropertyManager, under **Output Image Settings**, change the **Output image size** to **640X360 (16:9)**.

If it is selected, clear **Use background aspect ratio** so you can change the **Output image size**.

3. Select **Fixed aspect ratio**.
4. Set the width  to 1020. Because you selected **Fixed aspect ratio**, the height  adjusts automatically and the aspect ratio (16:9) remains the same.

You can increase the number of pixels in the image for a higher resolution while maintaining the aspect ratio.

5. Click .
6. On the Render Tools tab of the CommandManager, click **Final Render**.
The Final Render window opens and rendering begins.

You can make detailed post-processing adjustments in the Final Render window. For more information about Final Render window options, see *SOLIDWORKS Help:Final Render Window*.

7. When rendering is finished, click **Save Image**.
Browse to the directory, type a file name, and select the type of image to save.
8. Click **Save**.
The image is saved as a JPG.

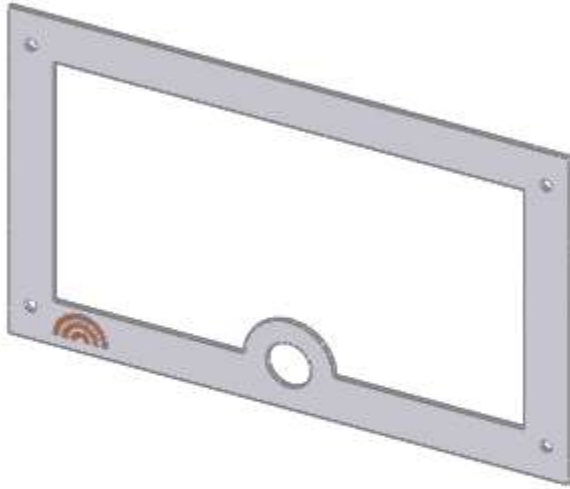
Expected Outcome : Draft of the rendered product

Tutorial – 4: Import/Export/Repair of a model

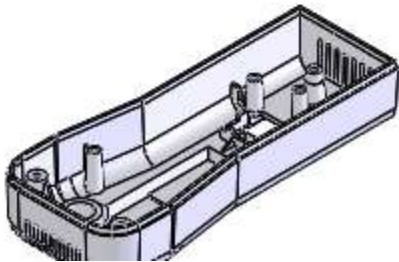
Import/Export Overview

There are two lessons in this tutorial:

- **Import/Export Basics:** Guides you through importing a gasket and logo, then exporting the model as a different file type.



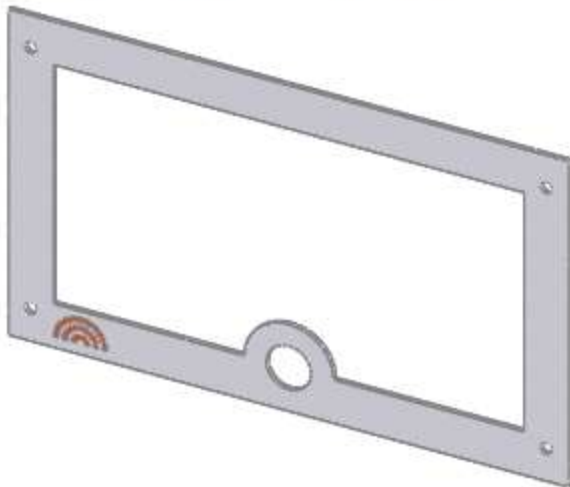
- **Diagnosing and Repairing Import Errors:** Guides you through repairing common import errors using Import Diagnostics and other tools.



Import/Export Basics

This lesson guides you through the import of a gasket and company logo, and demonstrates the following:


- Importing an IGES file
- Inserting a DXF file
- Exporting a SOLIDWORKS part document as an STL file

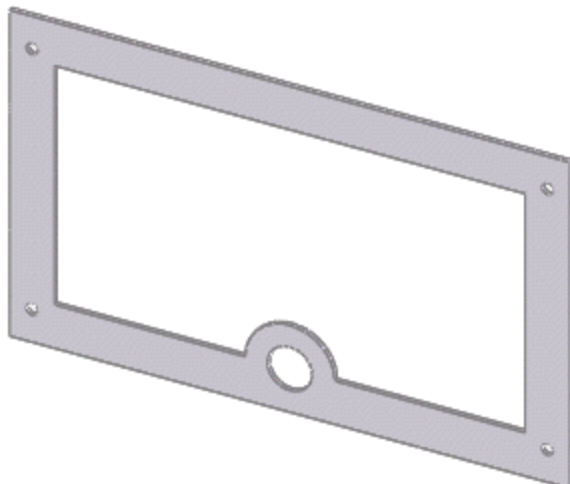


Importing an IGES File

You can import files to the SOLIDWORKS software from other applications. The geometry and structure of the resulting SOLIDWORKS model matches that of the model in the source application.

In this lesson, you import surfaces from an IGES file. Because the surfaces form a closed volume, you can use them to create a base feature.

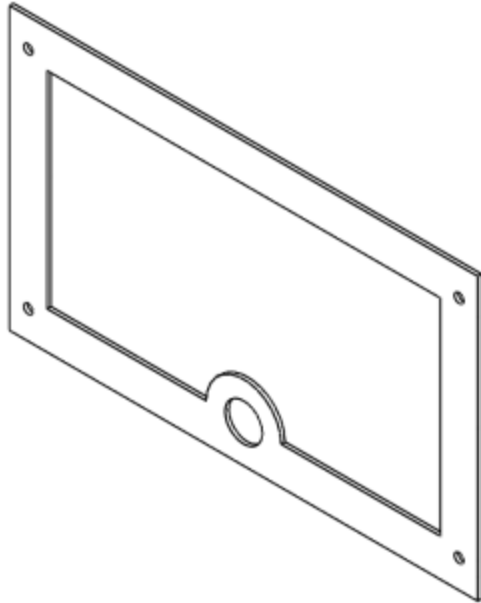
1. Click **Open** .
2. In the dialog box, select **IGES (*.igs; *.iges)** in **Files of type**.
3. Click **Options** to set the import options.
4. In the Systems Options dialog box, click **Import**.
5. Verify that **General** is selected for **File Format** and clear **Enable 3D Interconnect**.
6. Select **Solid and Surface** and **Try forming solid(s)**.
The SOLIDWORKS software attempts to form solids from the surface or solid entities in the imported file.
7. Click **OK** to accept the other default settings.
8. Browse to `drive letter:\Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\importexport`, select `gasket.igs`, and click **Open**.
9. If prompted to run Import Diagnostics, click **No**.
The SOLIDWORKS software forms a base feature from the imported surfaces. The imported body appears in the graphics area.
10. If prompted to proceed with feature recognition, click **No**.
You can use [FeatureWorks](#) to recognize imported features as editable SOLIDWORKS features. For example, using FeatureWorks, you could recognize the gasket as an extrude feature with hole features.





Inserting a DXF File

You can insert a DXF or DWG file directly into the SOLIDWORKS document. The DXF file that you insert in this lesson contains the company logo for a fictitious company, Rainbow Corporation. The gasket file should still be open.

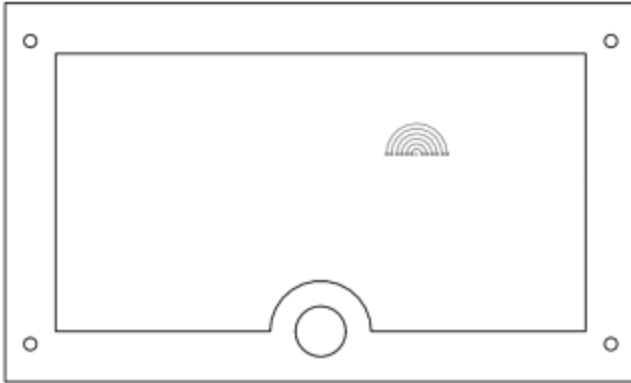
1. Click **Hidden Lines Removed** .



2. Click **Front**  on the Standard Views toolbar.
3. In the graphics area, select the front face of the gasket.
4. Click **Insert** > **DXF/DWG**.
5. In the dialog box, browse to drive letter: \Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\importexport, select rainbow.dxf, then click **Open**.
6. In the DXF/DWG Import dialog box, select **Import to part as** and **2D sketch**, then click **Next**.
7. In the DXF/DWG Import - Document Settings dialog box, clear **Add constraints**, to solve all apparent relations and constraints in the sketch, then click **Next**.
8. In the DXF/DWG Import - Drawing Layer Mapping dialog box, select **Merge points closer than** and accept the distance of **0.001**. This option merges points that, after import, are within a specified merge distance.
9. Click **Finish**.


10. Click **Rebuild** .


A new sketch that contains the company logo is created in the part.

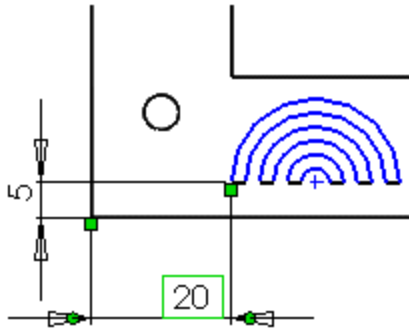



Repositioning the Sketch Entities


Now reposition the inserted sketch entities.

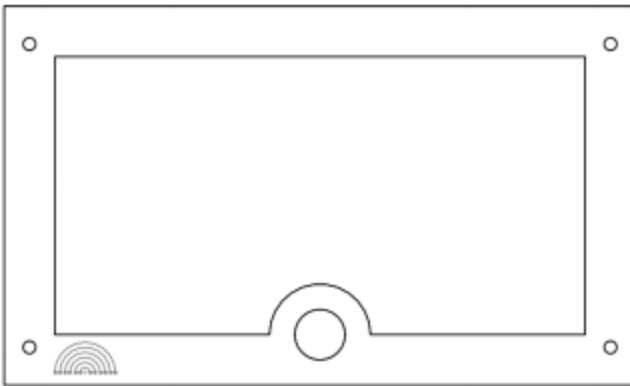
1. In the FeatureManager design tree, right-click **Sketch1** and select **Edit Sketch** .
2. Add the dimensions from the lower left corner of the rainbow to the gasket edges, as shown, to position the sketch entities.

If the sketch does not appear to move, click **Rebuild** .







You can click **Smart Dimension**  (Dimensions/Relations toolbar) to define the dimensions between the sketch point and the lines.

3. Click **Exit Sketch** .






Extruding the Company Logo

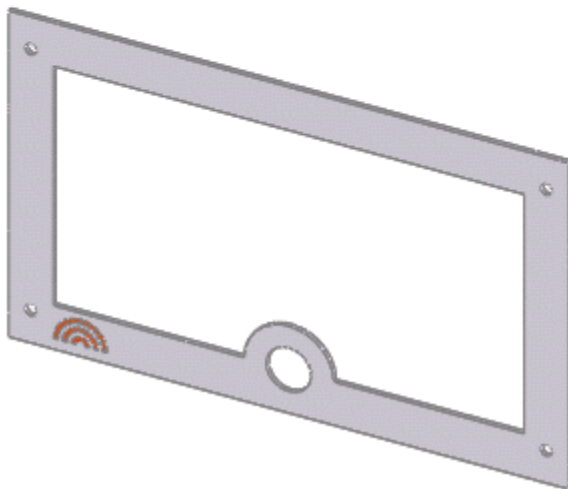
To finish the part, extrude the company logo into the gasket.

1. Click **Trimetric**  on the Standard Views toolbar.
2. Select **Sketch1** in the FeatureManager design tree.
3. Click **Extruded Cut**  on the Features toolbar.
4. In the PropertyManager, under **Direction 1**:
 - a. Select **Blind** in **End Condition**.
 - b. Set **Depth**  to 1.
5. Click .

Changing the Extrusion Color




Now change the color of the new extrusion.

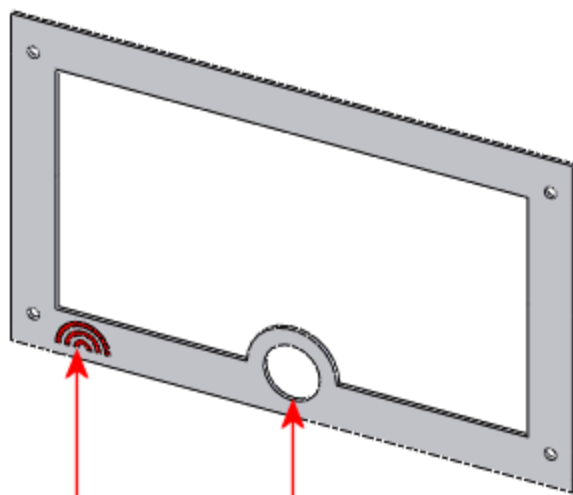
1. Select **Cut-Extrude1** in the FeatureManager design tree.
2. Click **Edit Appearance**  on the Standard toolbar.
3. In the PropertyManager, under **Color**, select a color from the color palette, then click .
4. Click **Shaded With Edges**  to display the colored logo on the gasket.



Editing Imported Features

You can replace an imported feature with geometry from a new file. In this example, you replace the gasket with another gasket that has a larger central hole.

1. In the FeatureManager design tree, right-click **Imported1**  and select **Edit Feature** .
2. Click **OK** in the parent/child message.
3. In the dialog box, browse to drive letter: \Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\importexport, select gasket_edited.igs, then click **Open**.
4. The **Imported1** feature is replaced with the new imported body that has a larger central hole. The software also rebuilt the pre-existing cut-extrude feature. The software rebuilds pre-existing features whenever possible.
5. Click **Shaded With Edges** .



Exporting an STL File

You can save a SOLIDWORKS part document as an STL file. The STL format is intended for transfer to rapid prototyping machines.

1. Click **File > Save As**.
2. In the Save As dialog box, select **STL (*.stl)** in **Save as type**, then click **Options** to set the export options.
The System Options dialog box appears with **STL** selected under the File Format.
3. Under **Resolution** select:
 - **Fine** to create a finely tessellated STL file.
 - **Show STL info before file saving** to display a dialog box when you save the file that contains numerical data about the STL file.
4. Click **OK**.

You can experiment with the **Resolution** settings to determine the best settings for your own rapid prototyping machines.

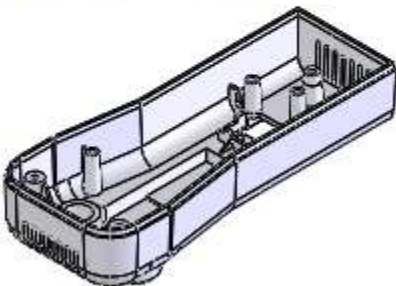
5. Click **Save** to save the file with the default name, *gasket.STL*.
A message box appears displaying the number of triangles, file size, and file format.
6. Click **Yes** to complete the save operation.
If a dialog box asking if you want to export all bodies appears, click **OK**.

Congratulations! You have completed this lesson.

Diagnosing and Repairing Import Errors

This lesson guides you through diagnosing and repairing common import errors using these tools:

- Import Diagnostics to heal gaps and bad faces
- Features to fill in gaps




Using the Import Diagnostics Tool

Inconsistencies can occur when you import geometry from another design system. You need to check, and sometimes repair, imported files. You can use Import Diagnostics to identify and repair problems with imported geometry.

1. Click **File > Open**.
2. Select **Parasolid (*.x_t,*.x_b,*.xmt_txt,*.xmt_bin)** in **Files of type** and browse to drive letter: \Users\Public\Public Documents\SOLIDWORKS\SOLIDWORKS version\samples\tutorial\importexport\importerrors.x_b).
3. If prompted to run Import Diagnostics, click **No**.

A message automatically prompts you to run Import Diagnostics when you open a part with an imported feature.

4. In the FeatureManager design tree, click to highlight each of the two imported surface bodies to familiarize yourself with the model.
5. Click **Import Diagnostics**  (Tools toolbar).
The PropertyManager reports 12 gaps between faces.
6. Under **Gaps between faces**, right-click a gap and select **Zoom to Selection** to zoom to the gap. Examine each gap to familiarize yourself with how the part currently looks.
The list displays the gap number and the number of free edges. For example, **Gap<2> [10]** means the second gap that has 10 edges.

To view a gap from the opposite direction, right-click a gap and select **Invert Zoom to Selection**.

Repairing Gaps and Faces

1. Under **Gaps between faces**, click **Attempt to Heal All** to repair the reported gaps.

Import Diagnostics checks for faulty faces, then tries to re-knit gaps. If that fails to completely repair the model, Import Diagnostics attempts to close gaps by creating new smaller faces and knitting them into place.

The PropertyManager reports one gap and 19 faulty faces.

2. Under **Faulty faces**, right-click a face and select **Zoom to Selection** to zoom to the face. Examine each face to check if it makes sense with respect to the correct part geometry.

Examination reveals that problems seem to exist around the area by the vent and screw holes and that **Face<2>** is a duplicate face. The model cannot be knitted together because the duplicate face causes invalid topology.

3. Right-click **Face<2>** and select **Delete Face**.

All faces now display a checkmark ✓, which indicates they are repaired. The message box reports that the last operation to heal a gap failed and that you can model a patch for the gap manually.

The message box changes colors to indicate the number of errors remaining. Red indicates a high number, while green means no errors remain.

4. For each of the remaining gaps, right click and select **Heal gap**.

5. Click ✓.


Within Import Diagnostics, use these optional tools from the shortcut menu:

- **Gap Closer**. Manually repair small gaps.
- **Repair Face**. Select individual faces to repair. The **Attempt to Heal All** command is very order-dependent. You might need to assist it by repairing individual faces in a different order after you run **Attempt to Heal All**.

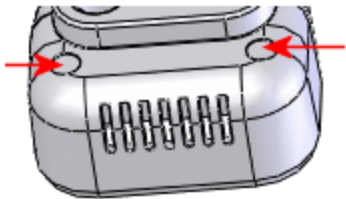
Correcting Repair Errors


Import Diagnostics mistakenly filled some surfaces during healing. You can delete these surfaces using Import Diagnostics.

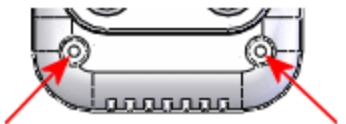
1. Click **Zoom to Fit**  (View toolbar).

2. Click **Shaded with Edges** .

Rotate the model to view it as shown. Examine the two screw mounting holes. Note that they have been incorrectly filled in.

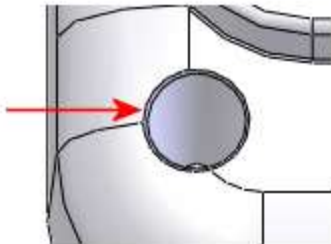


3. Click **Import Diagnostics**  (Tools toolbar).
4. In the graphics area, select the face covering one of the holes so it appears under **Faulty faces** in the PropertyManager.
5. Right-click **Face<0>** and select **Delete Face**.
6. Select the second face, then right-click and delete it.
7. Repeat these steps to delete the two incorrect faces covering the other screw hole.
8. Check the model to make sure the incorrect faces are deleted.

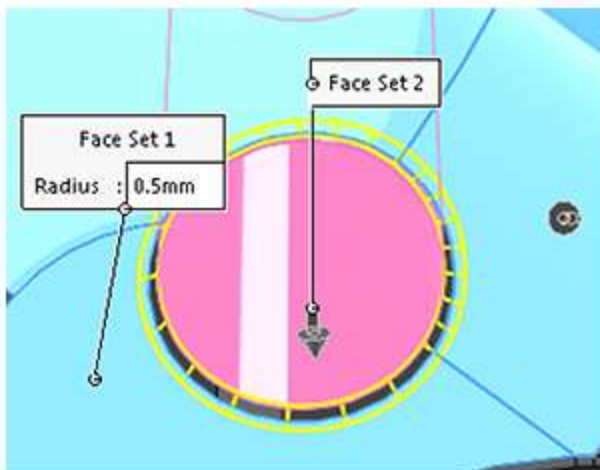




Recreating Missing Fillets

Import Diagnostics reports that four gaps remain because there are two missing fillets for each screw hole.



1. Click  to close the PropertyManager.
2. Click **Fillet**  (Features Toolbar).
3. Under **Fillet Type**, select **Face Fillet** .
4. Under **Items To Fillet**:
 - a. Select the top faces surrounding the screw hole for **Face Set 1**.
 - b. Select the inside face of the hole for **Face Set 2**.





5. Under **Fillet Parameters**, set **Radius**  to 0.5mm.
6. Click .
7. Repeat steps 2-6 to fillet the other screw hole.

Thickening the Model

In this lesson, you repaired all faulty faces and gaps using Import Diagnostics and fillet features. You can therefore thicken the model to turn it into a solid model.

Sometimes, healing the model fails to repair all import errors. In that case, you cannot form a solid model from the imported surfaces because they do not completely enclose a volume.

You can build the missing surfaces using surface tools, such as Fill Surface, Lofted Surface, or Planar Surface, and where necessary, the 3D Sketch tool. Then use the Knit Surface tool to knit all the surfaces together and either select **Try to form solid** in the PropertyManager or run the **Thicken** command.

1. Click **Insert** > **Boss/Base** > **Thicken** .
2. Select any face of the model in the graphics area.
3. In the PropertyManager, select **Create solid from enclosed volume** to turn the surface into a solid.
4. Click .

The **Thicken** body replaces the **Surface Bodies** folder.

Other tools exist to repair import errors:

- **Check Tools.** Checks model geometry and identifies undesirable geometry in parts.
- **Heal Edges.** Merges multiple edges into a single edge.

Expected Outcome : Drafting of the completed import and export repaired model