ME 24-688 – Week 3

Project 2 – Plastic Part Shell Method

Plastic Part Commands

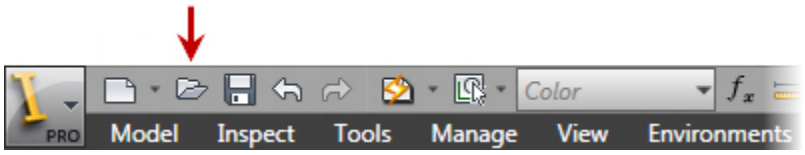
The following section will give an overview of the Autodesk Inventor plastic part commands.

1. Project 2

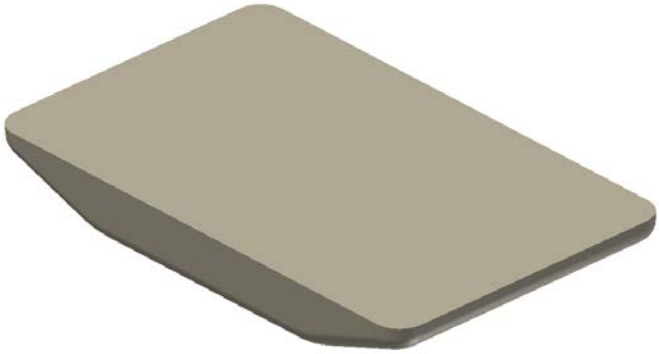
This project will introduce you to the Autodesk Inventor 2012 plastic part design commands. The main purpose of this project is to highlight the built in tools that allow for standard plastic part features to be created with minimal effort.

1: Open an existing part file.

- On the Quick Access toolbar, click Open.

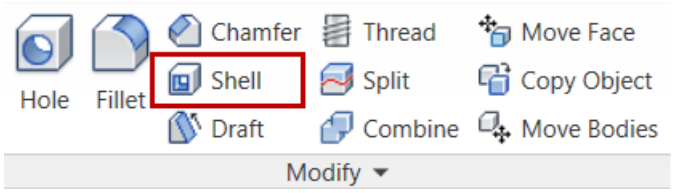


- In the **Open** dialog box, select the file *Plastic Case.ipt*
- Click **OK**.

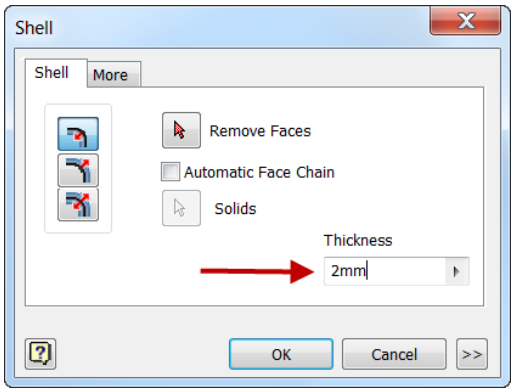


2: In order to create a finished plastic case from just a solid object you will need to shell or scoop out the middle portion. By using the shell command.

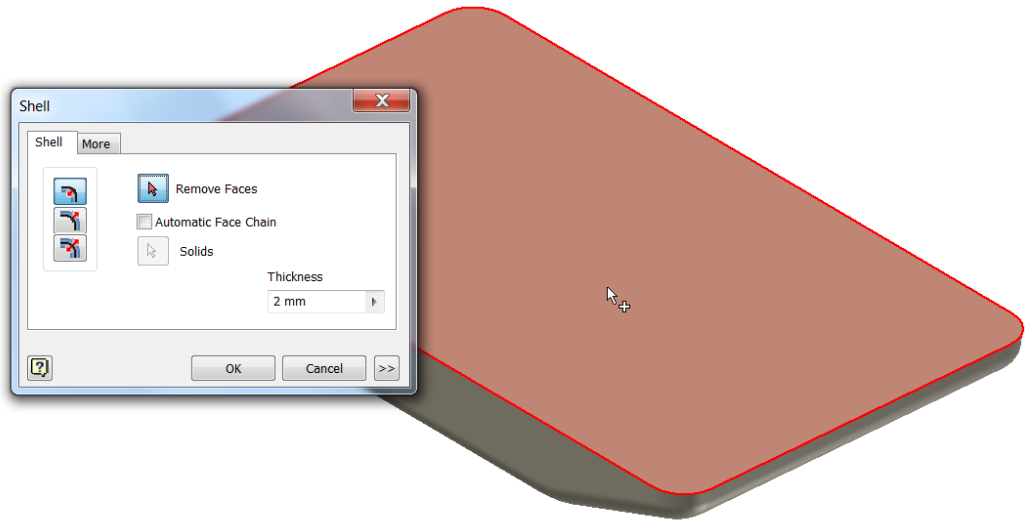
- Shell tool.
  Model Tab | Modify Panel | Shell



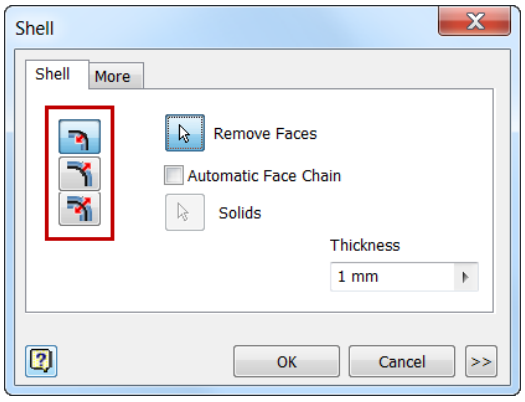
- In the shell dialog box you change the part thickness to **2mm**



- Click the face on the outer part shape that we would like to remove.



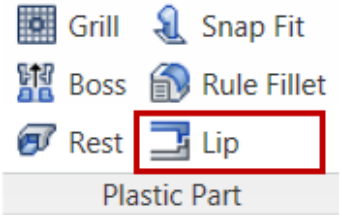
- The shell command gives you the option of creating the shell in various directions, inside, outside or both.



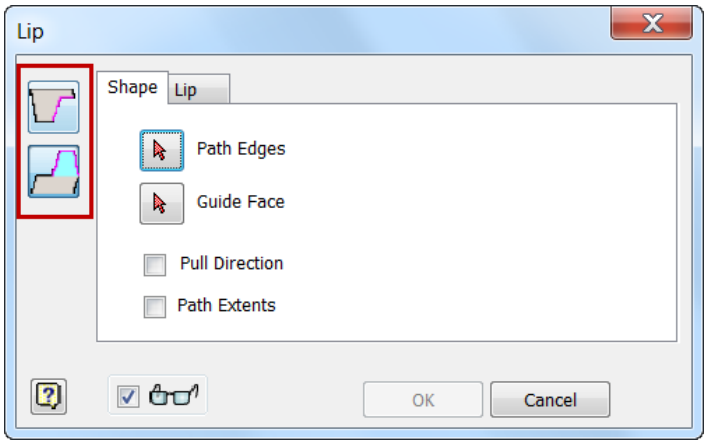
- Click ok to accept the chosen shell tool settings. You will be left with a shell of the outer shape.

3: Many plastic parts are made up of two half's and mate to each other along a seam. A standard way to create this seam is with a lip and groove. Let's create a groove on this lower case so the upper case can mate to it. We will use the lip tool to create this typical used plastic part feature.

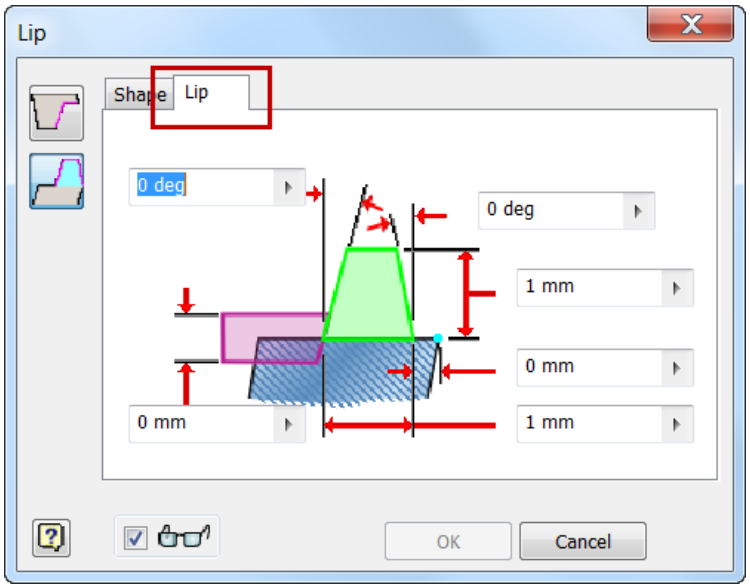
- Lip tool
  Model Tab | Plastic Part Panel | Lip



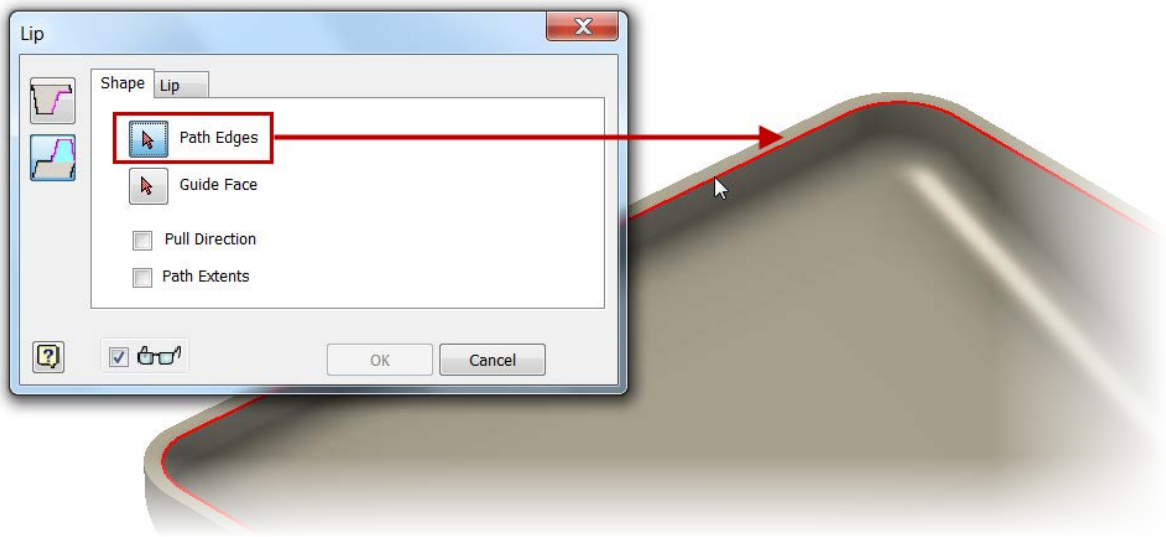
- You will have the choice of creating a lip or groove.



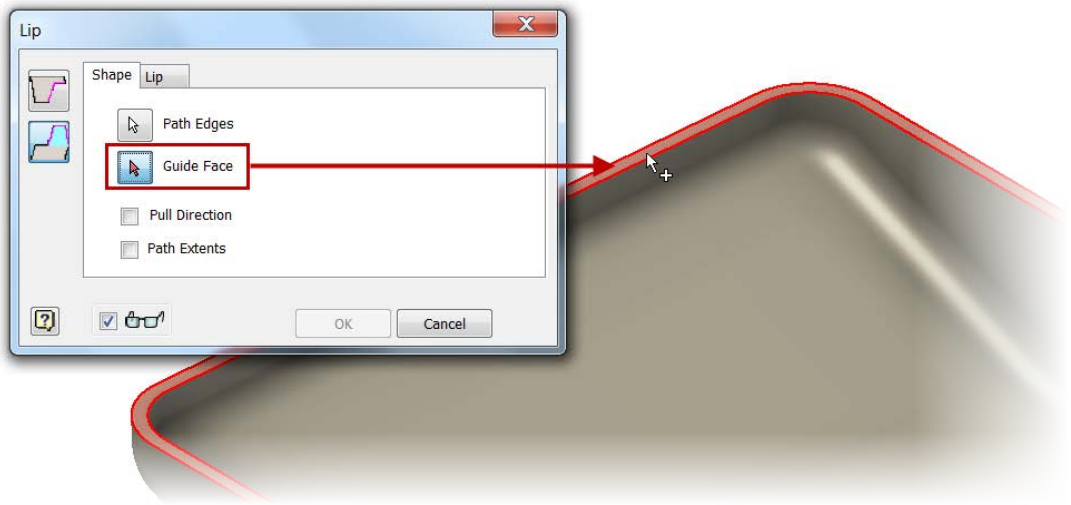
- You can change the size of the lip or groove by clicking on the attribute tab. Enter a **1mm** by **1mm** lip size. Return back to the shape tab.



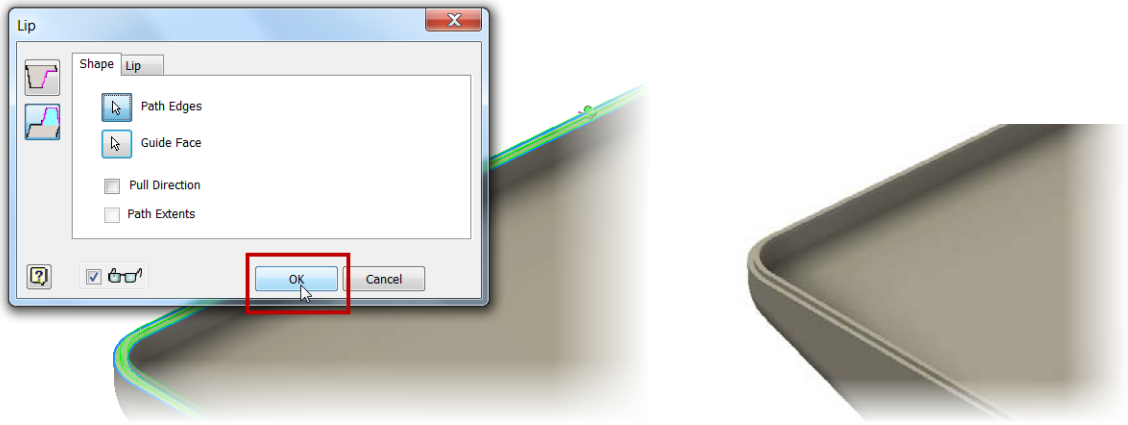
- Select the edge path you would like the lip to follow


Lip
Shape
Lip
Path Edges
Guide Face
Pull Direction
Path Extents
OK
Cancel

- We also need to select the face we would like the lip to be placed on.

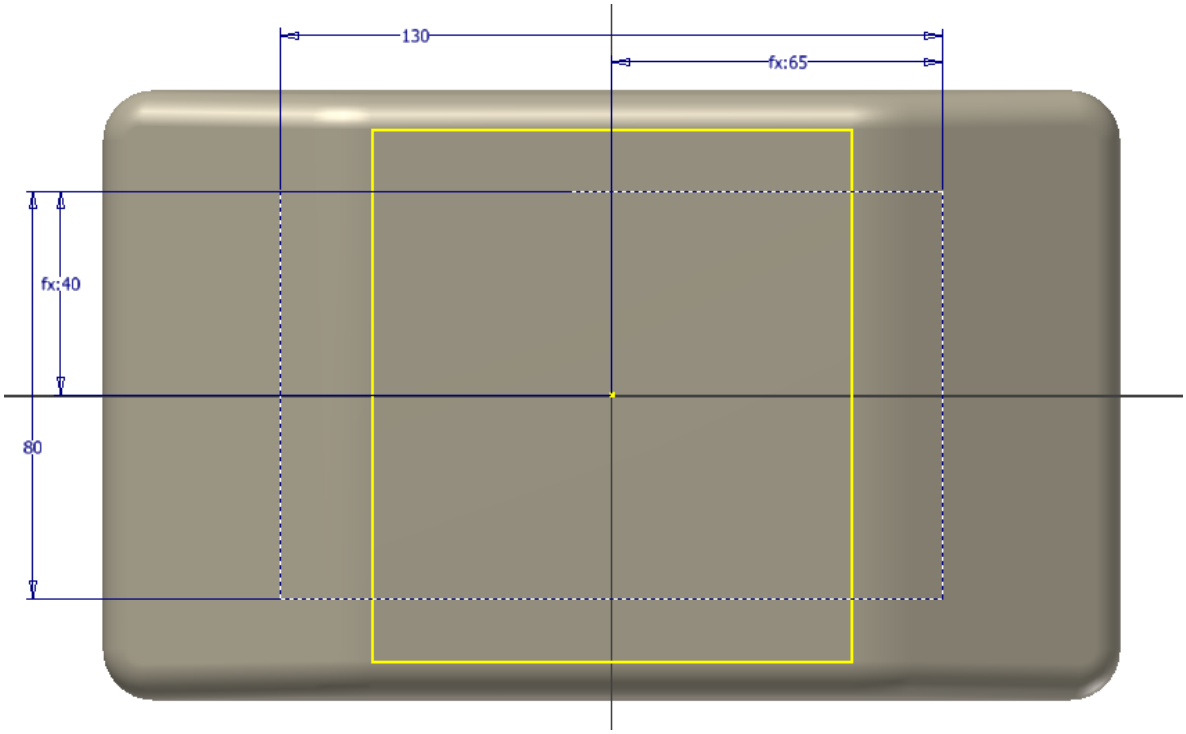


- Click ok to create the lip on the case.

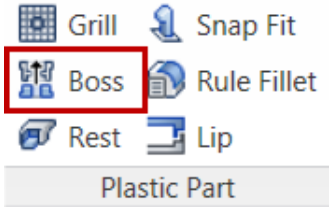


4: In order to mount rest pads to bottom of the lower case we will need to create bosses on the bottom.

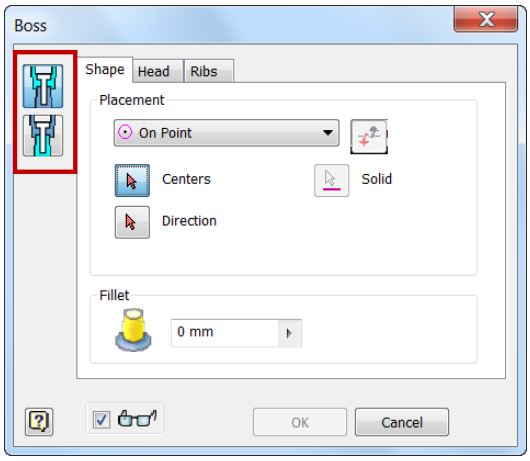
- Place a sketch on the bottom face of the case.
- Create a rectangle to establish the center locations of the bosses



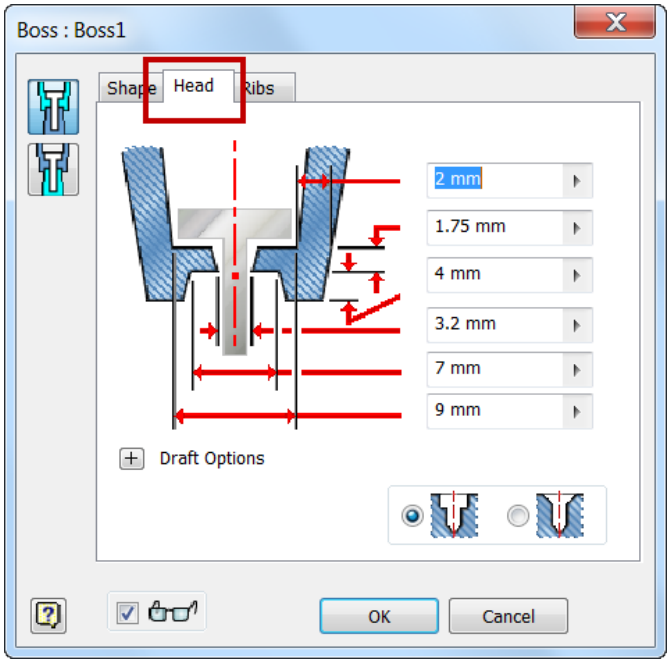
- Start the **Boss** Tool
  Model Tab | Plastic Part Panel | Boss



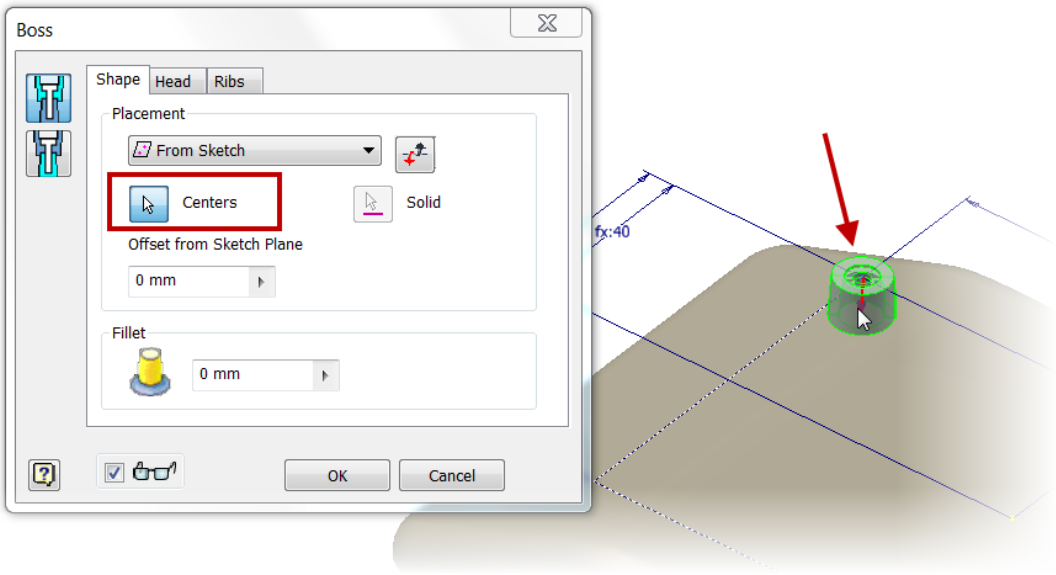
The boss tool allows for a threaded or headed style. We will use a headed style for the lower rest pad mounts.



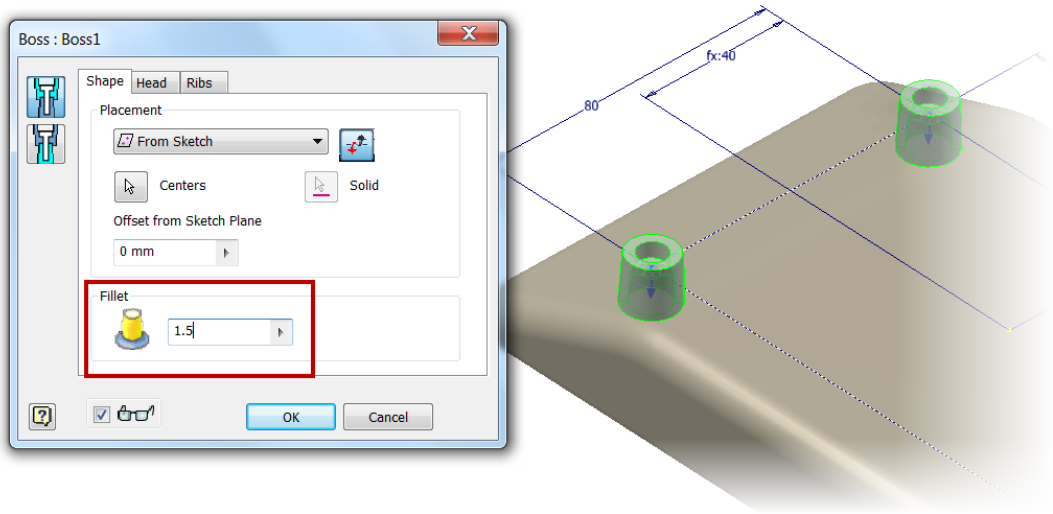
- You can control the size of the boss by changing the dimensions under the tool **Head** tab



- Return to the S**hape** tab and select the location for the bosses by clicking on the corner points of the rectangle.



- We can also place a fillet at the interfering edge to the mating surface. Enter **1.5mm** in the interesting fillet size dialog box.

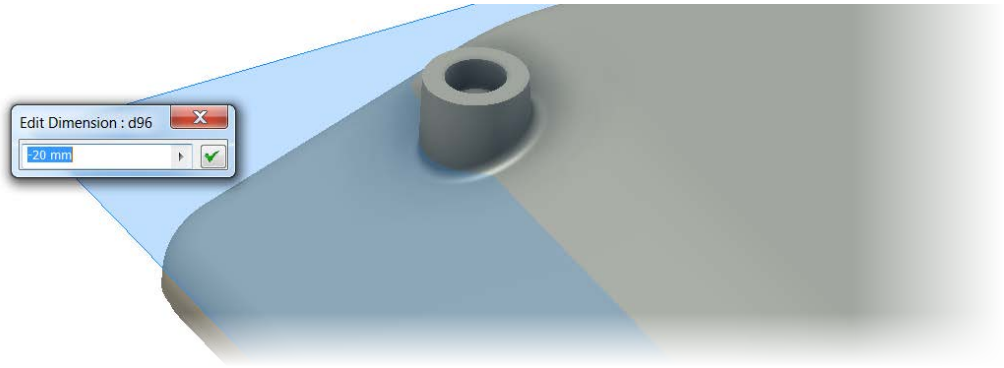


- Click **Ok** and the lower bosses will be created.

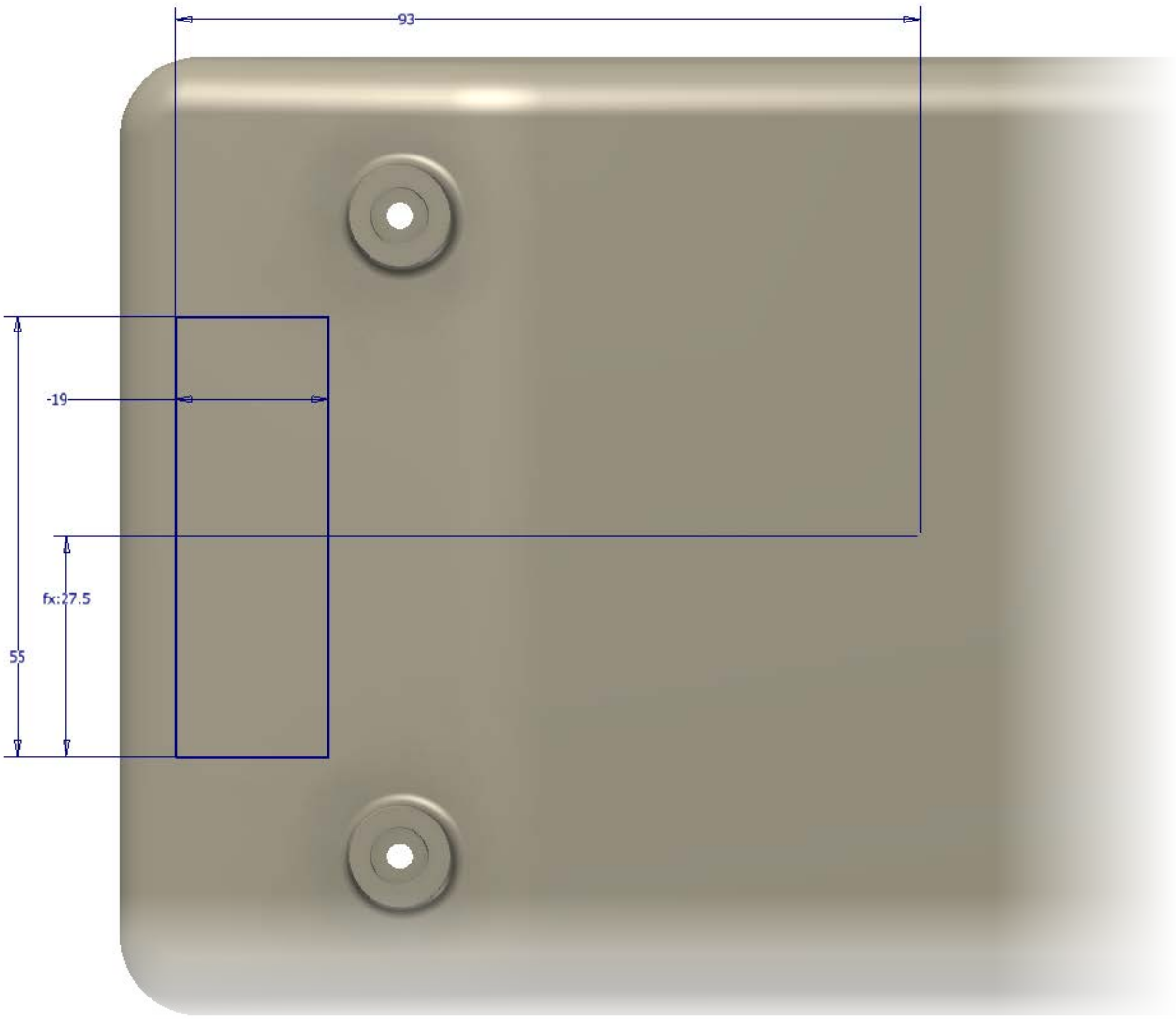
5: In order to mount and electrical connector to the case will need a flat surface near the edge of the case. By using the rest tool we can easily transform the shape and create a flat mount surface.

- Since we want the rest surface to be near the middle of the case thickness, let's create a work plan off the bottom face of the case.

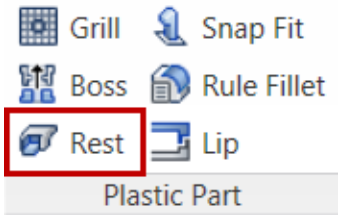
- Place a work plane **20mm** of the bottom face of the case.

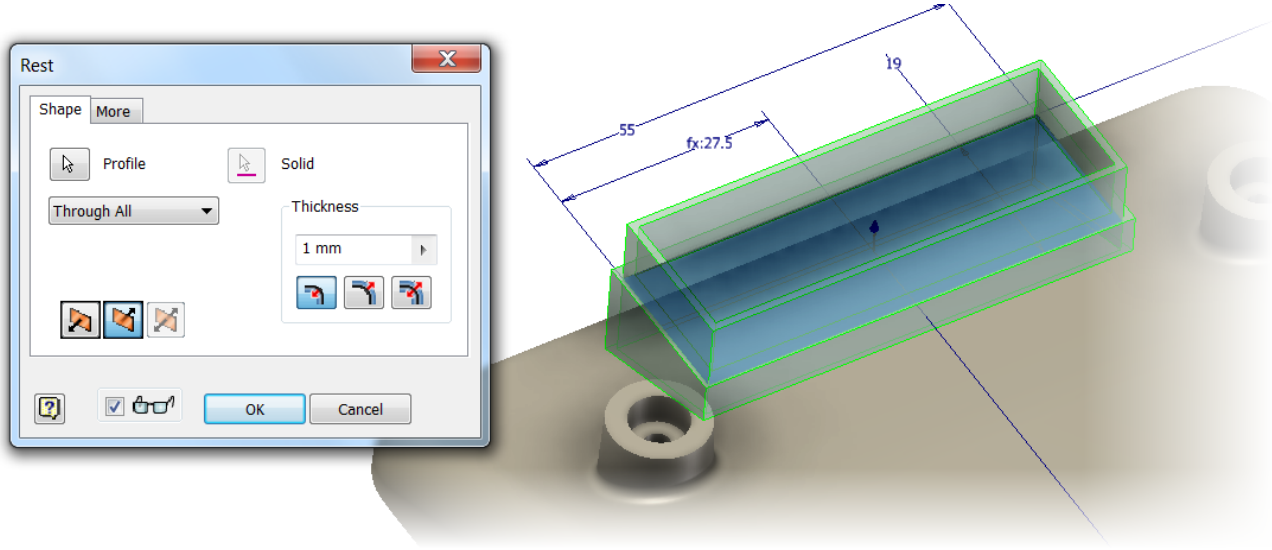


- Create a sketch on the work plane. Create a rectangle to define the size of the rest surface.

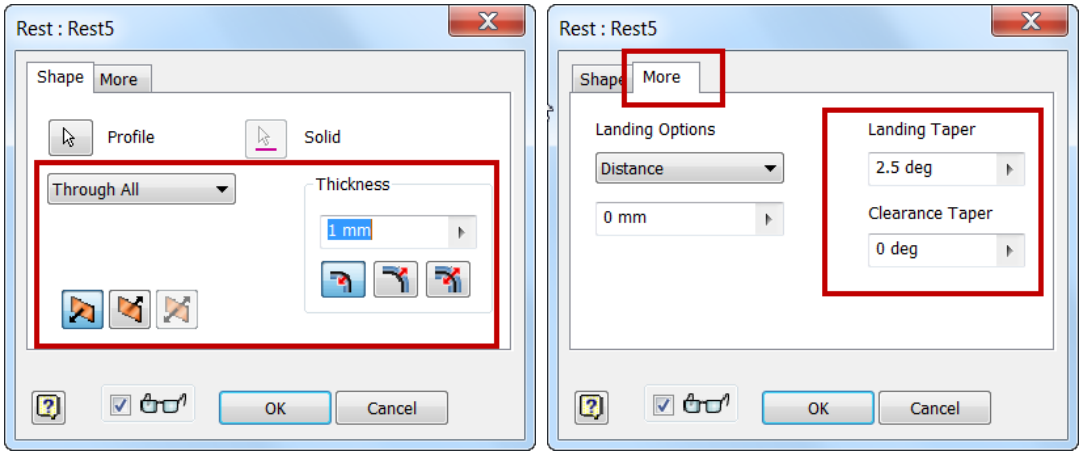


- Rest tool
  Model Tab | Plastic Part Panel | Rest

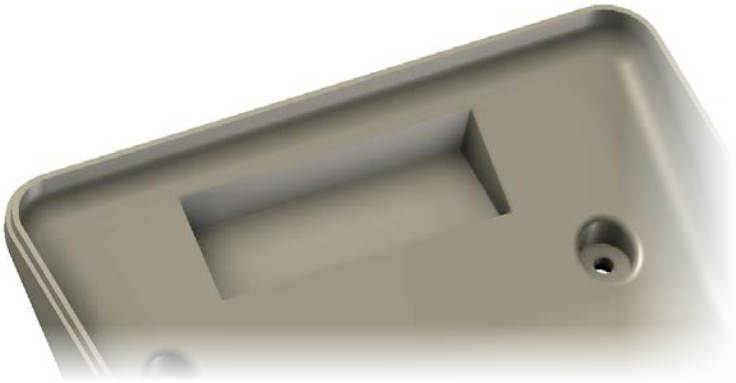




- You can control the direction and the material thickness of the rest feature. You can also control the draft angle for the feature under the **More** tab.

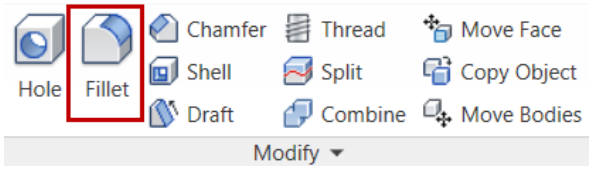


- Click **Ok** to create the rest feature. Roll the part around to see that the rest tool automatically shelled out the inner part of the case for this rest surface.

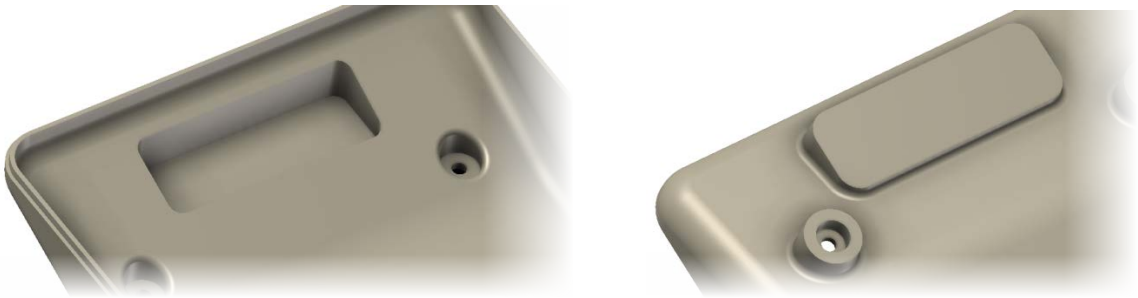


6: Use the fillet command to place some fillets onto the rest feature.

- Fillet Tool
  Model Tab | Modify Panel | Fillet



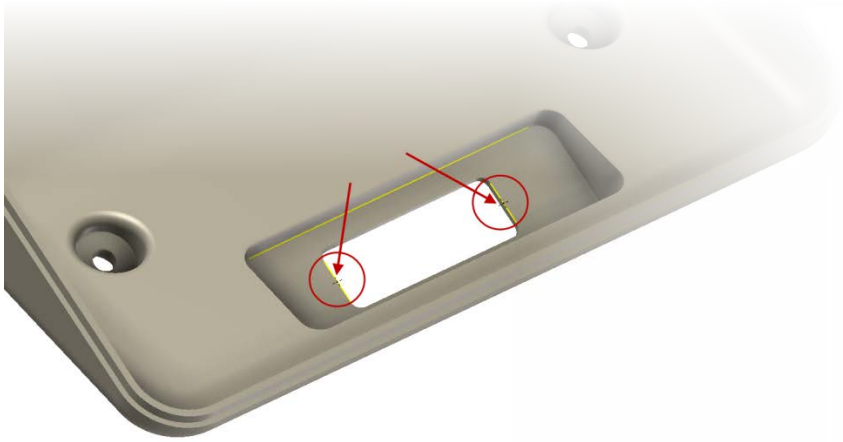
- Create fillets on desired edges of the rest feature



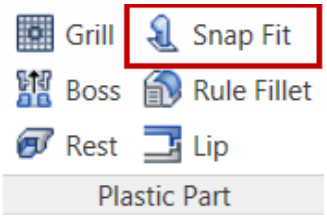
7: Using the **Extrude** command create a cut out for the electrical connector on the rest surface.

8: A very common technique in plastic part design is to design in the means to allow other items to snap into the main plastic part. A hook or loop feature is commonly used to create this snap device in the part design.

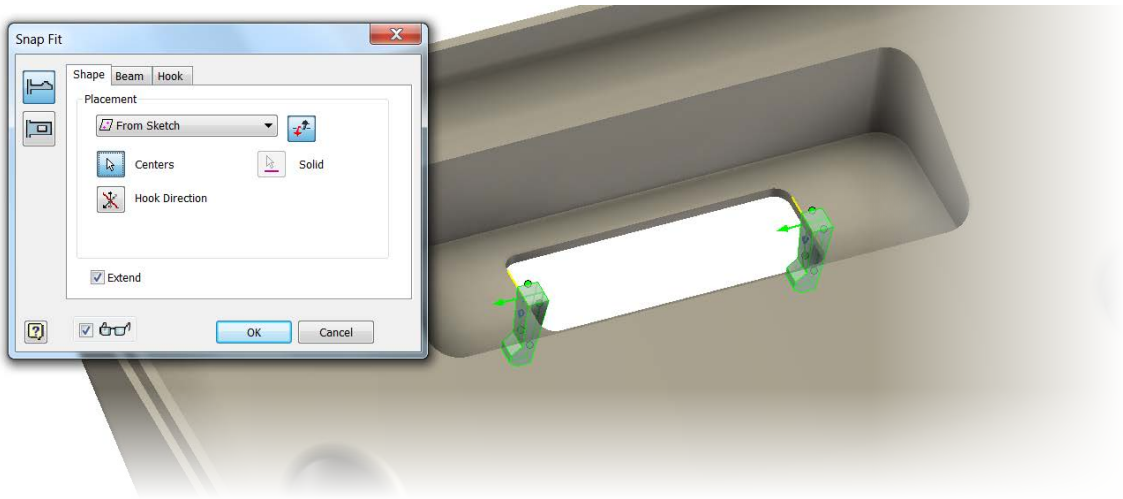
- Create a new sketch on the inside flat surface of the rest. Place center points where we would like to place a hook feature that will hold in the electrical connector.



9: Snap Fit tool
Model Tab | Plastic Part Panel | Snap Fit

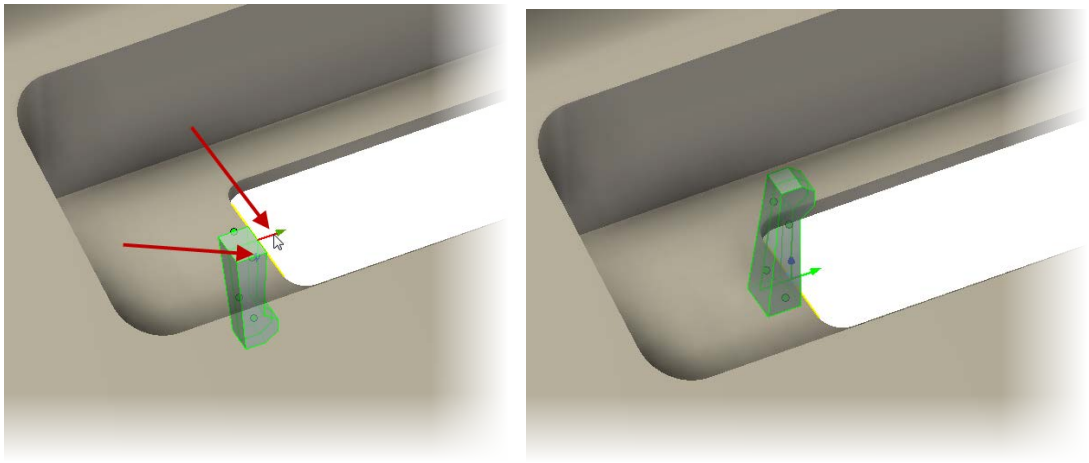


- You will see a hook feature placed onto the sketch center points.

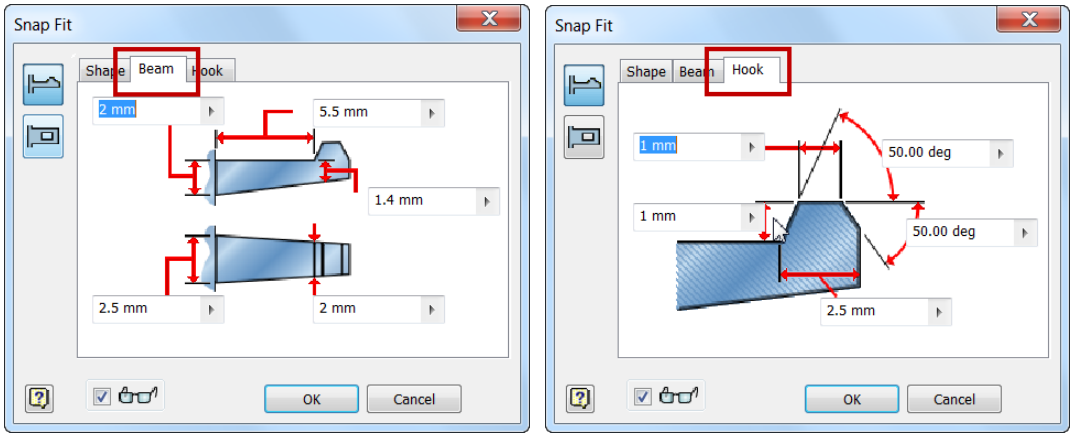


- Since we have a feature on the left side of the opening and another on the right side. We will need to deselect the right side one and complete the left side one first. Hold down the **Control** key and click the center point for the right side.

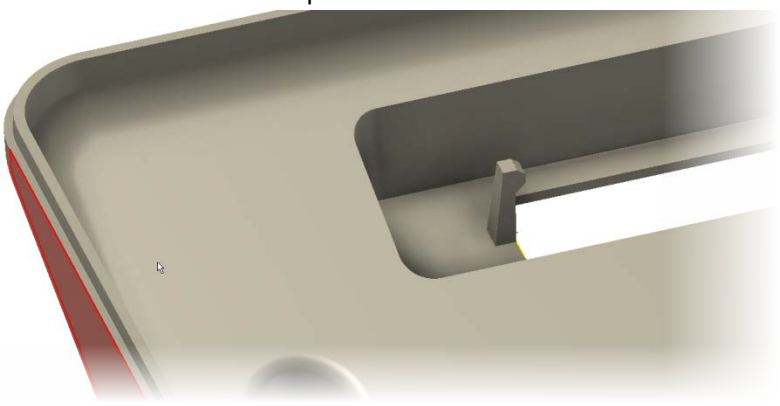
- You can control the position of the snap fit feature by clicking on the modifier arrows shown during the preview.



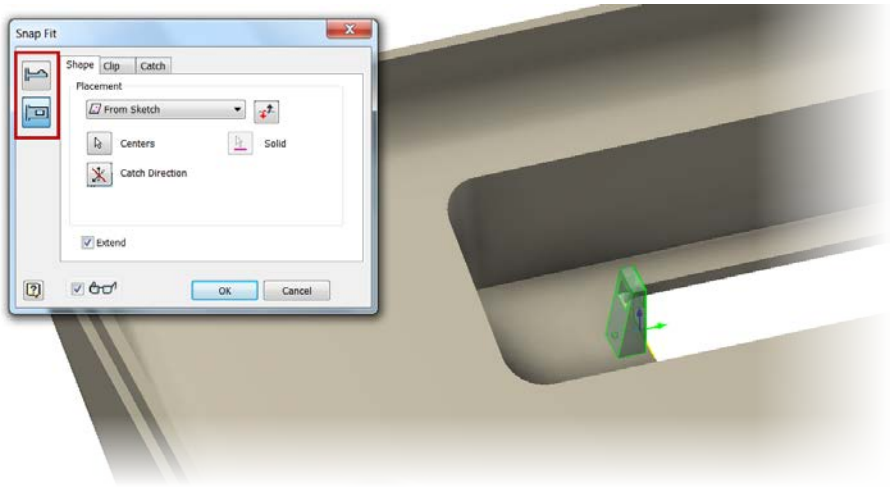
- The size of the feature can be changed by changing the values under the **Beam** and **Hook** tabs.



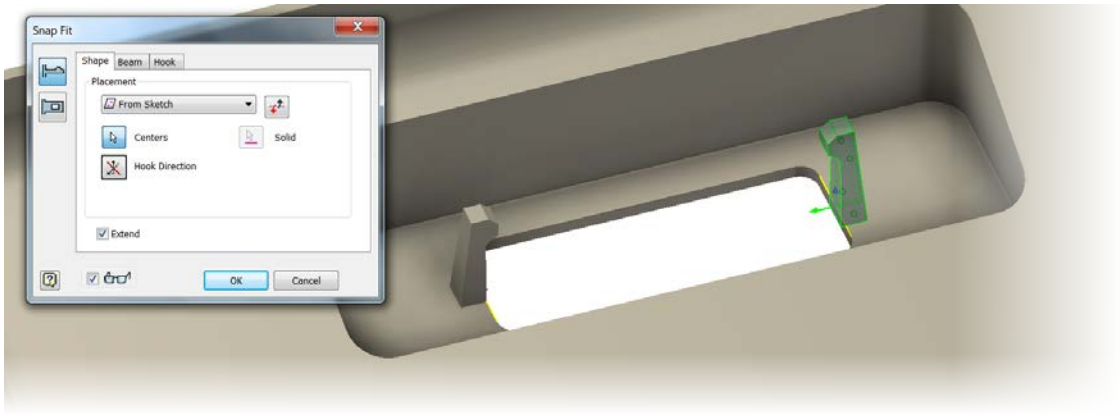
- Click **Ok** to create the Snap Fit feature



- The Snap Fit tool also allows you to select from either a **Hook** feature or a **Loop** feature.

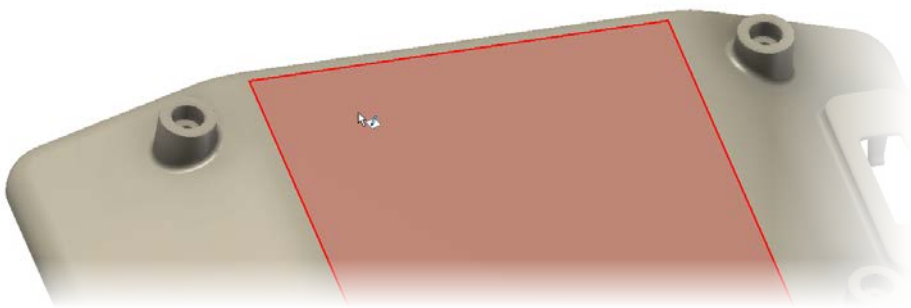


- Place a second Snap Fit feature on the right side of the rest surface.

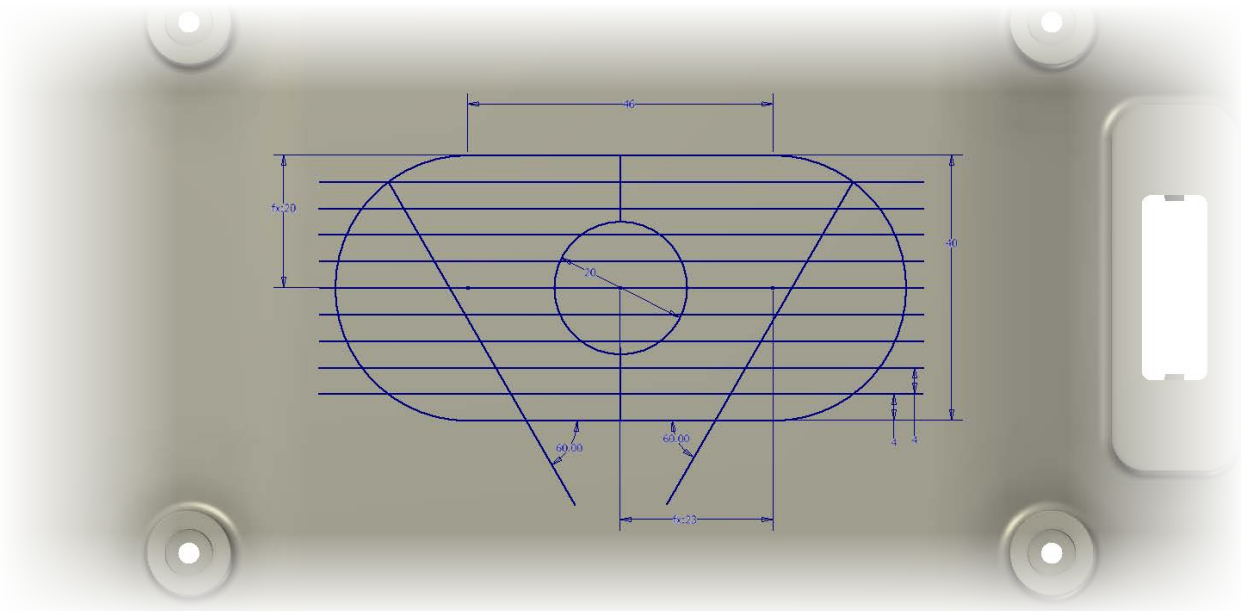


10: With a large number of plastic parts housing electronic items a grill opening for cooling is extremely common. Creating grill features is very easy using the Grill tool.

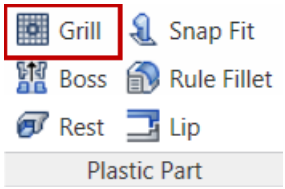
- To use the grill tool you will need to start on new sketch on the face that requires the grill opening. For this exercise select the bottom face of the case for your sketch.



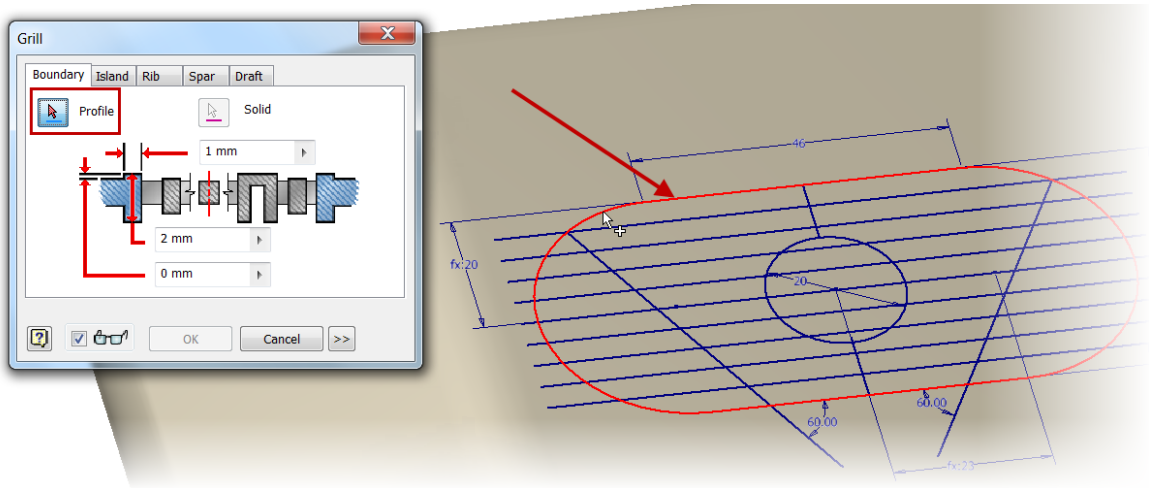
- The different parts of the grill opening are defined by geometry from a sketch. Begin by sketching the shape and look of the grill opening.
  - This is one of the few examples where using a pattern in the sketch is an acceptable workflow.



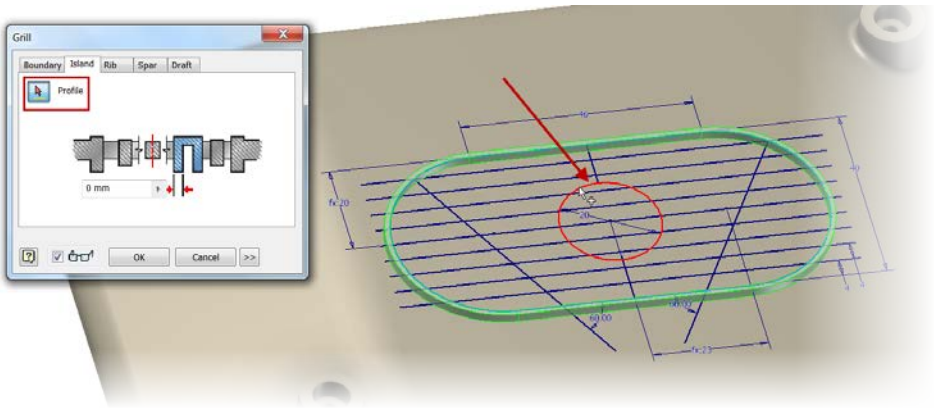
- Grill tool
  Model Tab | Plastic Part Panel | Grill



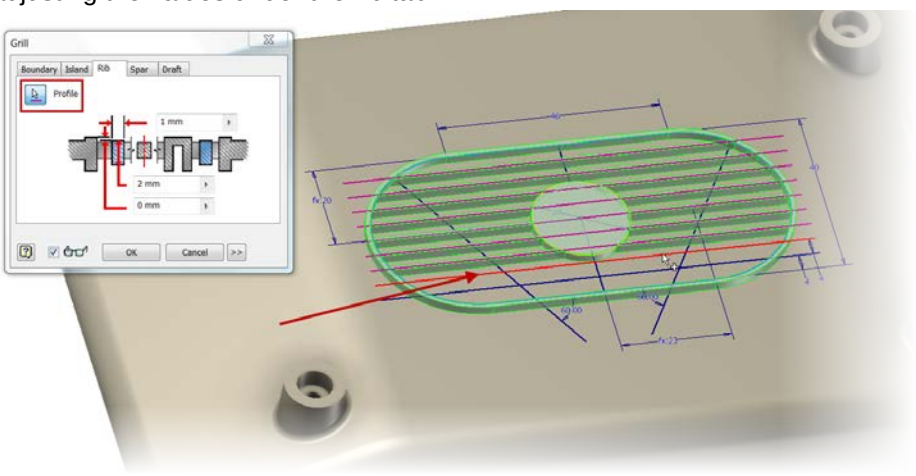
- The first part of the grill opening is to define is **Boundary** shape. Click on the outer shape of the grill. You can change the size of the outer perimeter shape by changing the values in the dialog boxes.



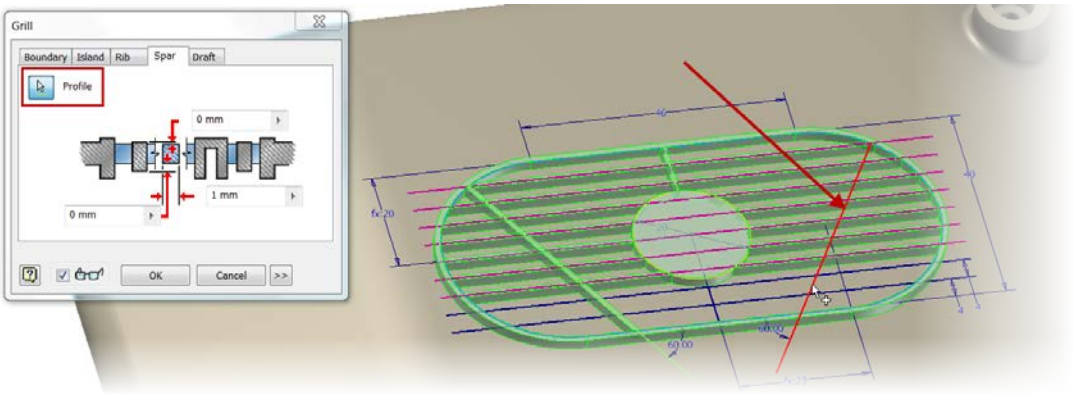
- To define the center portion of a grill opening you can select a part of the sketch that will be looked as the **Island**.



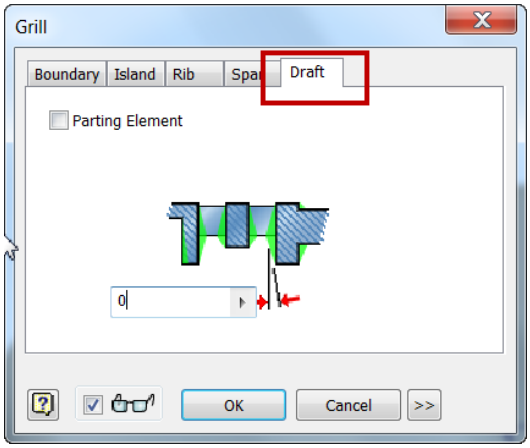
- The main portion of a grill opening is all the separate slot openings. We will establish these by moving to the **Rib** tab and selecting the horizontal lines. You can control the size of these ribs by adjusting the values under the rib tab.



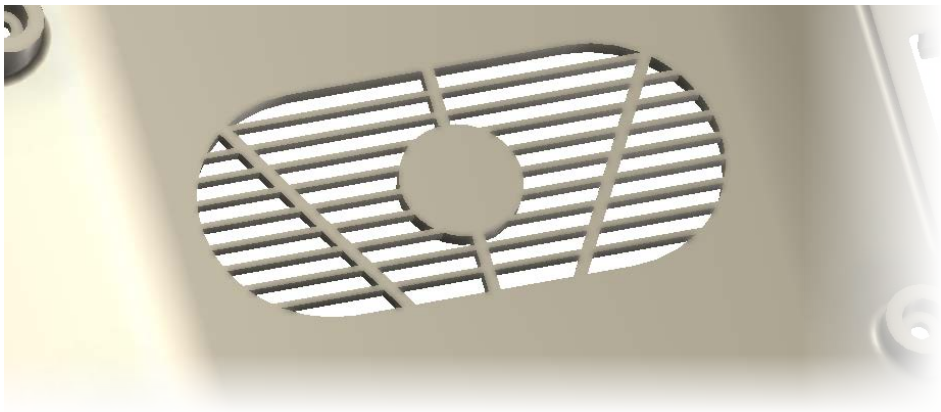
- To create support ribs for the slot openings. Switch over to the **Spar** tab and select the vertical shapes from the sketch.



- You can also control the draft angles of the opening by changing the values under the **Draft** tab.



- Click **Ok** to create the grill opening the bottom of the case.



11: We have complete creating a plastic case from a solid shape.