
Interactive Solid Modeling

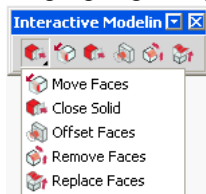
Interactive Solid Modeling opens up a greater possibility of modifying Solid models in a Non-conventional way. It enables the user to perform Free-Form editing without being limited by the parameters that govern the shape of a feature or a Model, yet controlled by dimensions. The editing process can be as simple as selecting a face and dragging it to a new location. All other faces linked to the selected one will change their shape to keep the model consistent. Here are five different Interactive Modeling commands that can be used effectively to alter the shape of solids without modifying any of the feature governing parameters like its profile or driving dimensions. Let's see more on these commands individually!!

Table of Contents

1. Step 1: Move Faces	1
2. Step 2 : Extend Faces/Close Solid.	5
3. Step 3 : Offset Faces	9
4. Step 4 : Remove Faces	11
5. Step 5 : Replace Faces	16
6. Step 6 : ISM on a Mechanical model	21
7. Step 7 : ISM on a Plastic model	27

1. Step 1: Move Faces

In this step we are going to learn how to use the Move Faces command and various options along with it. Before we get going, let's pull up the Interactive Modeling Toolbar from the Tools > Customize menu.

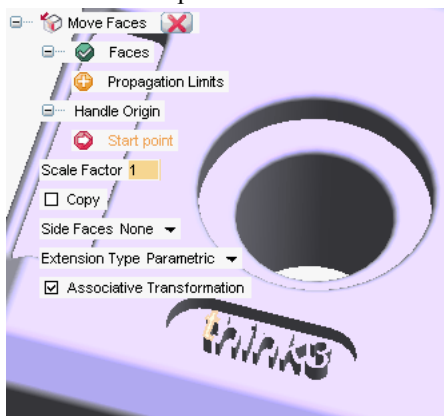


The Move Faces functionality is used to Translate or Rotate a set of user-selected faces. Once the command is applied, the faces are moved while keeping the geometry consistent. It also gives you the option of copying and scaling a selected set of faces while retaining the original feature intact. As the selected faces are transformed, other faces that are connected to the selected ones also change their shape to maintain the consistency of the model. The Move Faces command makes use of the translation handles to specify the magnitude and direction of the face transformation.

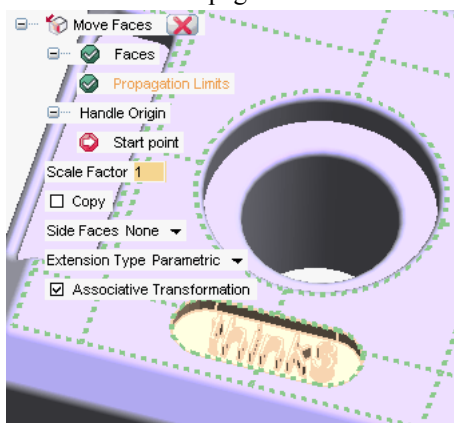
Load the model file Move_Faces.e3 from the installation folder which usually will be C:\My Training\... and activate the command **Modify** ➤ **Interactive Modeling** ➤ **Move Faces**.



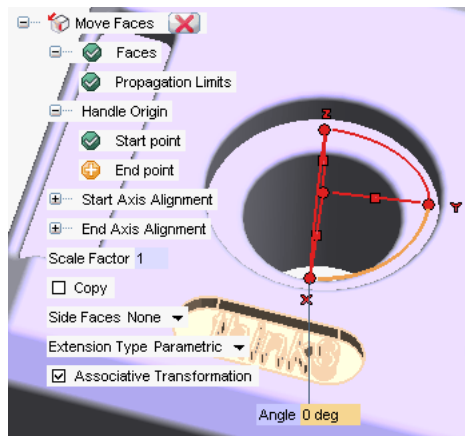
- Select the top face of letter "t" as Faces.



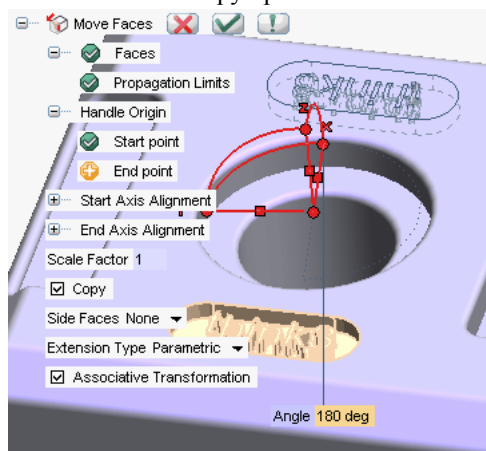
- Click on Propagated limits and select the top face of the solid as shown in the image below.



- Set the Start point as the centre point of the hole and set the Start Axis Alignment X-Axis - line and select an edge of the Linear Slot.



- Check the ☒ Copy option and rotate about Z-axis by Angle-180 deg. Let the scale Factor be 1.




- Click ☒ OK to complete the operation. The result is as shown below.

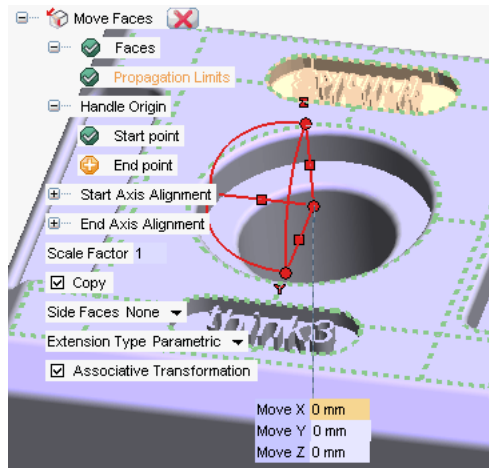


Propagation Limits.

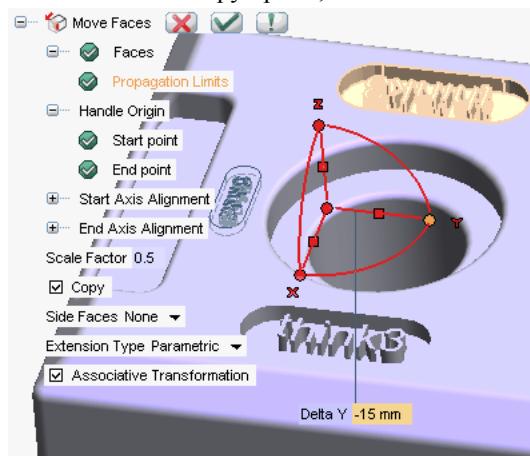
The Propagation Limits option is available in the selection list of the Interactive Modeling commands except for **Extend Faces/Close Solid**. This option assists in the face selection process. At times there can be multiple faces


that radiate out from a single face. The Propagation Limits option can be used to select all those faces in a single step. The faces selected for the Propagation Limits will act as a limit such that the command will be applied on all the faces lying between it and the face selected for the Faces selector.

Again activate the command Move Faces and select the Copied Faces as  Faces and set the Start Point as in the previous operation.



- Check the ☒ Copy option, Enter a scale factor of 0.5.



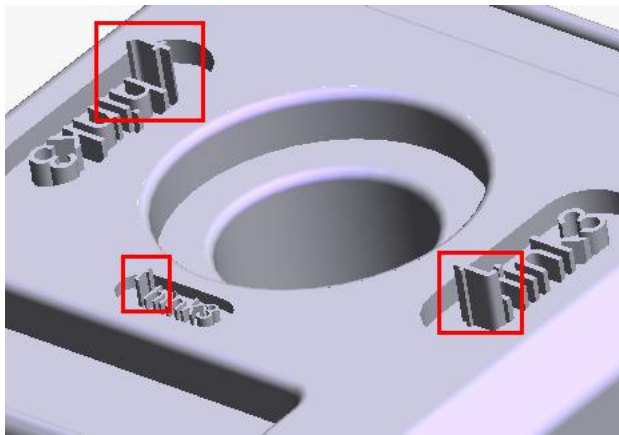
- Rotate about the Z-axis by an Angle-90 deg and translate along the direction Y by Delta-15 mm. Hit  OK to see the result. In this step we Scaled, Moved and Rotated the think3 Logo by a scale factor of 0.5, -15 mm and -90 deg.



Since the option Associative Transformation, available under More Options, was checked any modifications that are made to the parent features will be updated in the all its copies.

Associative Transformation.

For E.g. Redefine Linear Sweep for the alphabet "t" and change the Depth value to 12 mm and hit Rebuild. You can see the same changes being updated in all the copies.



2. Step 2 : Extend Faces/Close Solid.

In this step we will learn about another important command, **Extend Faces/Close Solid** in ISM. The **Extend Faces/Close Solid** functionality is used to close an open solid by extending and trimming the faces adjacent to the open boundaries.

In case the open solid contains multiple open loops, it is possible to selectively close some of the loops while leaving the others open. In this case only face extension will take place to close the open loops and no close solid will be formed. To facilitate this functionality, the **Extend Faces/Close Solid** selection list has the Mode drop-down list with the following two options,

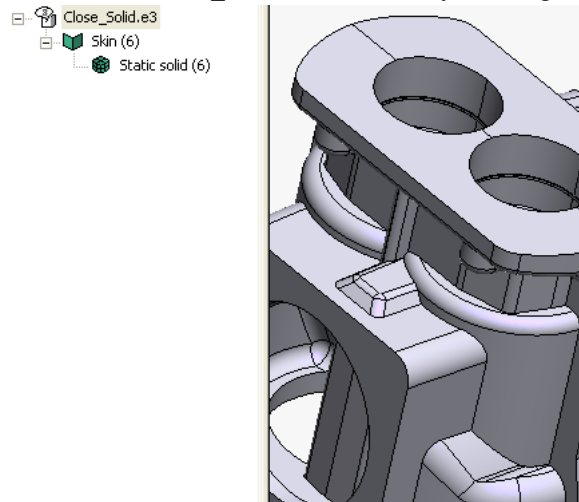
- Global: All the open loops will be closed to form the close solid.

- Local: Only the selected open loops will be closed to form the close solid.

NOTE:-

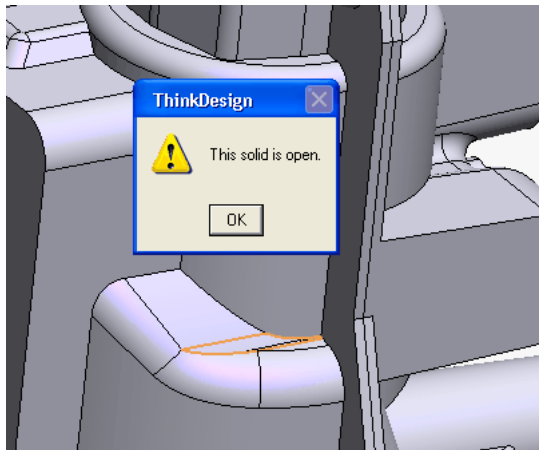
The Extend Faces/Close Solid functionality is used to close an open solid by extending and trimming the faces adjacent to the open boundaries. If the extended faces are not converging (for example because they are parallel) the feature will fail. No new face is created to close the gaps.

Load the file Close_Solid.e3 from the My Training folder.

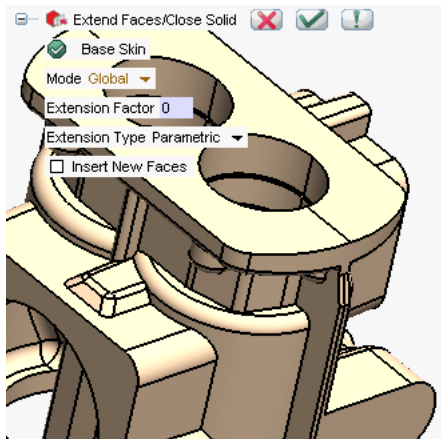


From the History Tree you can see that the model we have opened is a Skin (Open Solid). Let's make a quick Topology check to find out open edges.

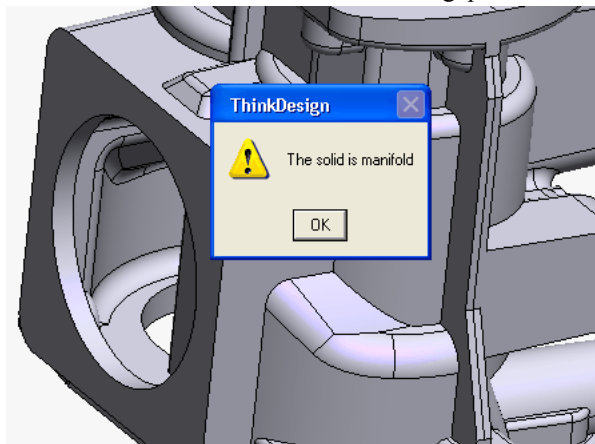
- Activate the command **Tools** → **Modeling** → **Check Manifold** and select the Skin.



- thinkdesign highlights the edges that open as shown in the above image.
- Activate the command Close Solid and select the Open Solid as Base Skin. Let the Mode be Global

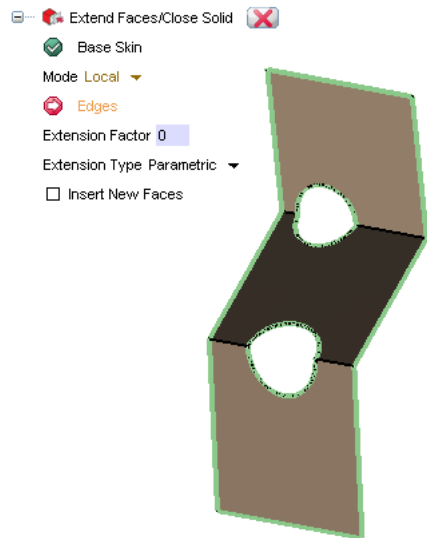


- Click OK to complete the operation. Zoom in at the edges which were open, you can see that the adjacent faces has been expanded to close the gap.
- Again check whether the solid has been closed completely using the command **Check Manifold**. This shows that the Solid is Manifold and all the gaps have been closed.

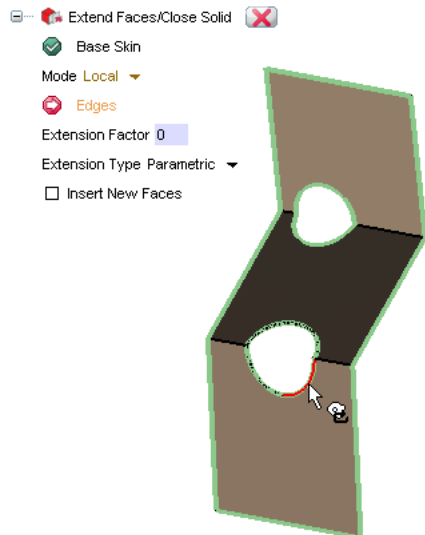


Let's try Close Solid command on a skin that has multiple openings. As said earlier we have the option by which we can select the Open loops manually with the Local Mode.

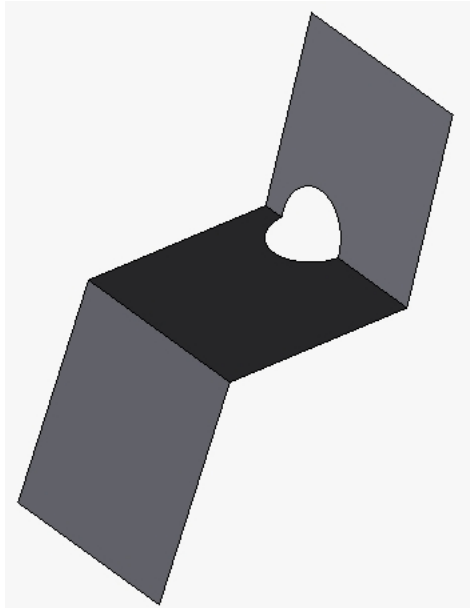
- Load the model file ISM_Selective_closure.e3 and activate the command Close Solid.
- Select the Open Solid as Base Skin and change the mode to Local. thinkdesign highlights all the Open Edges.





- Select the open edges that forms a Closed loop as shown below. Note that even if you select a portion of a loop thinkdesign automatically selects the complete loop.



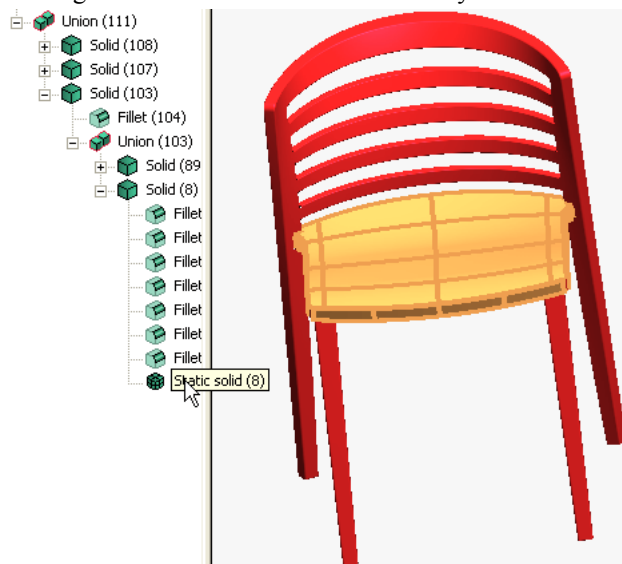
- Hit OK and result is as shown below.



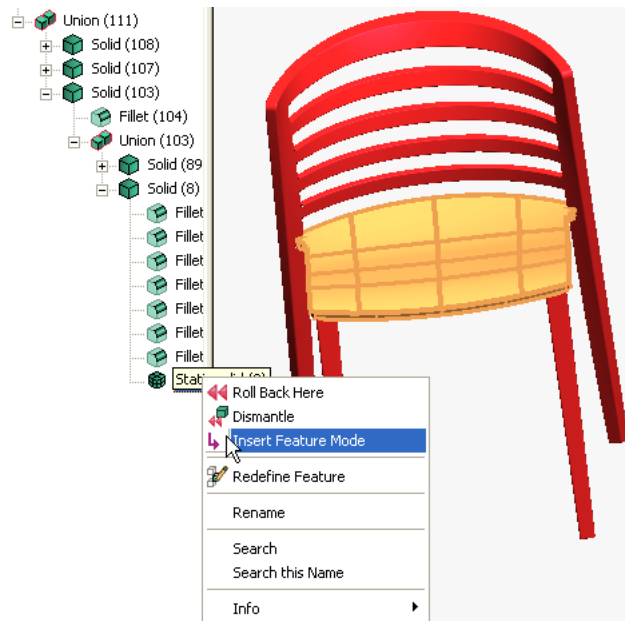
3. Step 3 : Offset Faces

In this step we will learn about the **Modify**  **Interactive Modeling**  **Offset Faces** command. The **Offset Faces** functionality is used to offset a set of user-selected faces by a specified value in a particular direction. This functionality can be effectively used to change the radius of holes and adding thickness to objects.

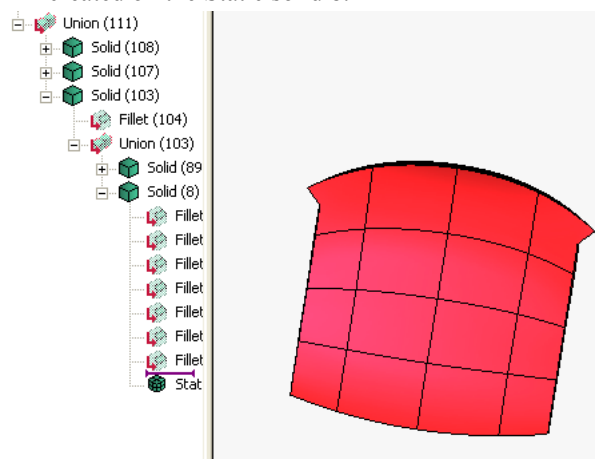
- Load the model file Offset_Chair.e3.
- The goal of this task is to increase the thickness of the seat of the chair. Let's add some material to the seat. This material has to be added at the initial stage of this model.
- Right Click on the model in the History tree and select **Expand All** so as to select the base solid of the seat.



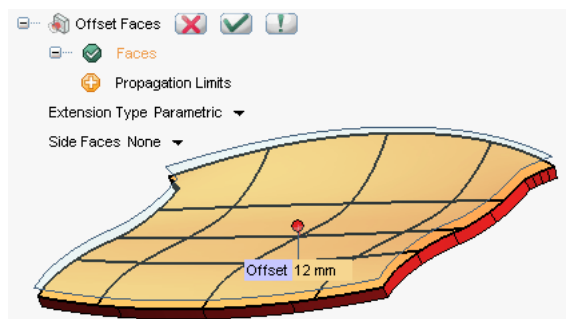
- Right Click on the Static solid 8 and select **Insert Feature Mode**




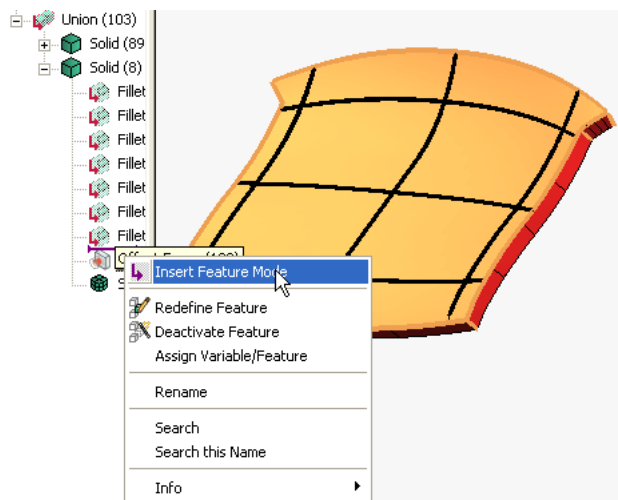
- Once you do Insert Feature mode, thinkdesign rolls back in the history without losing the features that are created on the Static solid 8.



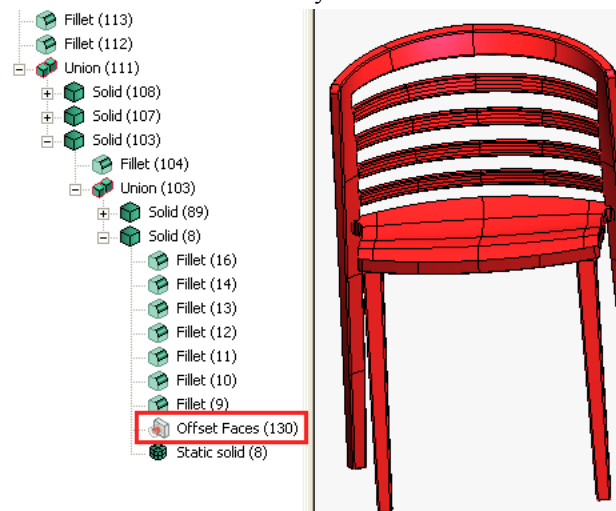
- Activate the command **Modify** → **Interactive Modeling** → **Offset Faces**. Select the Faces to be Offset as shown and enter an Offset 12 mm.



- Hit OK to add the material.
- Remove Insert Feature mode from the History Tree and  Rebuild the model to update the changes made.





The final result of the **Offset Faces** command looks like this. You can modify the offset by redefining Offset faces feature from the History tree.



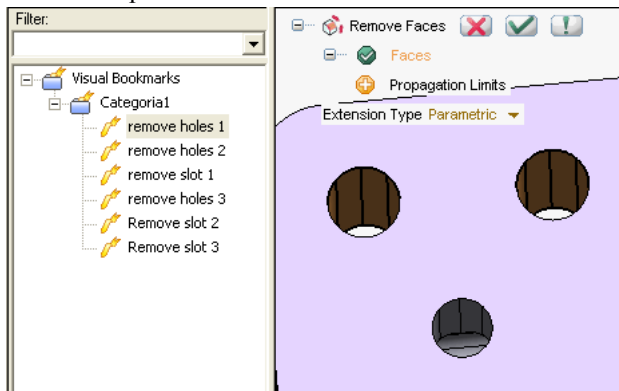
4. Step 4 : Remove Faces

The **Remove Faces** functionality is used to remove a set of user-selected faces. This is very helpful to remove unwanted details like fillets, chamfers, holes and slots from the model. As the selected faces are removed, their adjacent faces are extended and trimmed to fill up the void and maintain the model consistency.

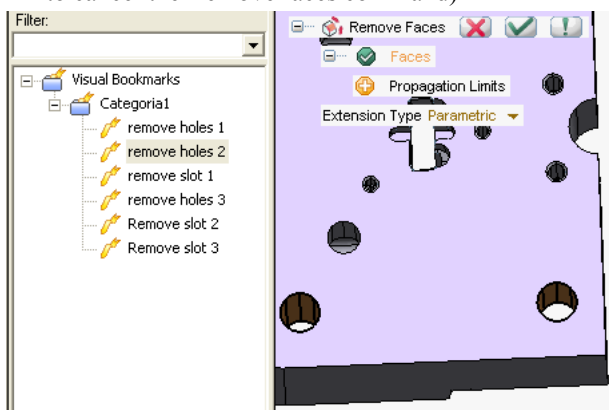
- Load the model file Remove_Faces.e3 and activate command **Modify**  **Interactive Modeling**  **Remove Faces**.



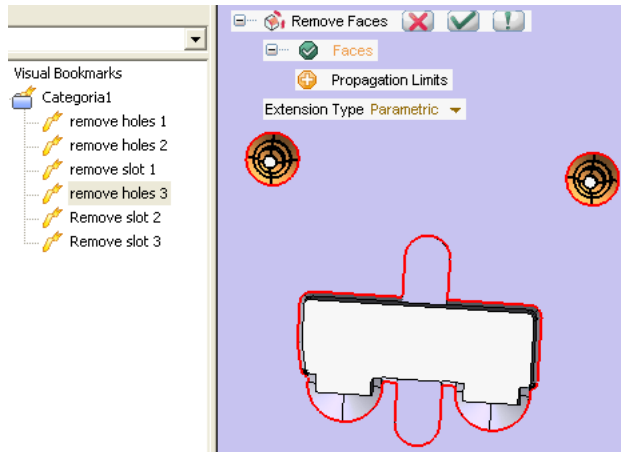
- To facilitate the easy selection of the faces, you can use Visual Bookmarks. With Remove faces command active open the Visual Bookmark tree and activate the Visual Bookmark Remove holes 1 as shown



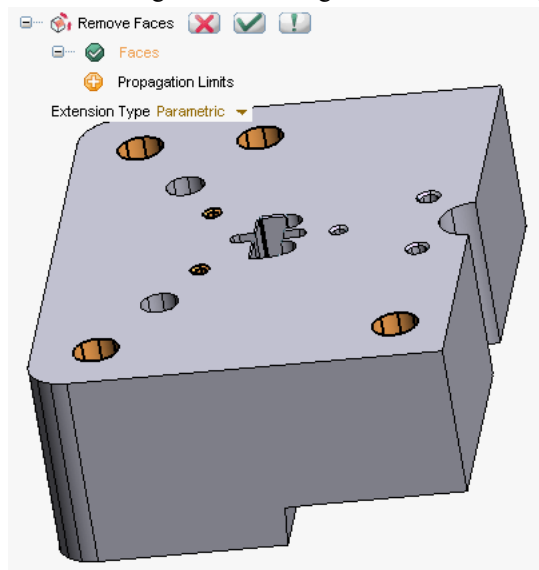
- Similarly activate Visual Bookmark Remove holes 2 and select the faces as shown below (Remember not to cancel the Remove faces command)




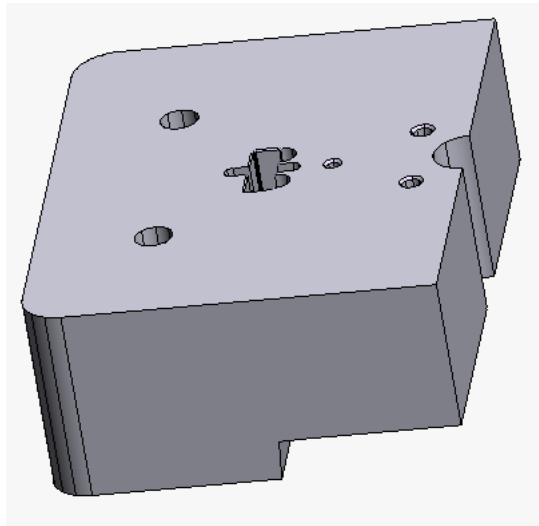
- Again activate the Visual Bookmark Remove holes 3 and select the faces as shown below



After selecting the holes using Visual Bookmarks, the Faces selection should look like this,



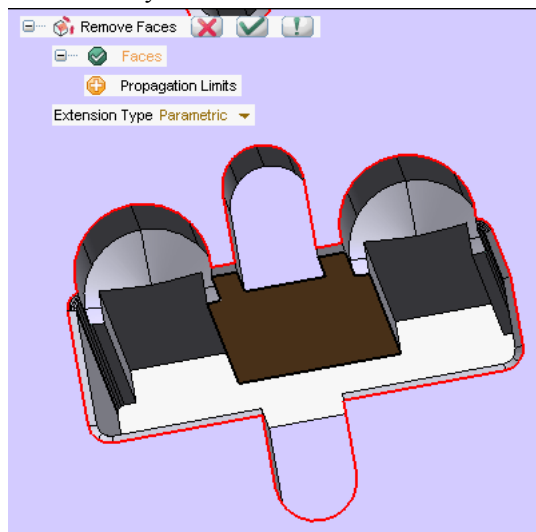
Hit  OK to remove the selected holes.




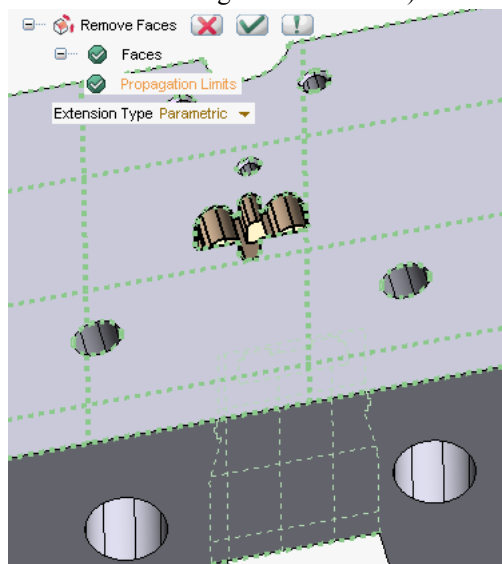
Let's remove some of the slots that are there in this model using the same approach as above.

- Activate **Remove Faces** command and from Visual Bookmark tree activate Remove Slot 1

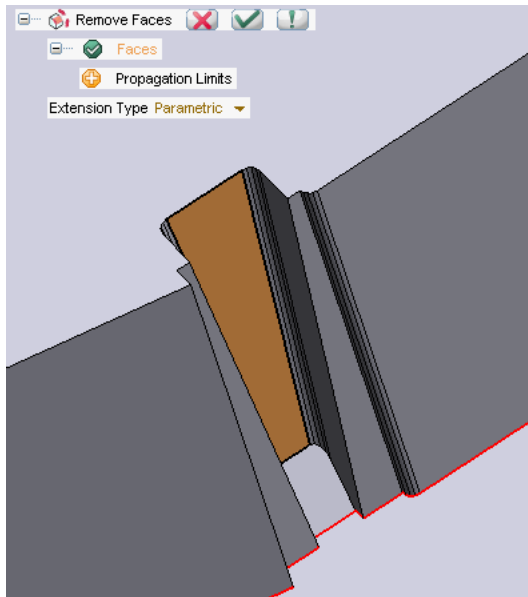
- Select any one face of the slot as shown




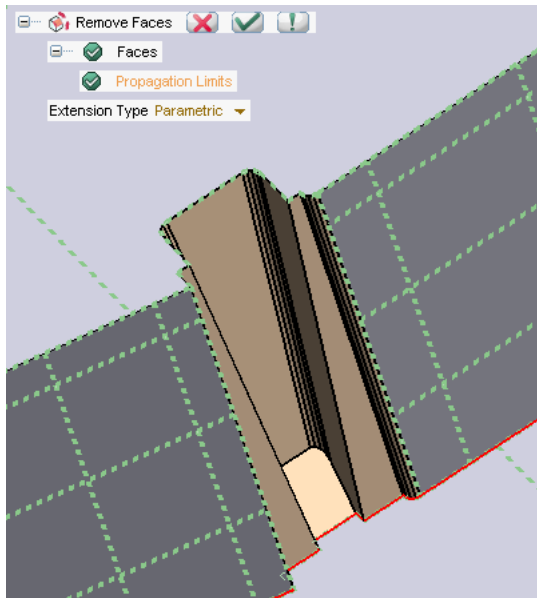
- Select top face of this solid and bottom face of this slot as propagation limits as shown below (the faces that are shown with green dotted lines). Click  Apply to remove the slot.




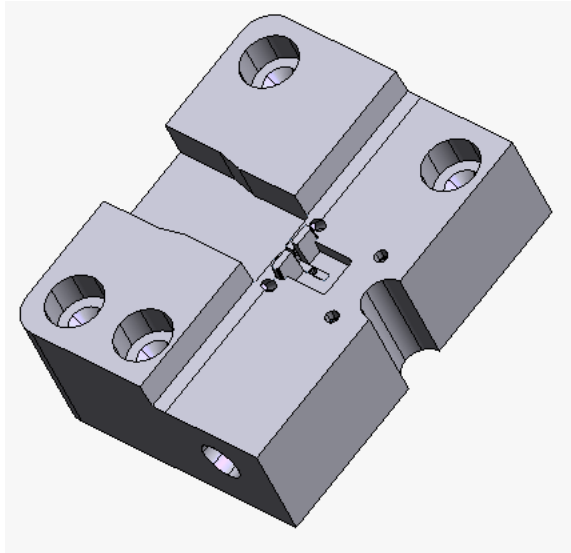
- Activate Visual Bookmark Remove Slot 2.
- Select one of the faces of the Slot as shown below.



- For Propagation Limits select faces as shown below. Hit  Apply to remove this feature.

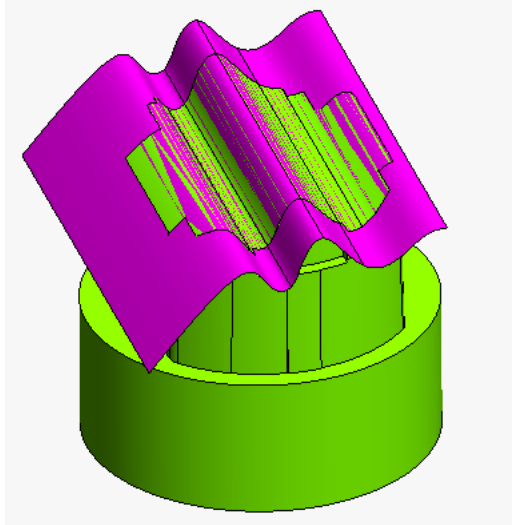


- Activate the Visual Bookmark Remove Slot 3, the faces and the Propagation Limits selection procedure is same as the previous command. Hit  OK to complete the removal of the faces. The resultant model is as shown below.

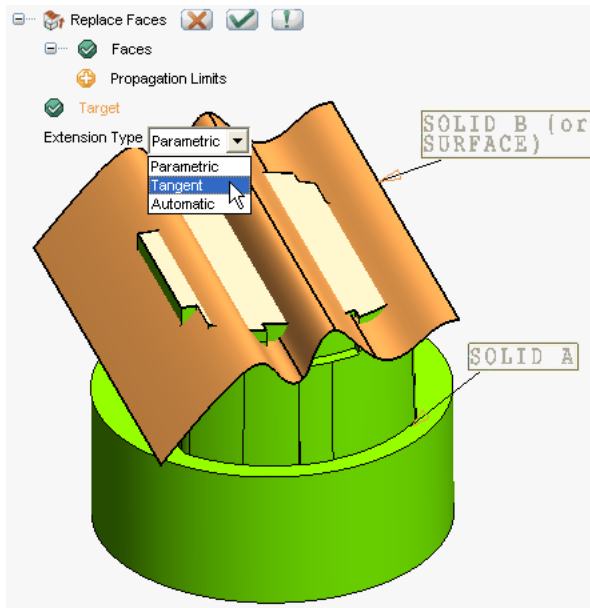



5. Step 5 : Replace Faces

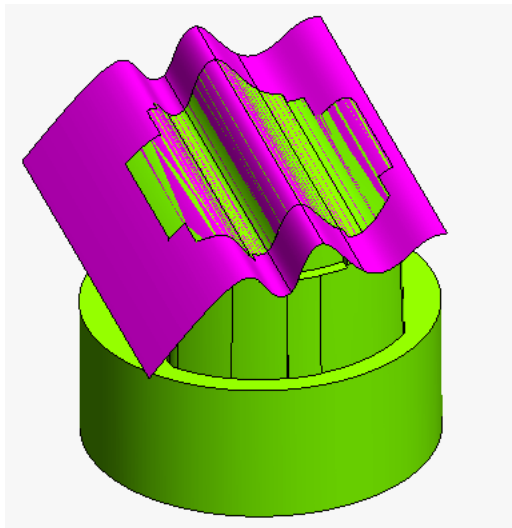
The **Replace Faces** functionality is used to replace a set of user-selected faces with another one. The face to be replaced will undergo the necessary transformations to make the replacement feasible. The selected face will take up to contour of the one it is being replaced with. Other faces that are attached to the selected face will be extended and trimmed to keep the model consistent.




- Load the model Replace_Faces.e3, Activate the command **Modify** ➤ **Interactive Modeling** ➤ **Replace Faces**.
- Select the top surface of the Solid A as Faces and Solid B as Target. The Target could be a Face of a Solid or a Surface or a Skin (Open Solid).
- Select the extension type as Tangent.

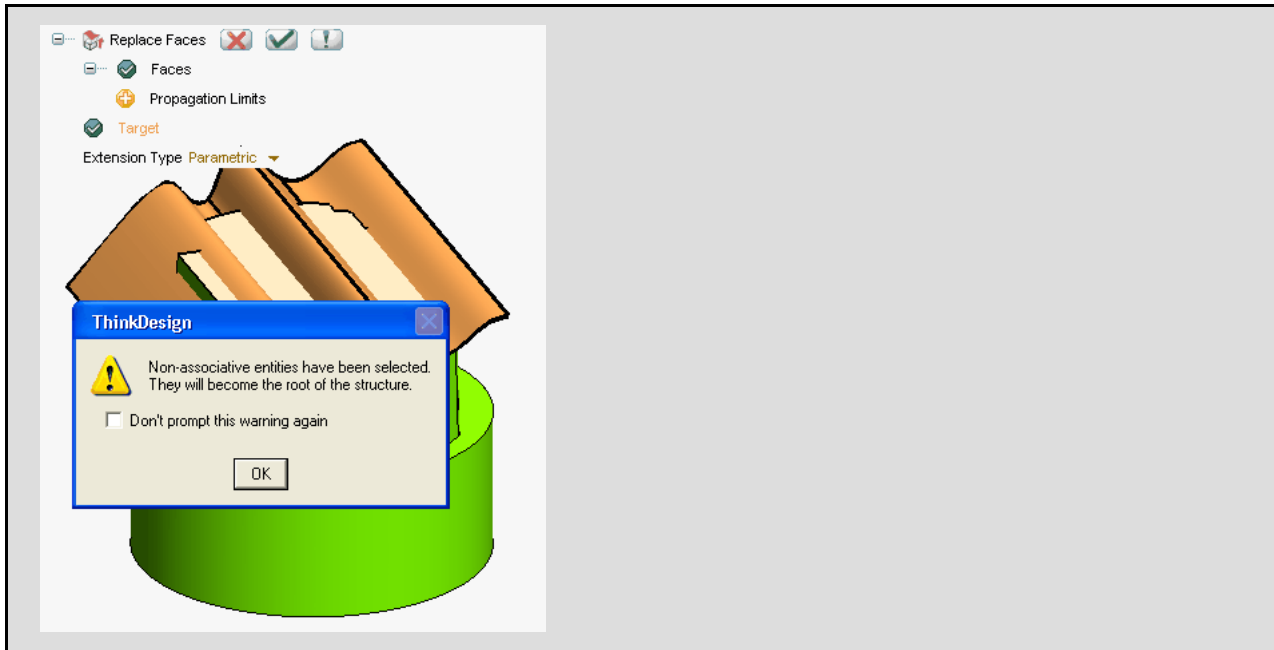


- Hit  OK to replace the face.

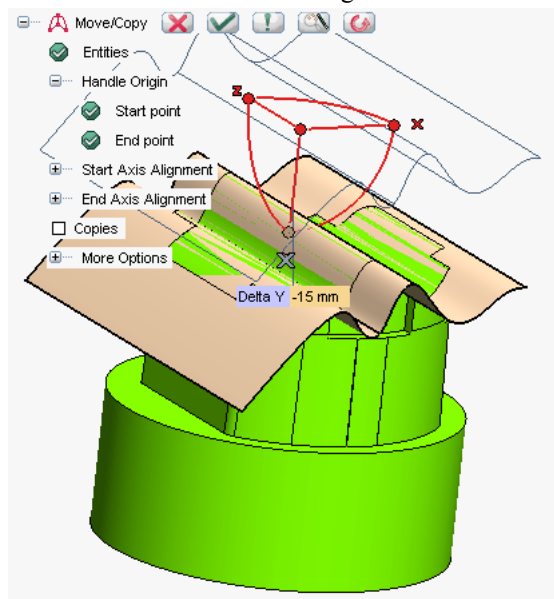



NOTE:

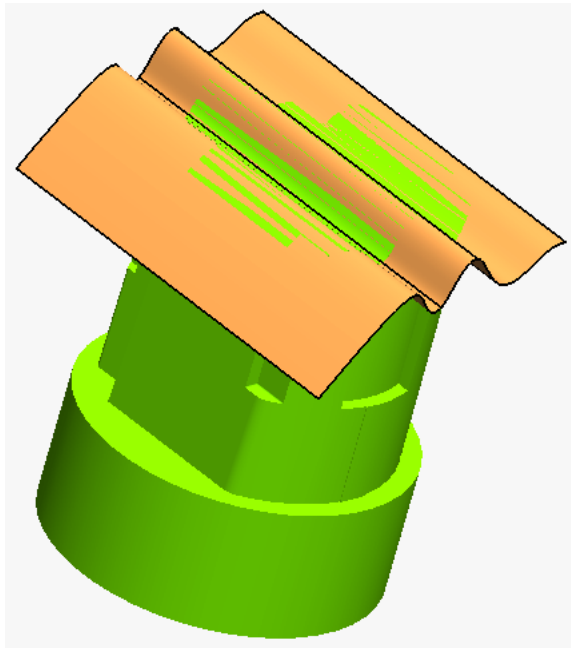
The  Target could be a Surface, Skin (Open Solid) or face of a Solid. When selected a surface, Replace faces command will automatically convert the surface as Skin with a warning message as shown below.



The **Replace Faces** command is associative which means any modification made on the Target face will be updated in the final model. For E.g.: Transform the Target in Y-direction by Delta Y15 mm.

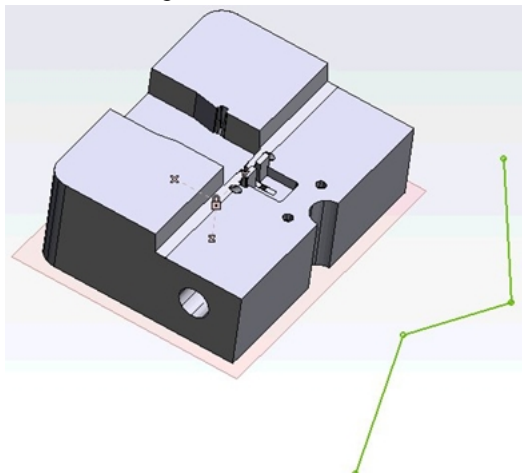


- Hit  OK to see the changes.

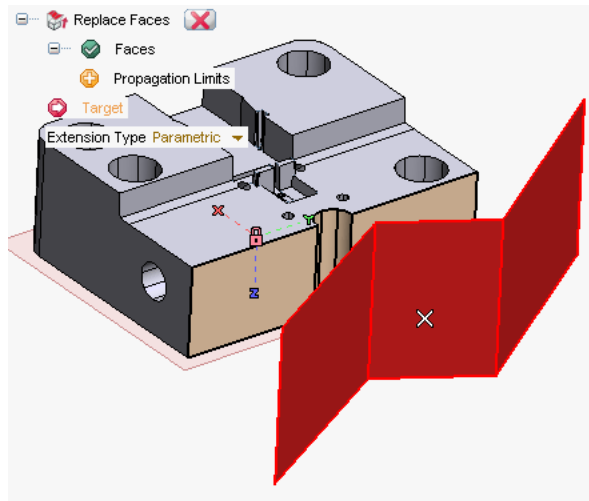


Let's try with one more example.

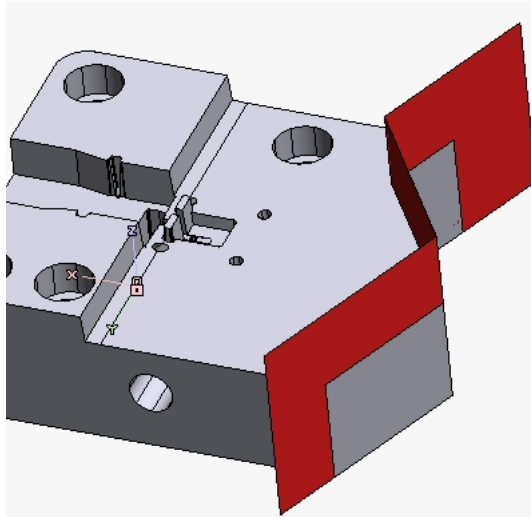
- Load the model file Model_remove.e3
- Place workplane on the bottom face of the solid and create an Open profile as shown in the image below.



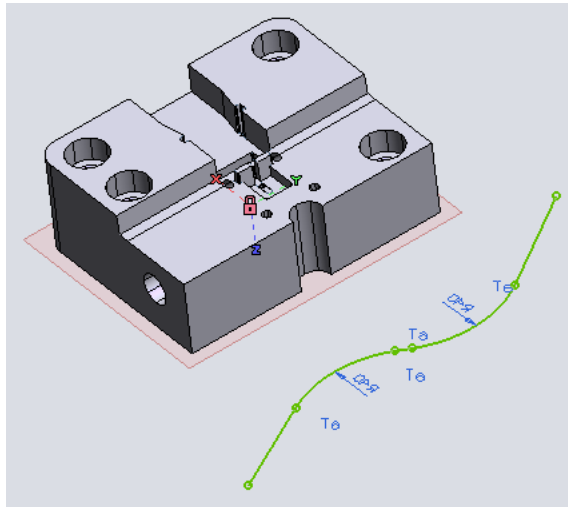
- Create a linear Solid from this open profile, the result will be a Skin(Open Solid).
- Activate the command **Replace Faces** and select the 3 highlighted faces as faces to be replaced




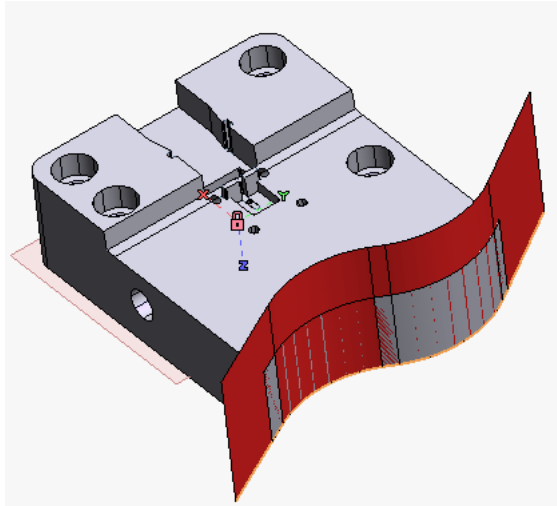
- Select the Skin as  Target and hit  OK.



- Edit the Profile and add 2 fillets of radius R 40mm.



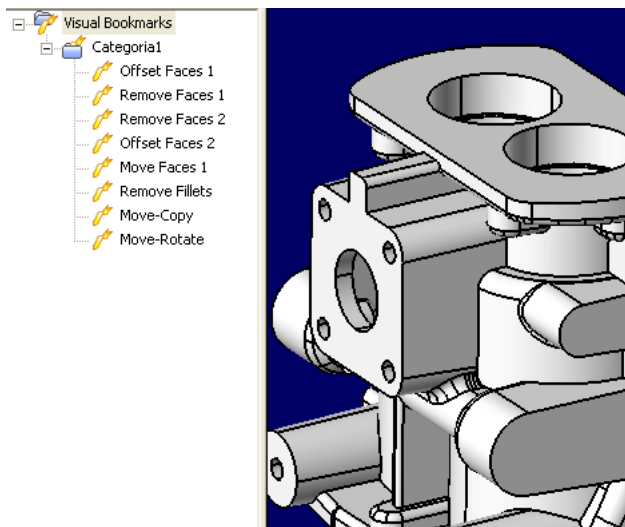
- Switch to Model and  Rebuild to update the changes. The resultant solid looks as shown below.



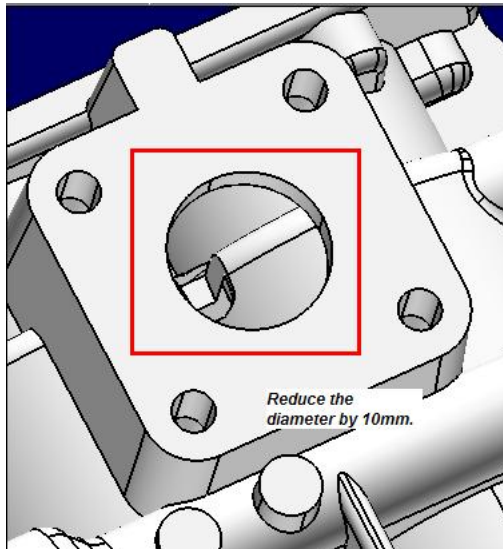
6. Step 6 : ISM on a Mechanical model

In this step let's try these ISM commands on a Mechanical Model.

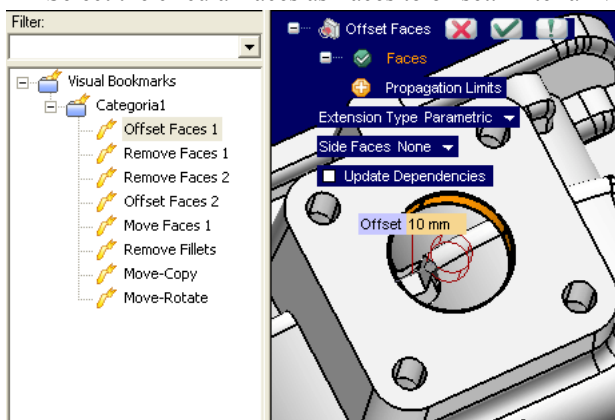
- Load the model file Mechanical_model.e3.
- To facilitate the easy selection of the faces make use of the visual bookmarks that are provided with this model.




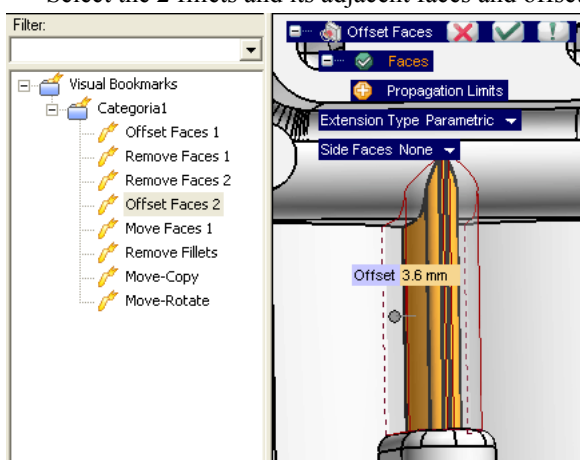
- Activate the Visual Bookmark Offset Faces 1. The aim is to reduce the diameter of the hole by 10mm.




- Activate the command **Offset Faces**.
- Select the circular faces as Faces to offset. Enter an value Offset10 mm.

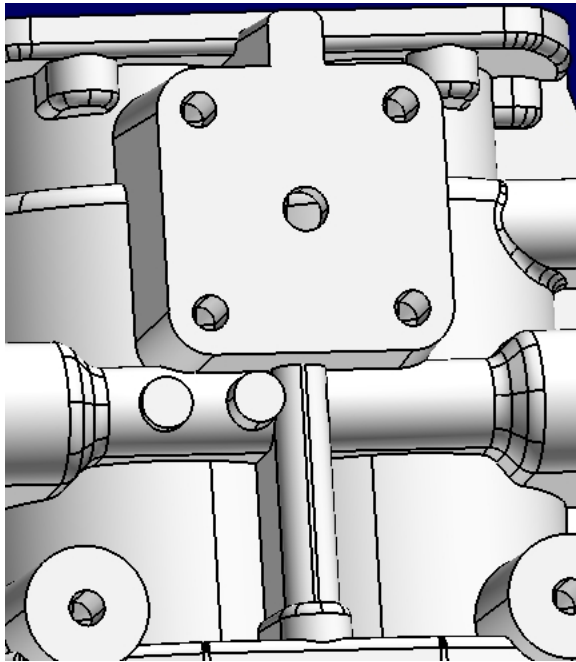


- Hit  Apply and without quitting from the command activate the visual bookmark Offset Faces 2.
- Select the 2 fillets and its adjacent faces and offset them by 3.6 mm as shown below.

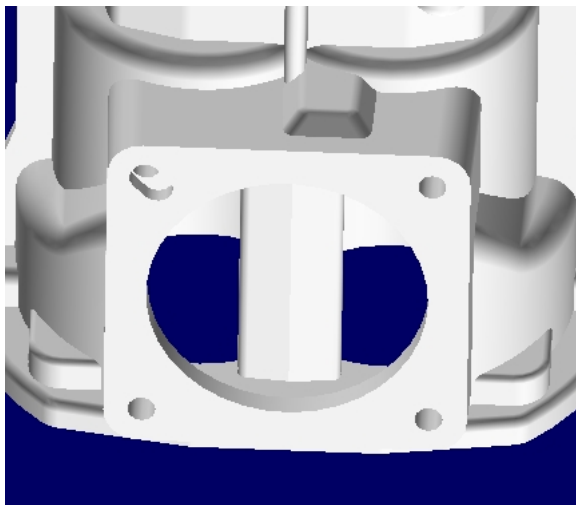



- Hit  OK and exit from command.

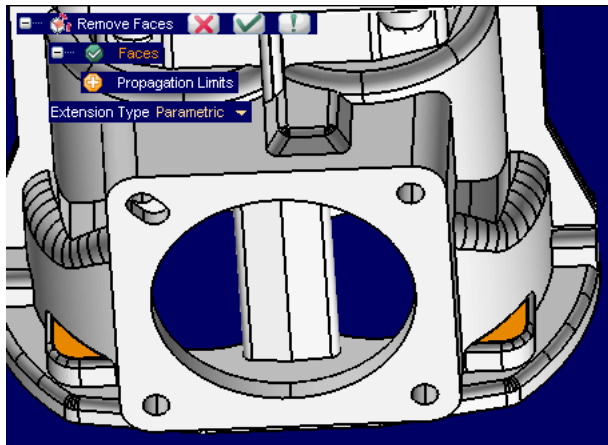
The result of the above operation is as shown below. Note that **Offset Faces** command extends the selected faces automatically where ever required to fit the feature correctly.




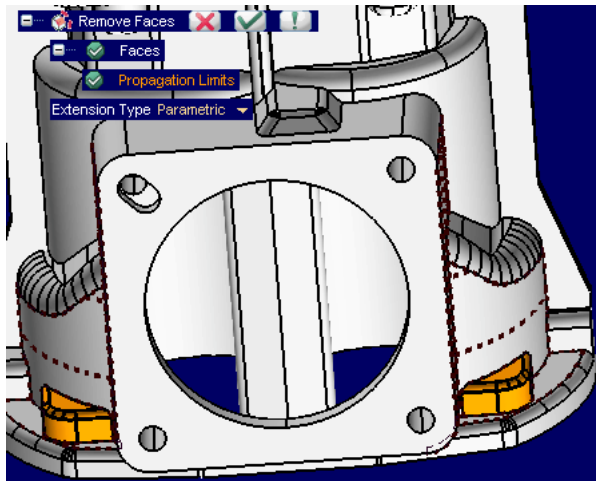
- From the Visual Bookmark tree activate Remove Faces 1.




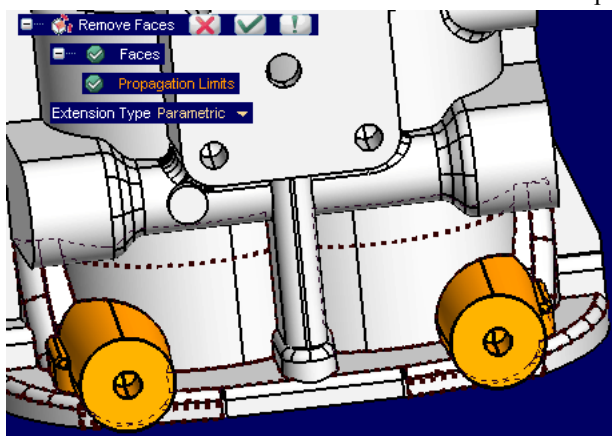
- Select the faces as shown below as  Faces to be removed.




- Select the adjacent faces as Propagation limits as shown below.
- Hit  OK to remove the selected features simultaneously.




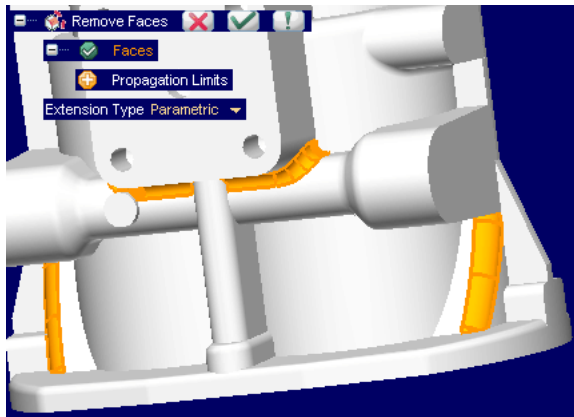
- Activate the Visual Bookmark Remove Faces 2 and select the highlighted faces as  Faces to be removed and the faces shown with dotted line borders as Propagation limits.



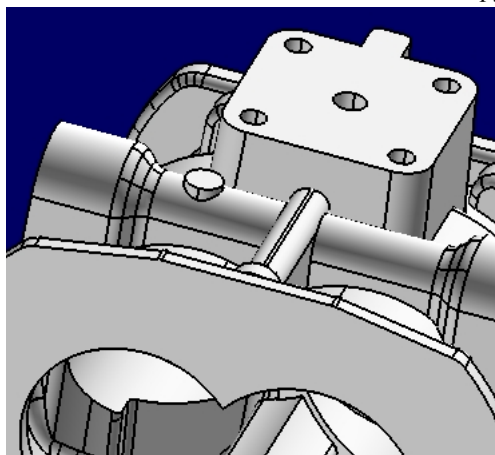
Hit  OK to remove the selected features simultaneously. Removing unwanted features on a Static Solid is made very easy with **Remove Faces** command. One last example of the same on Fillets.

- Activate the Visual Bookmark Remove Fillets. Activate the **Remove Faces** command.

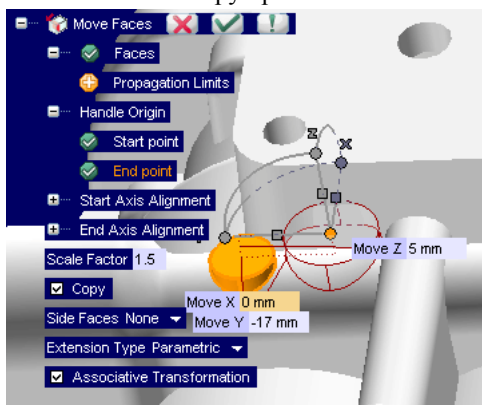
- Select both the Fillet  Faces as the faces to be removed. Make sure you select the complete chain of Fillets. If not you might come across Open Solid warning.



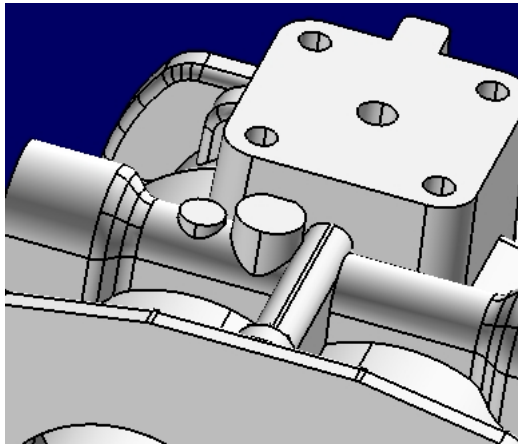
- Activate the Visual Bookmark Move-Copy and activate the command **Move Faces**.



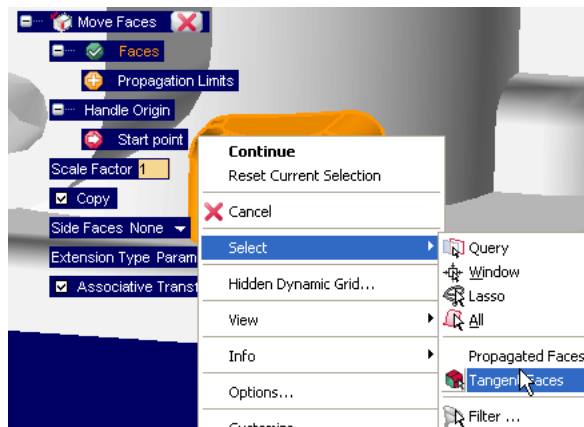
- Select the faces as shown and enter the transformation values as Move Y -15 mm and Move Z 5 mm.
- Check the ☒ Copy option and enter the scale factor as 1.5.




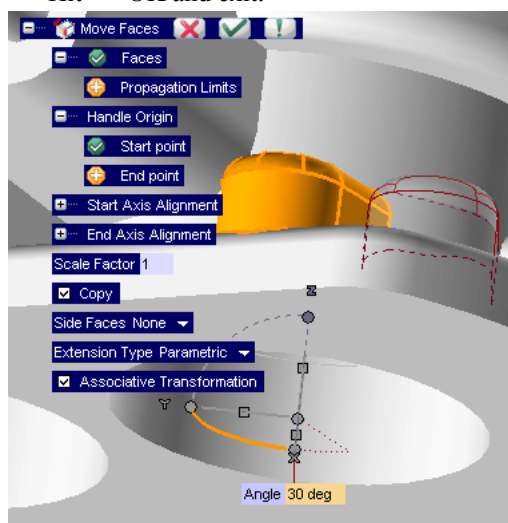
- Hit  Apply. The feature has been Scaled, Copied and Moved in one single step.



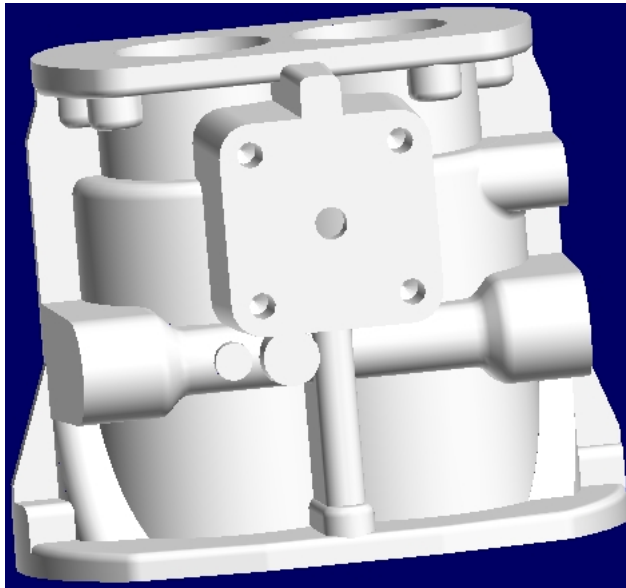
- Without exiting from the command activate the Visual Bookmark Move-Rotate.
- For the easy selection of the faces you can use Tangent Faces selection from the context menu.
- Select the faces as shown below.



- Set the Start point as the Centre point of the hole and rotate about Z axis by 30 deg.
- Hit  OK and exit.

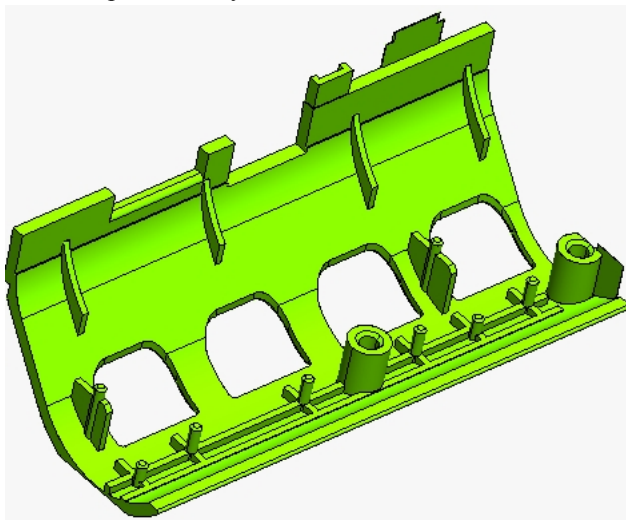


Save the Model with a different name and compare it the original one. The finished model should look as shown in the image below.

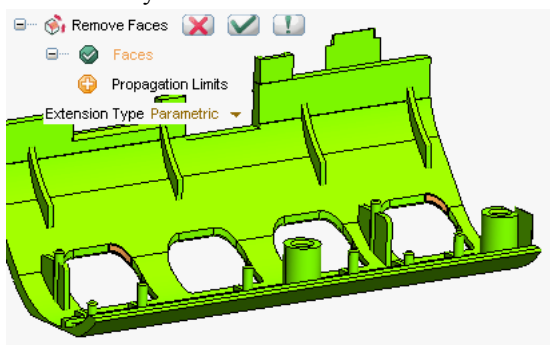


7. Step 7 : ISM on a Plastic model

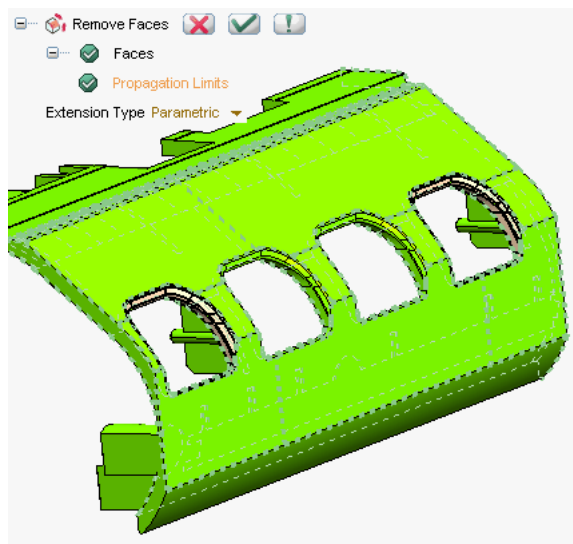
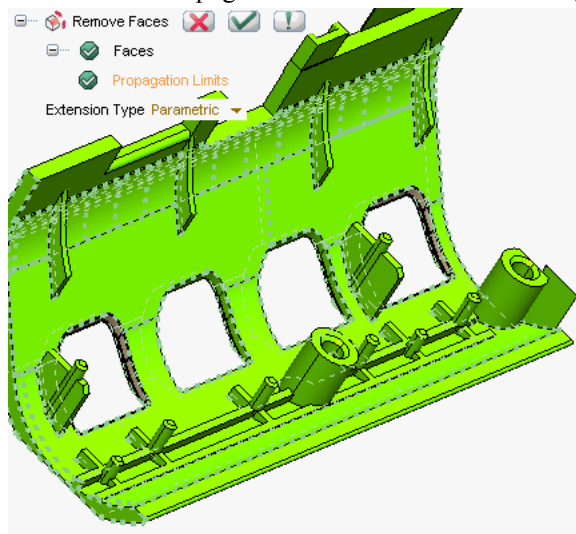
In this step we will try some of the ISM commands on an Plastic model.



- Load the model file Plastic_model.e3
- Activate the visual bookmark Remove_Slot. Activate the command **Remove Faces**.
- Select any one face on the 2 slots as shown.

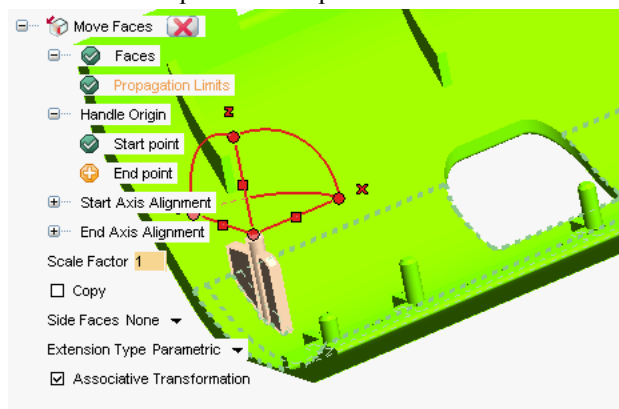


- Select the Propagation limits as shown in the image below.

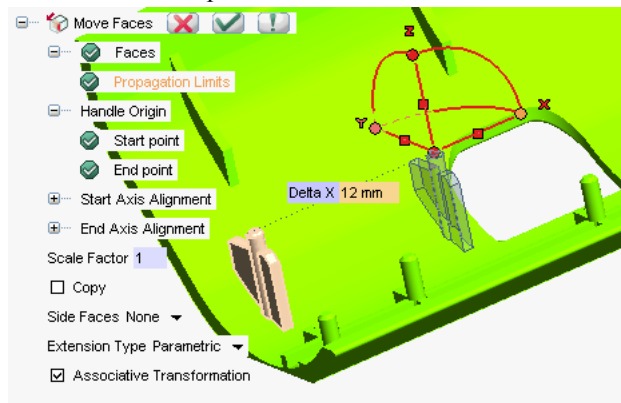


Hit OK to remove the selected Slots.

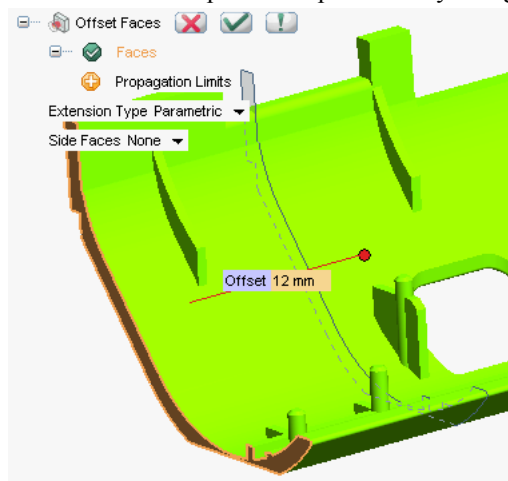
- Activate the Visual Bookmark Move_Part and activate the command **Move Faces**.
- Select the top face of the part as Faces and the bottom face as Propagation limits.



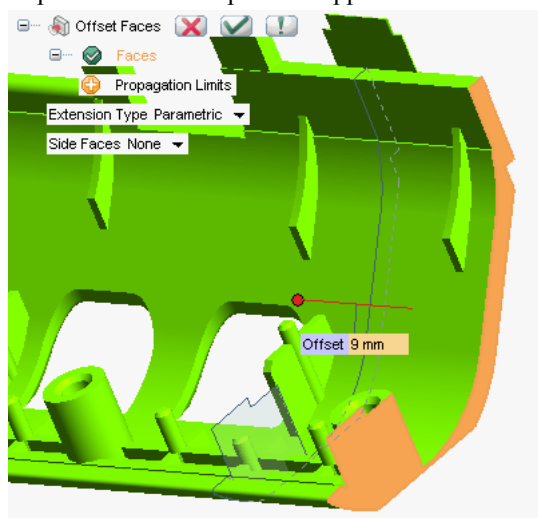
- Move the selected part in X direction by Delta X 12 mm.
- Hit OK to complete the transformation.



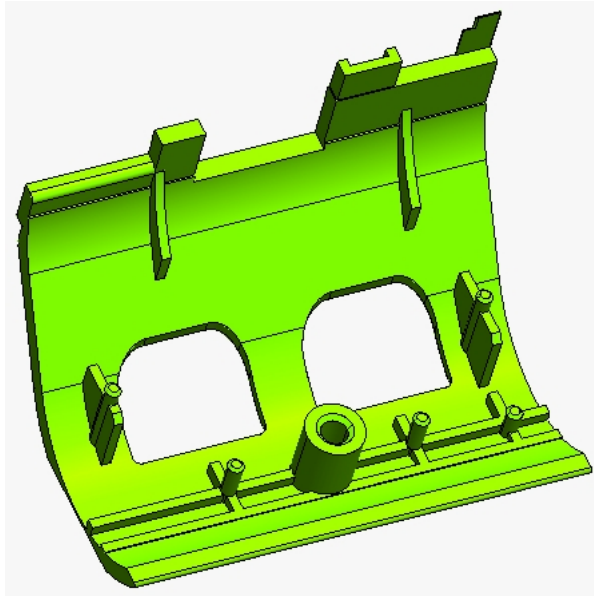
- Activate the command **Offset Faces** and select the outer most faces as face to Offset.
- Enter an Offset value of 12 mm
- Hit OK to complete the operation. By doing this step we are actually reducing the length of this model.



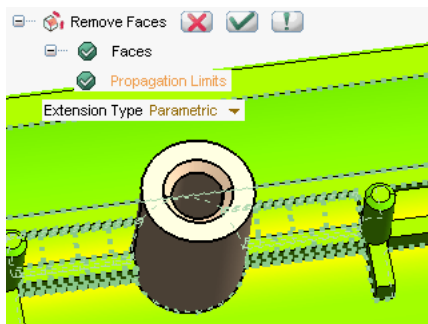
Repeat the above step on the opposite face with an Offset value of 9 mm.



After these 2 steps the model will look as shown in the image below.



- Activate the Visual Bookmark Remove_Feature and activate the command **Remove Faces**.
- Select the top face of highlighted feature and all other faces connected to it as Propagation limits.
- Hit Ok to remove the feature.



The model after applying all the ISM command looks as shown in the Image below. These set of ISM commands certainly make the design change much easier compared to traditional methods.

