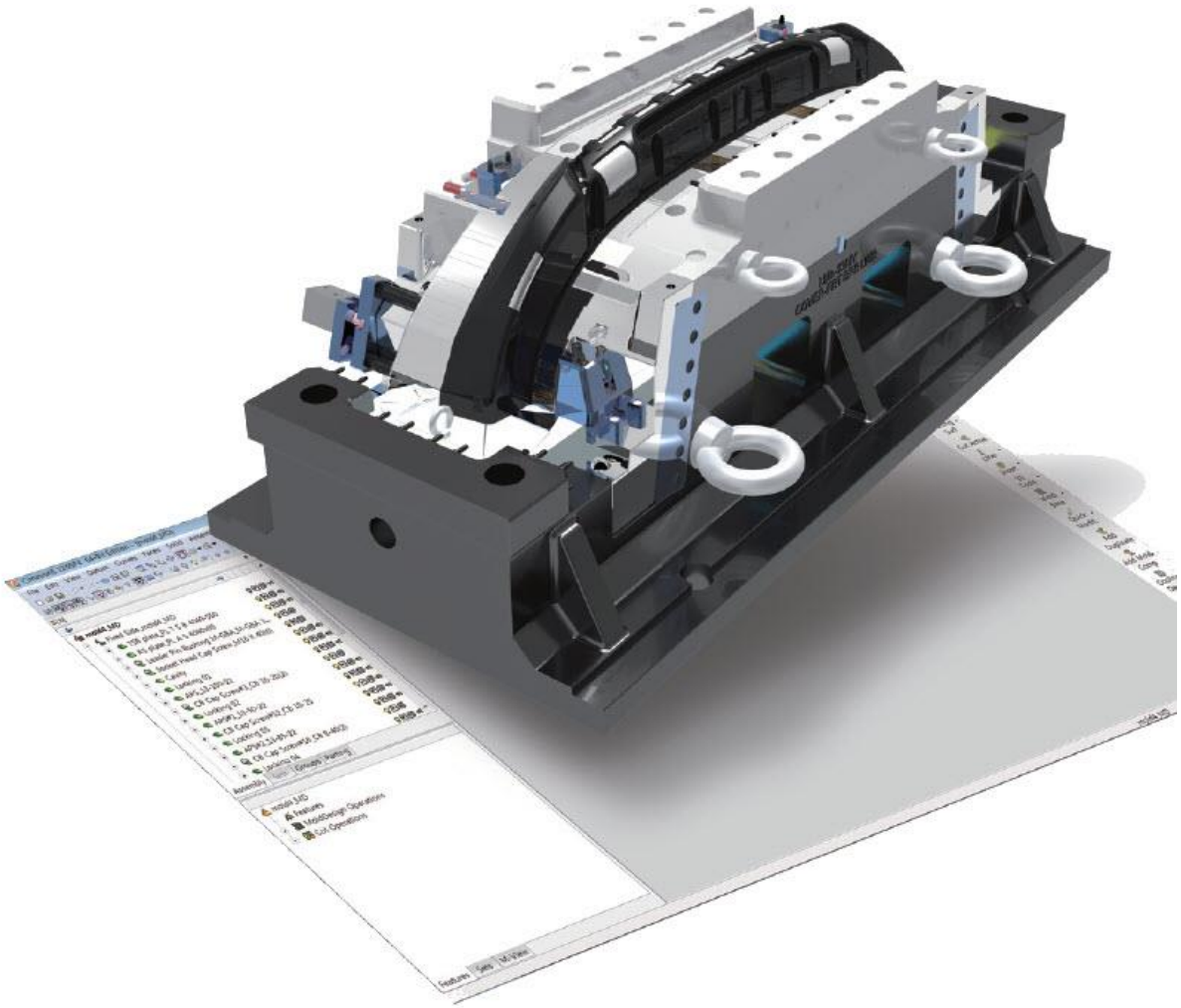
CIMATRON

Parting and Analysis Tutorial

Cimatron 16



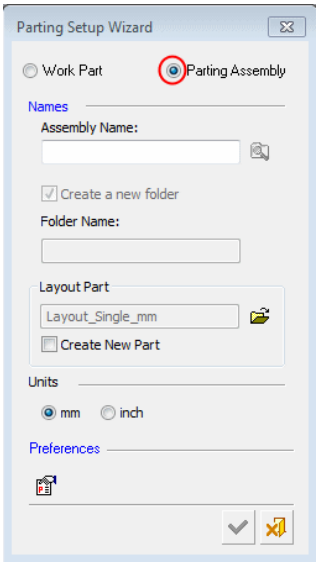
CAMBRIO

Contents

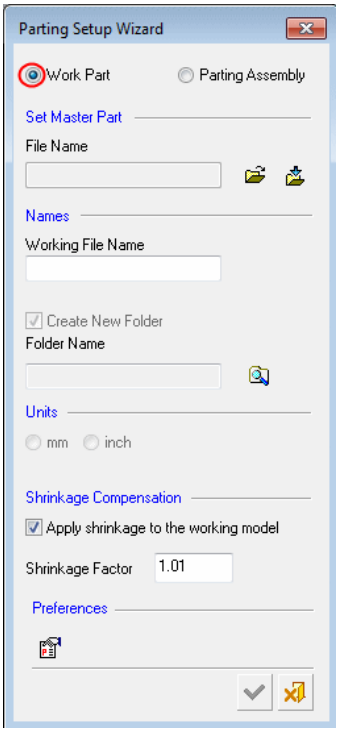
Exercises

Parting Assembly

1. Click the **Parting Setup Wizard**. Notice the tab, **Parting Assembly**.



2. The previous Parting dialog can be accessed by checking the **Work Part** option.



3. Check the **Parting Assembly** option.

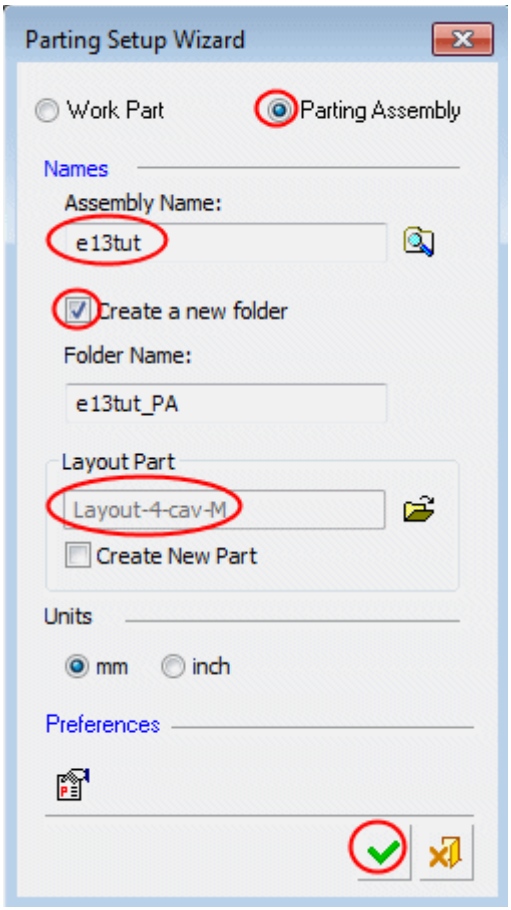
   The Parting Assembly dialog enables create a parting assembly that can be later loaded to a new or existing mold project (mold assembly). In this way, the two tasks (mold and parting) can be done separately and merged together later.

   The parting assembly is automatically loaded with the layout part. This assembly environment offers options in the guide such as **adding work parts** and **analysis tools**.

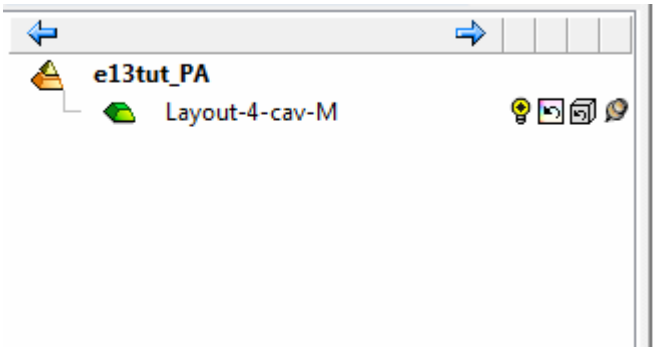
   Copy the **Layout-4-cav-M.elt** file from the Tutorial folder to the following folder:

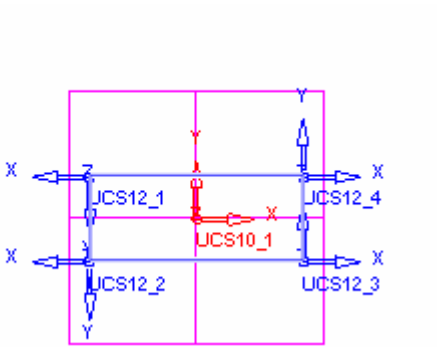
   \<ProgramData>\Cimatron\Cimatron\16.0\Data\dat\ApplicationsData\Layout-parts\MM

4. Name the assembly. Set as **Layout Part** as the **Layout-4-cav-M** supplied with this exercise. Press **OK**.



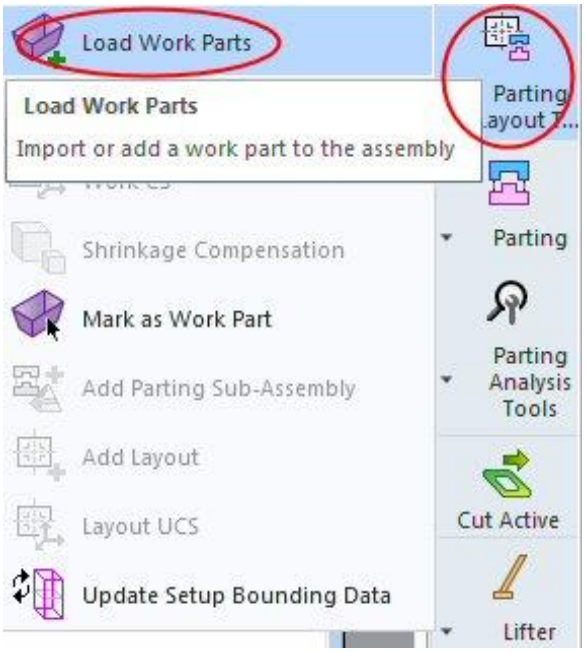
5. The assembly loads with the layout part.





Note: Notice the new parting guide, which contains more functionality such as more analyze tools, Parting Surface part functions, etc.

6. From the **Parting Layout Tools** menu, select the **Load Work Parts** option.

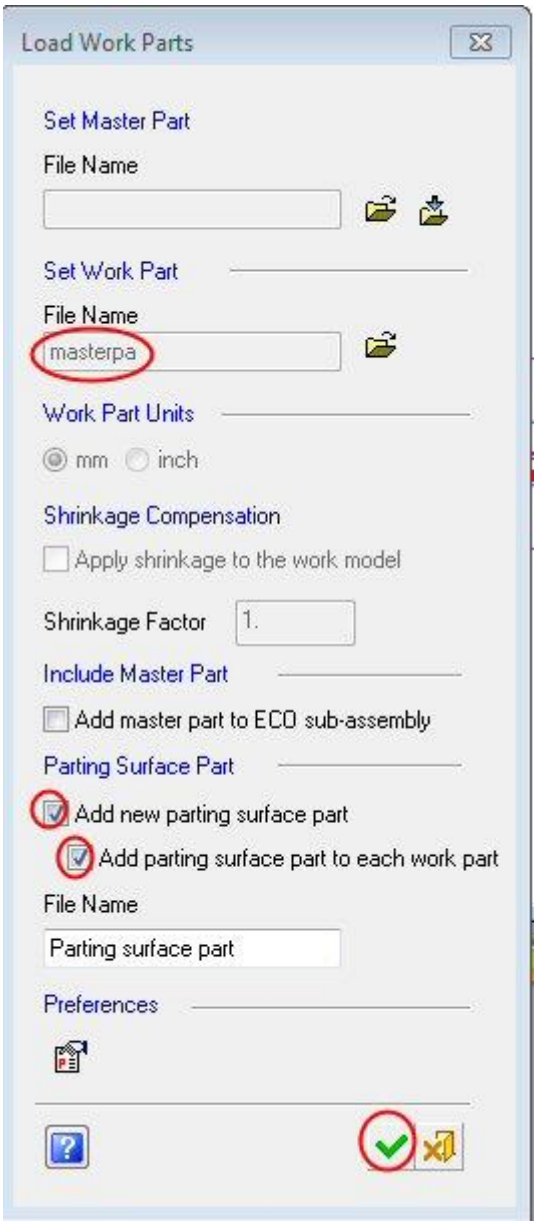


7. Load the work part, **masterpa.elt**. Select the following check boxes:

   Add New Parting Surface Part
   Add Parting Surface Part to Each Work Part

   When the checkbox is selected, the **File Name** field displays the default part name, **Parting Surface Part**.

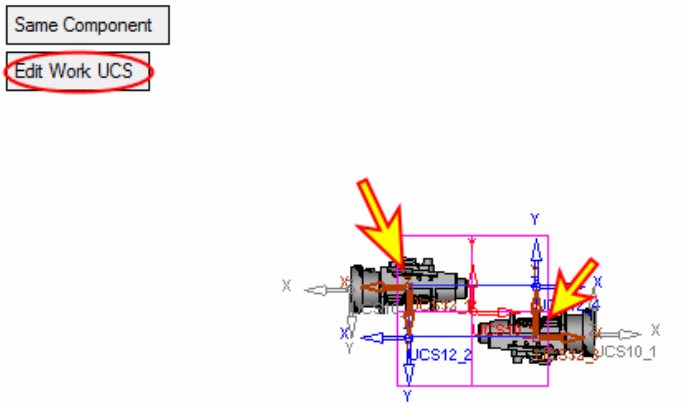
   A parting surface part will be added with each work part instance added on all UCSs selected for placing the work parts (copies of the layout UCS).



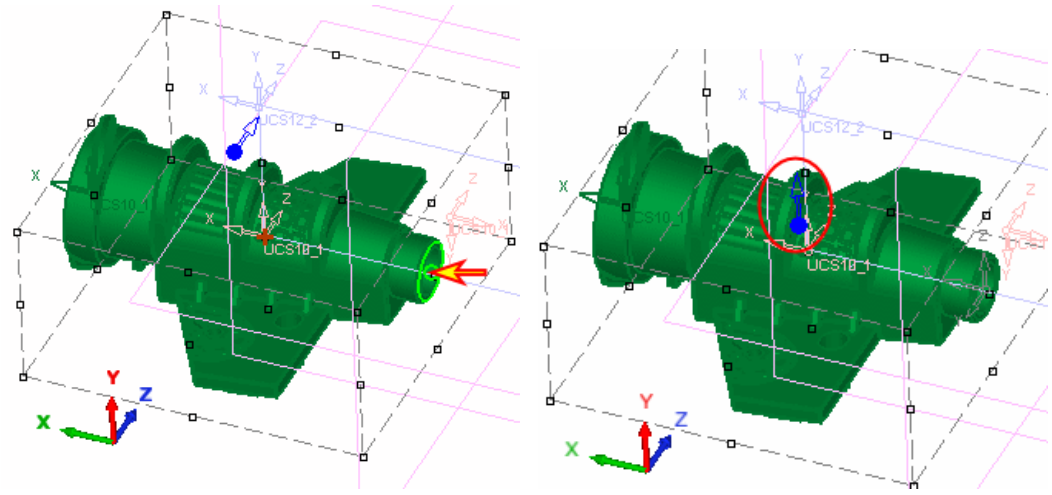
8. Press **OK**.

   The parting surface part is added as **Same Component** or as **Different Component** per option specified in the second step of the Load Work Part function.

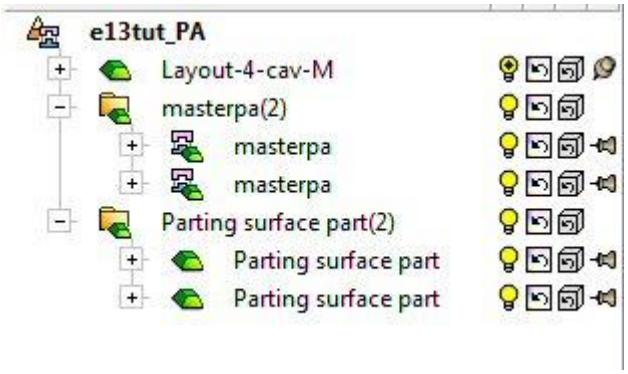
9. Place the work part twice (Same Component) as indicated by the arrows below (below Edit Work UCS):



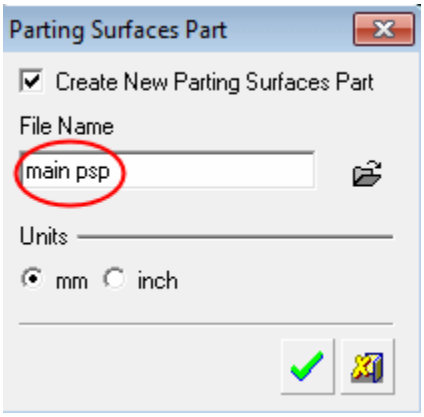
10. Select the center of the cylindrical face as the UCS origin point. Select **Along Y.**

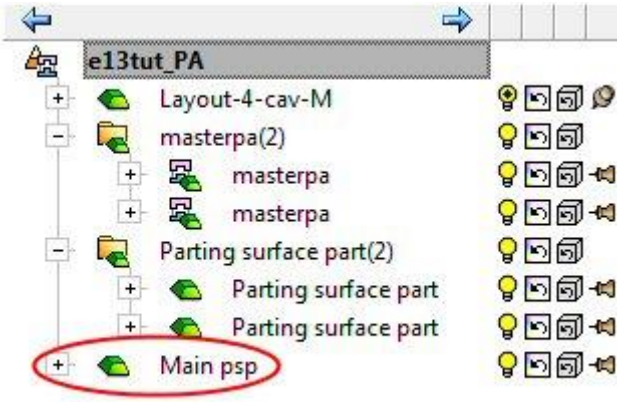


The work parts and their parting face parts displayed in the tree.



11. Add another parting surface and name it, **Main**. Place it on the assembly UCS.

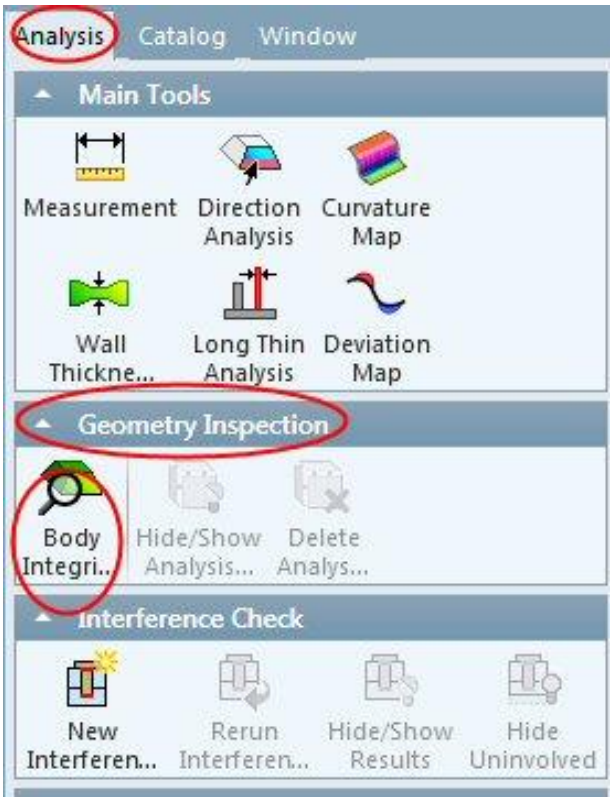




Analysis Tools – Body Integrity Analysis

Analyze various geometry quality issues of a selected object**.**

1. From the Main menu bar, select **Analysis > Geometry Inspection > Body Integrity Analysis**.



2. The **Body Integrity Analysis** tool is mainly used to detect mending operations required for the work part. This Analysis group contains the following options:

   Open Edges: This option is equivalent to the old **Analyze by Object** tool. It displays all open edges in the selected closed or opened solid object.

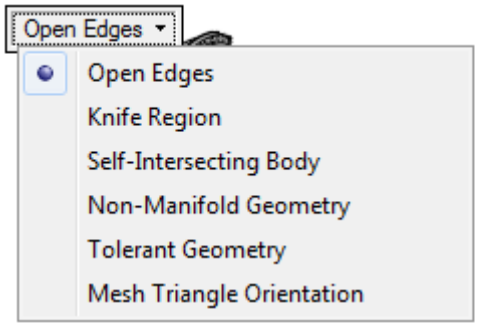
   Knife Region: Display all areas with very sharp edges (up to a defined angle) in the selected object. Then set the maximum angle that defines the sharp edges. (Minimum value: 0 degrees. Maximum value: 45 degrees.)

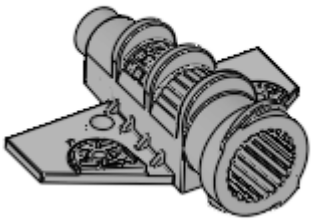
   Self – Intersected Body: Display all intersections between faces in the selected object.

Non – Manifold Geometry: This option is the same as the one in the **Entity Information tools**. Display non-manifold (zero thickness or non-manufactured) edges/vertices.

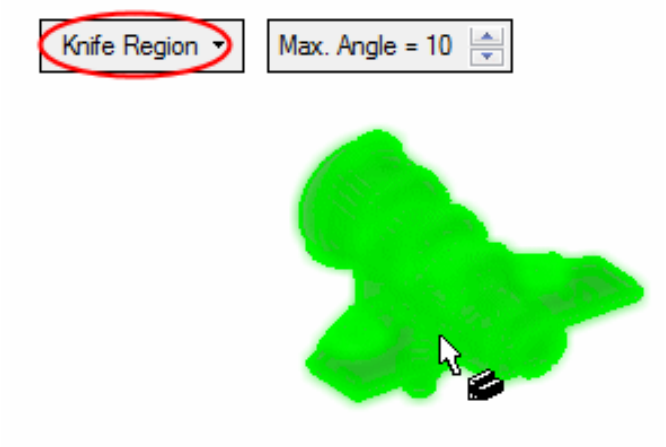
Tolerant Geometry: This option is the same as in the **Entity information tools.** Display tolerant edges/vertices that are larger than the defined tolerance value.

Mesh Triangles Orientation: Check the orientation of mesh triangles. Only mesh objects can be selected. When the checks are completed, a status message is displayed. If bad orientation was found, a possible fix is suggested using the **Check & Fix Tools for Mesh**. This option is only displayed if the file contains mesh objects.

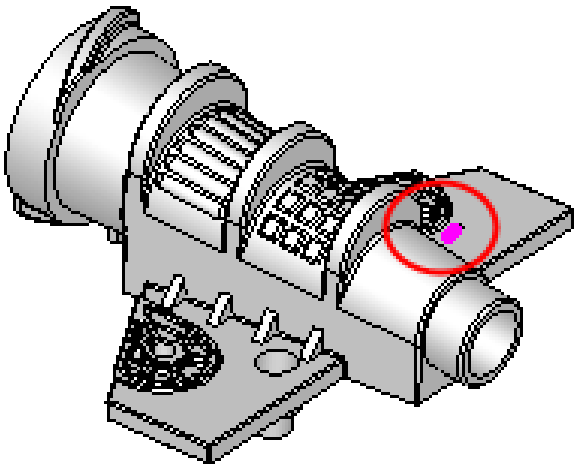




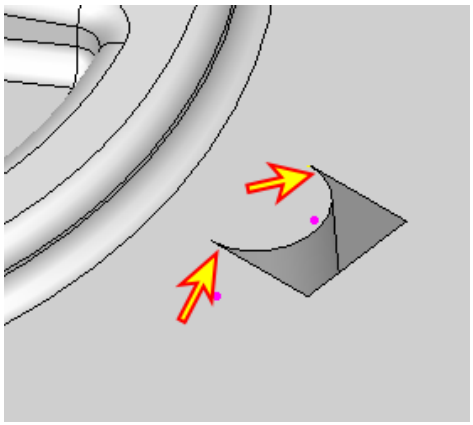
3. Select the **Knife Region** option. **Max Angle** = **10**. Select the body. Select **OK**.



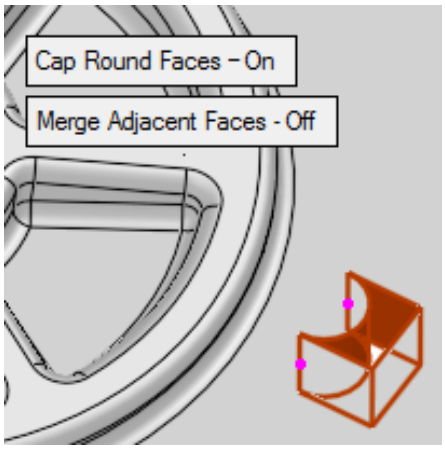
4. All faces with angle above 10 degrees (as set in the **Max Angle** value) are marked with **PINK** dots. Zoom-in to the area marked in **RED** circle.



5. Notice the two areas:

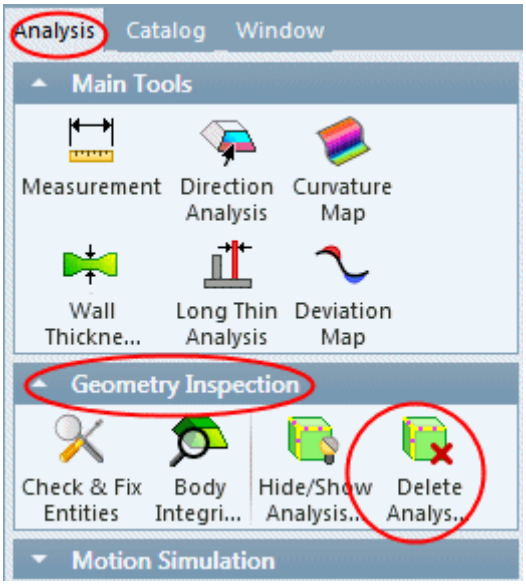


6. Activate the mater part. From the **Solid** menu, select the **Remove and Extend** option to remove this pocket.

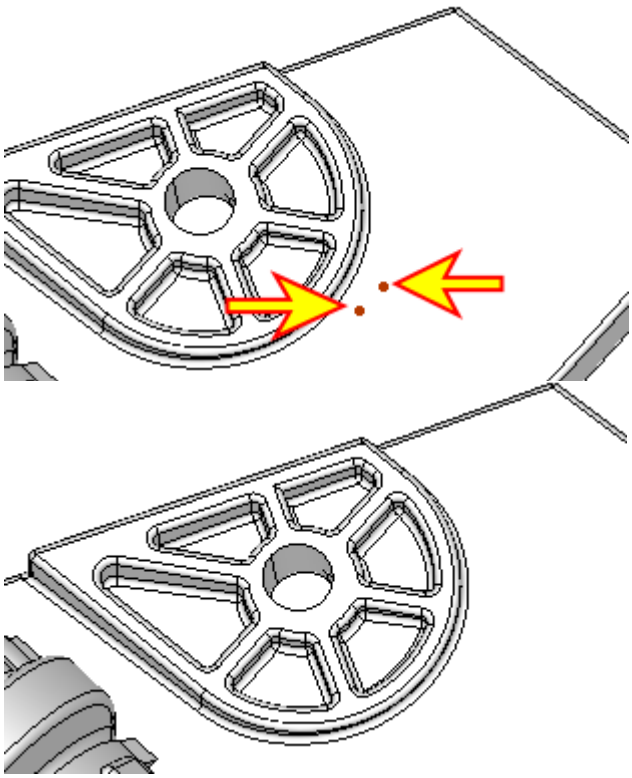


In addition to the **Hide/Show Analysis Marks** option, you can delete these marks. It is recommended to delete the marks of areas that were fixed to get a clearer view of what else needs to be done (without the need to rerun the analysis).

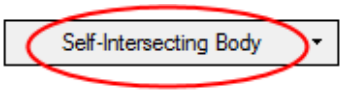
7. Select the **Delete Analysis Marks** option.

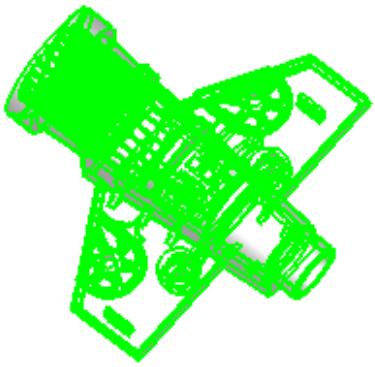


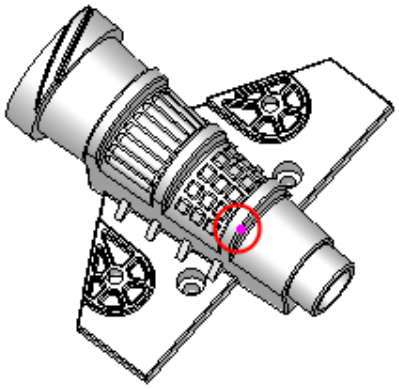
8. Delete the points indicated by the arrows:



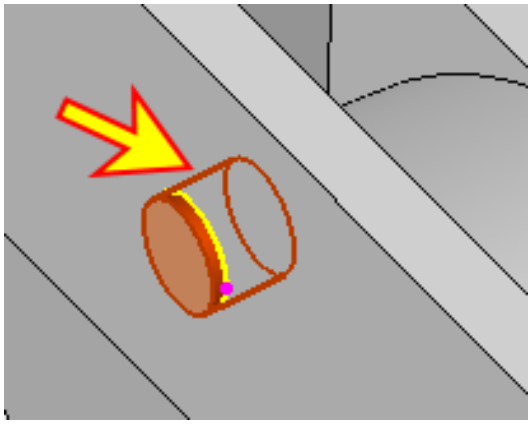
9. Set the analyze option to check **Self Intersecting Body**.



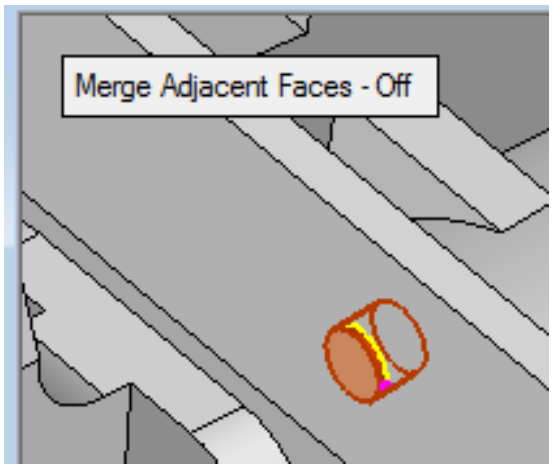




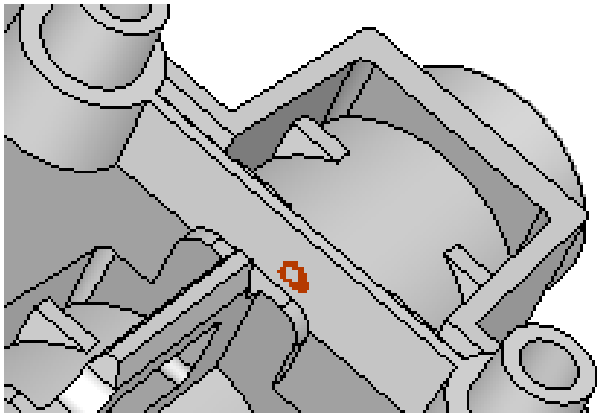
Notice the problematic face:



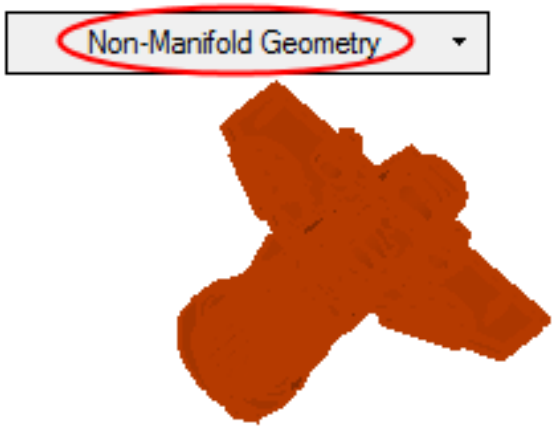
10. Use the **Remove and Extend** option to eliminate these faces.



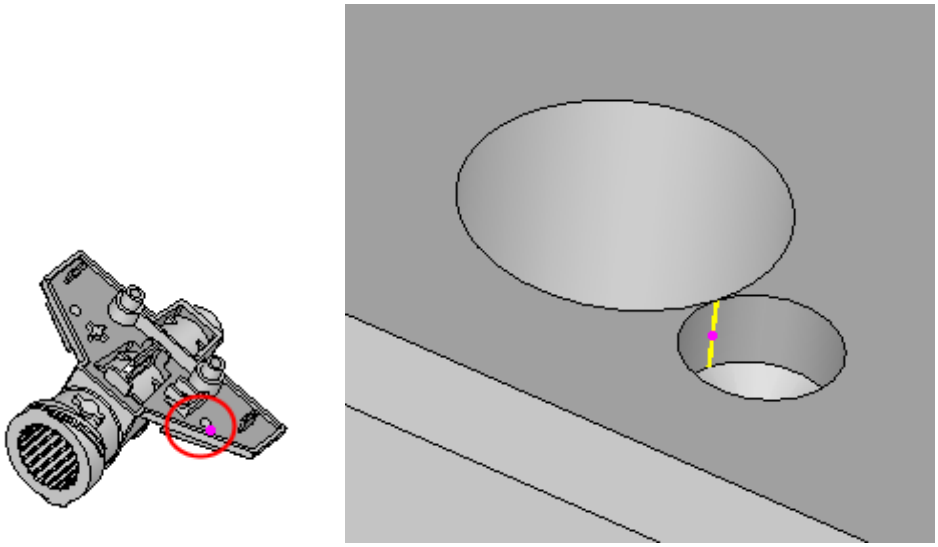
11. Delete the analysis mark.



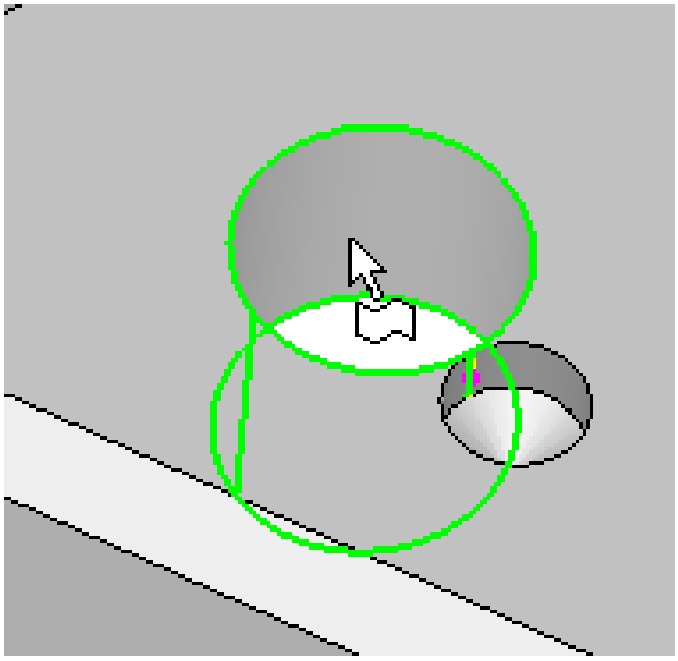
12. Set the analyze option to check **Non-Manifold Geometry**.



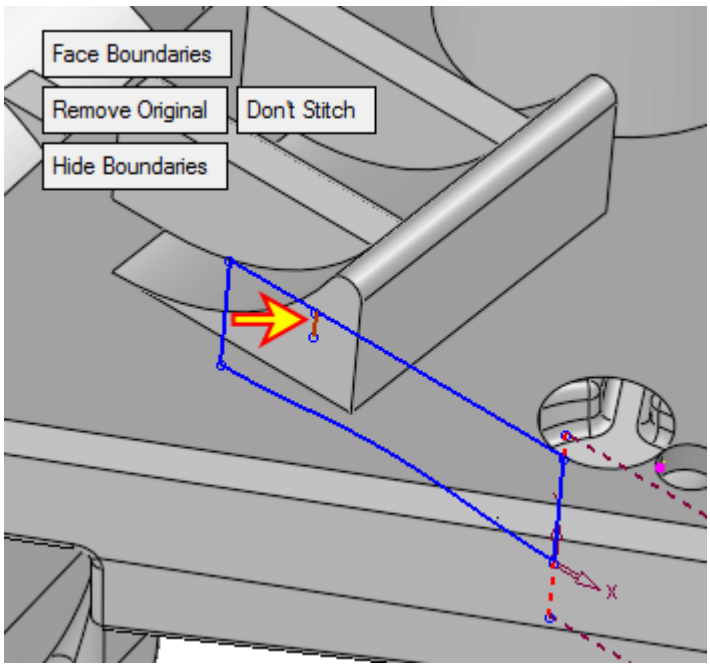
13. Zoom-in to the area marked in the **RED** circle shown below:



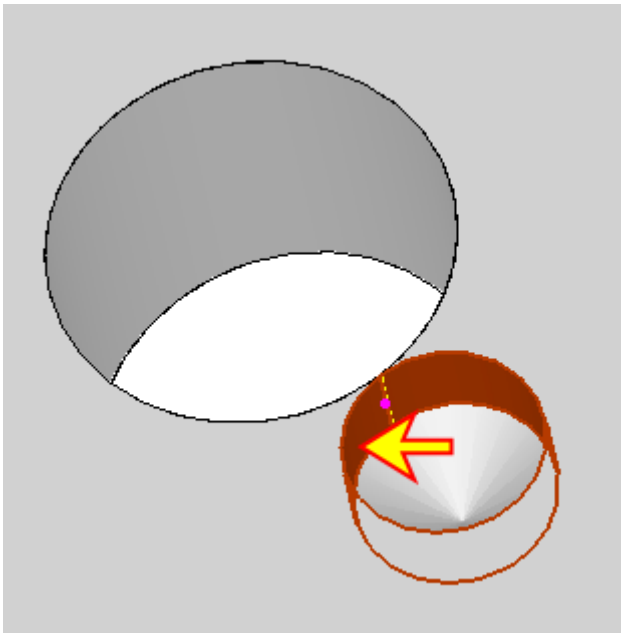
14. Modify **By Sketcher** the highlighted face as shown below:



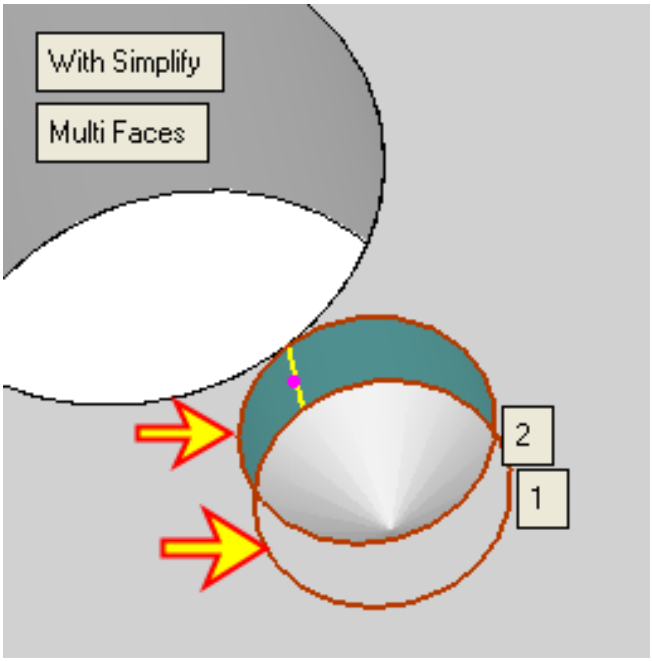
15. Delete the extra edges. **Exit** the Sketcher.



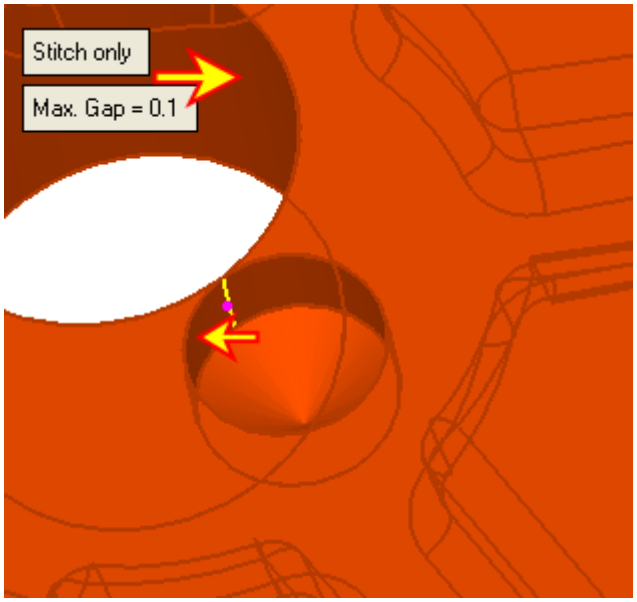
16. Delete the face indicated by the arrow:



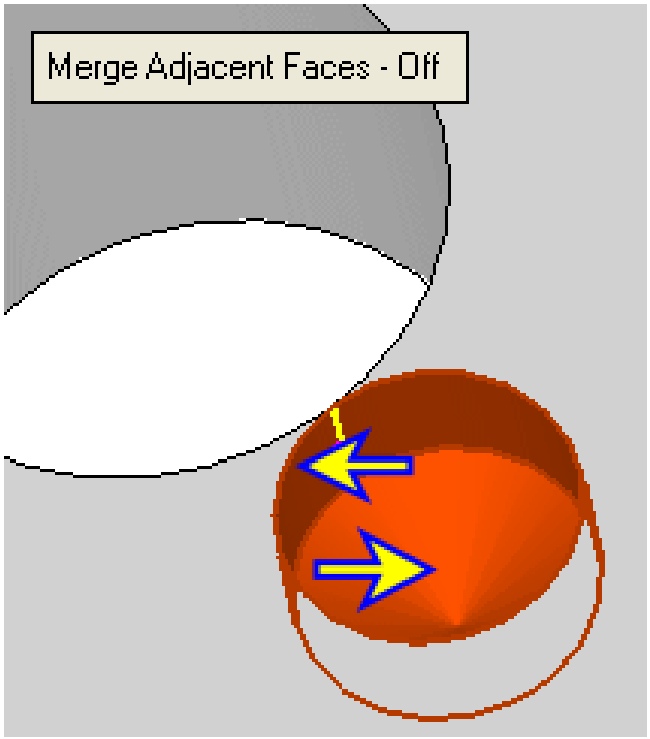
17. Create a blend face using the input edges indicated by the arrows below:



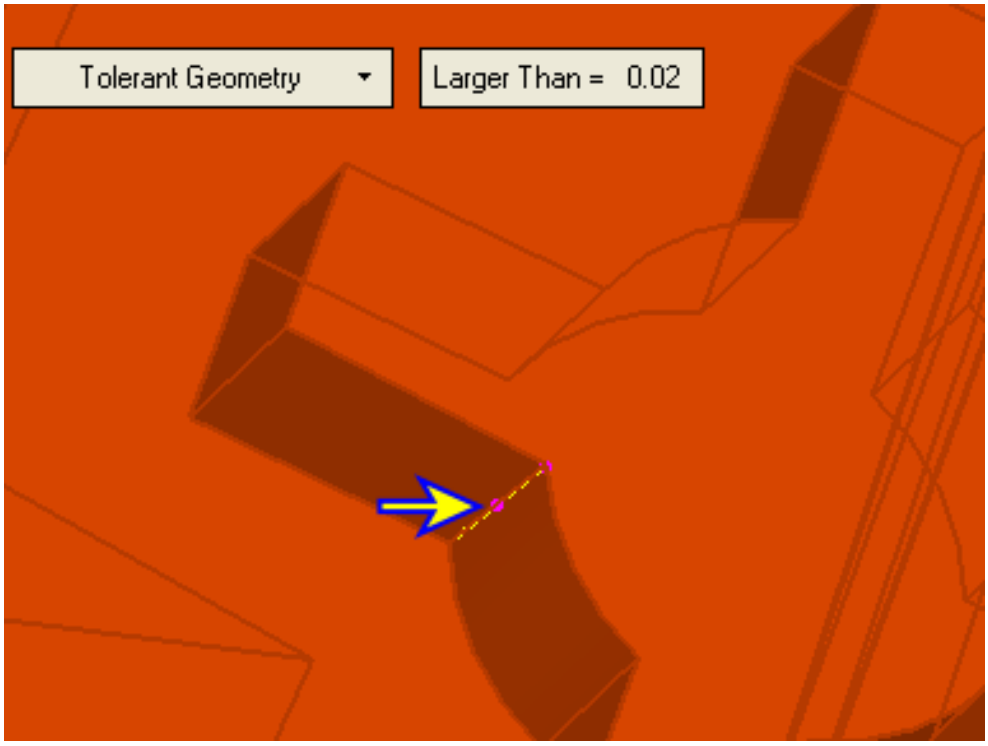
18. Stitch the two new faces with the main object.

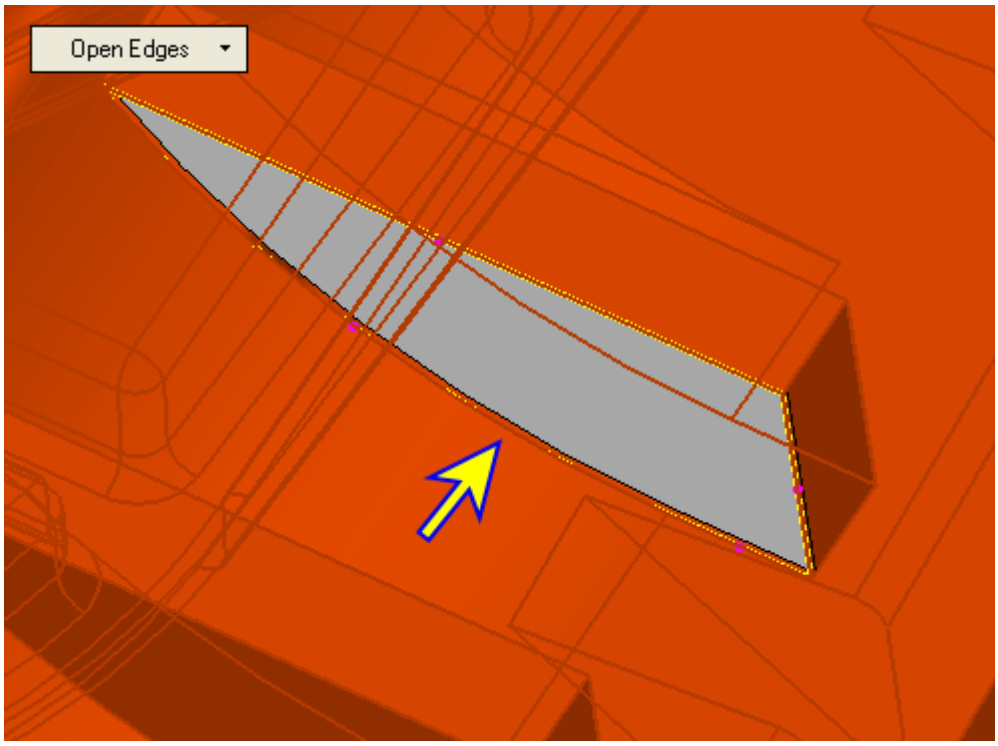


19. Perform **Remove and Extend** to eliminate the pocket.

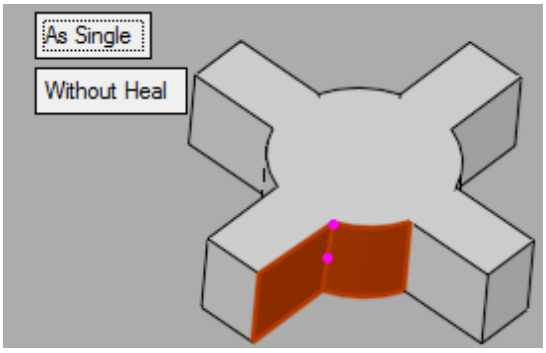
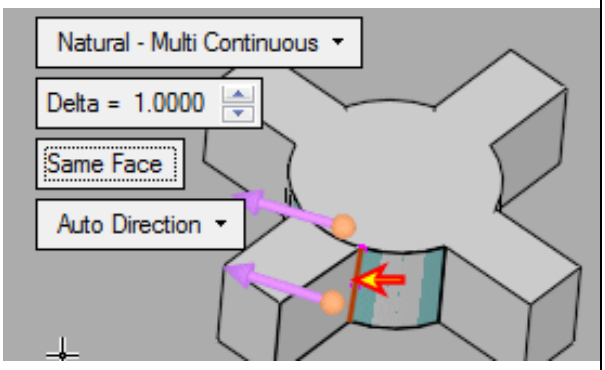
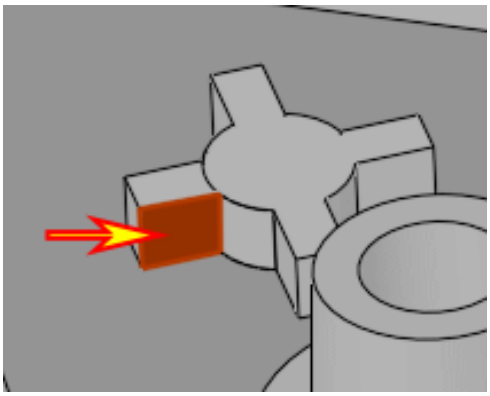
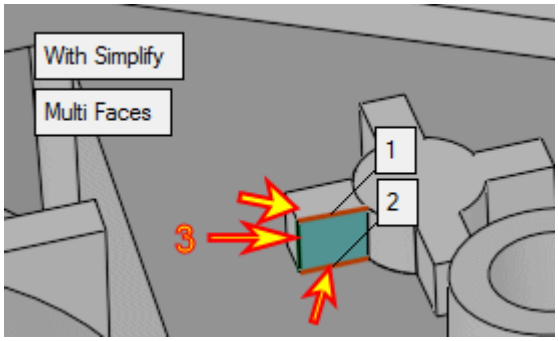
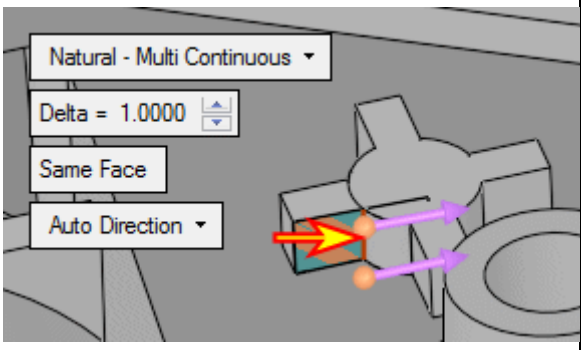
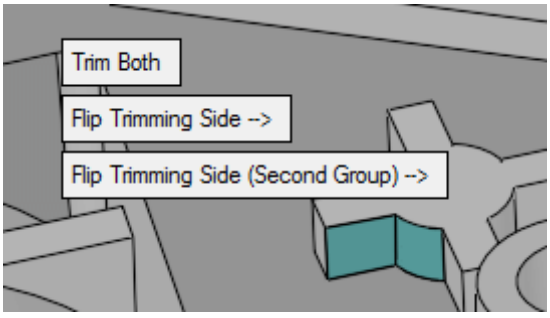
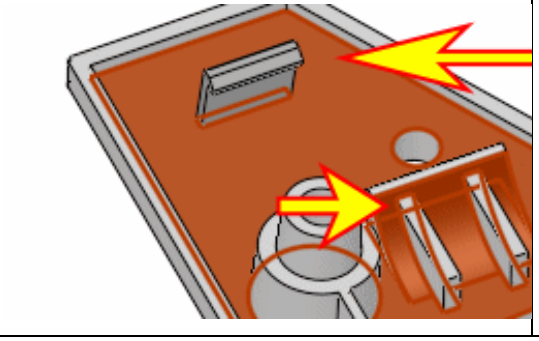
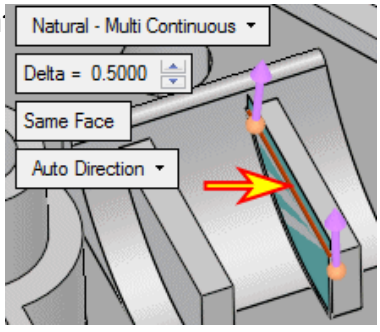


20. Also perform the **Tolerant Geometry** and **Open Edges** analysis:

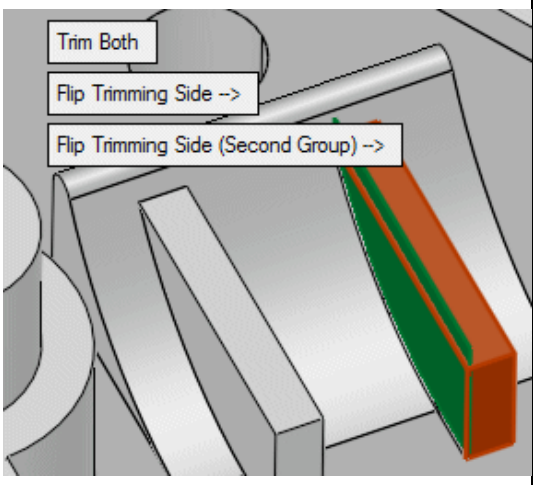




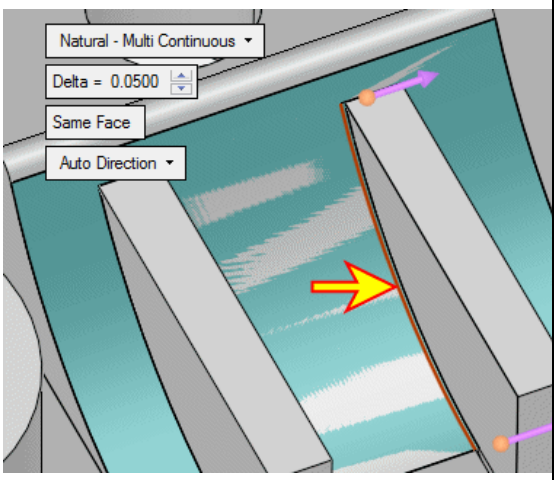
21. Fix the geometry. You may follow the steps described in the table below:

1. Unstitch the two faces. 	**2. Extend the cylindrical face.** 
3. Remove the face. 	**4. Create a blend face, use the edge number 3 as a boundary.** 
5. Extend the blend face. 	**6. Trim both faces with each other. Stitch all faces.** 
7. Unstitch the two faces. 	**8. Extend the face, use the indica** 

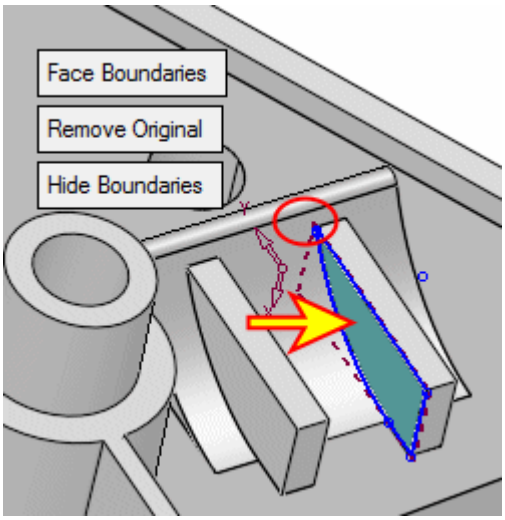
9. Trim faces.



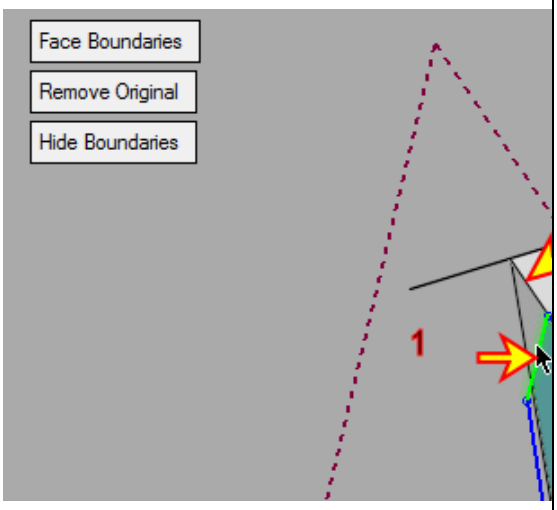
10. Extend face



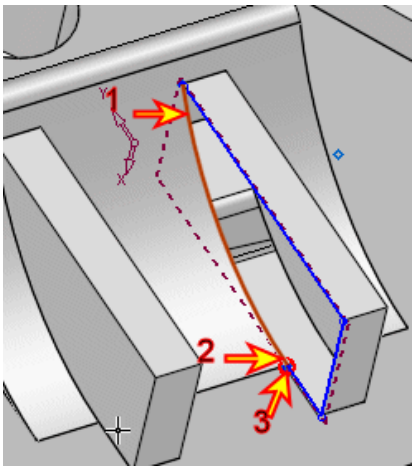
11. Modify by Sketcher. Zoom-in to the area marked by red circle.



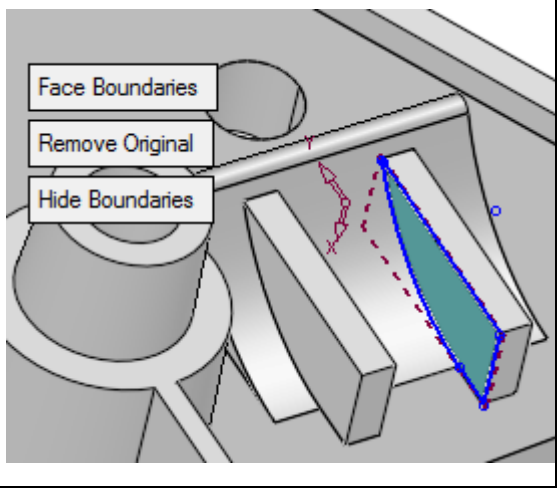
12. Delete edge 1 and add edge 2.



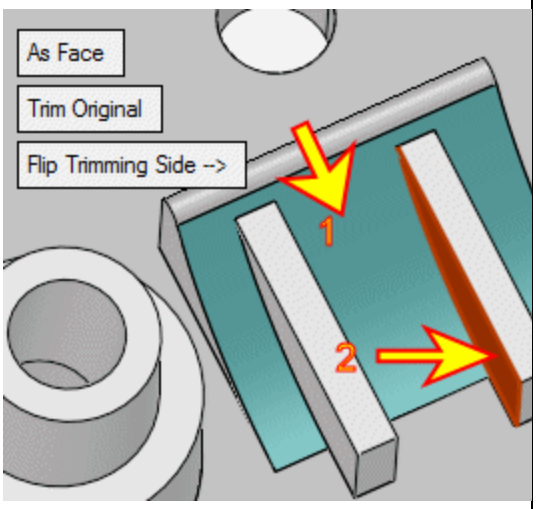
13. Add the edge 1 and close the gap between points 2 & 3 using the Close points tool.



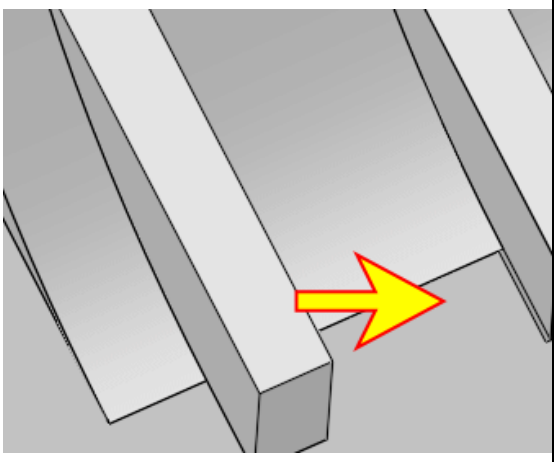
14. Select OK.



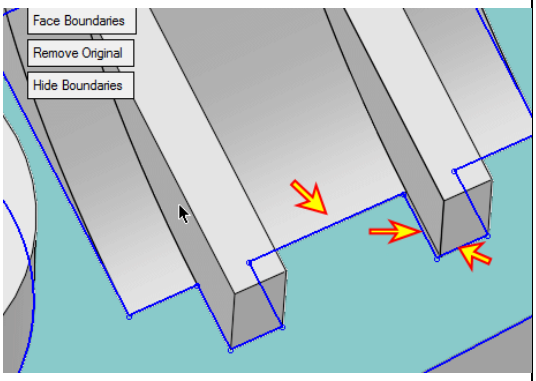
15. **Trim face 1 with face 2.**



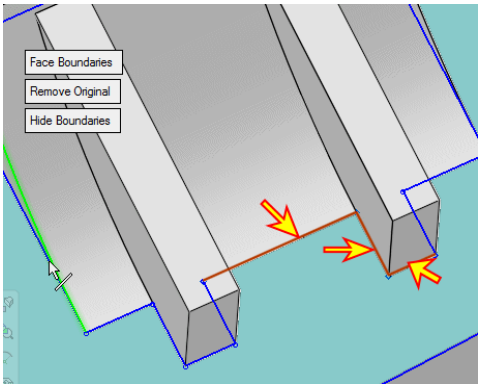
16. **Modify by Sketcher the indicated face.**



17. **Delete the three indicated edges.**

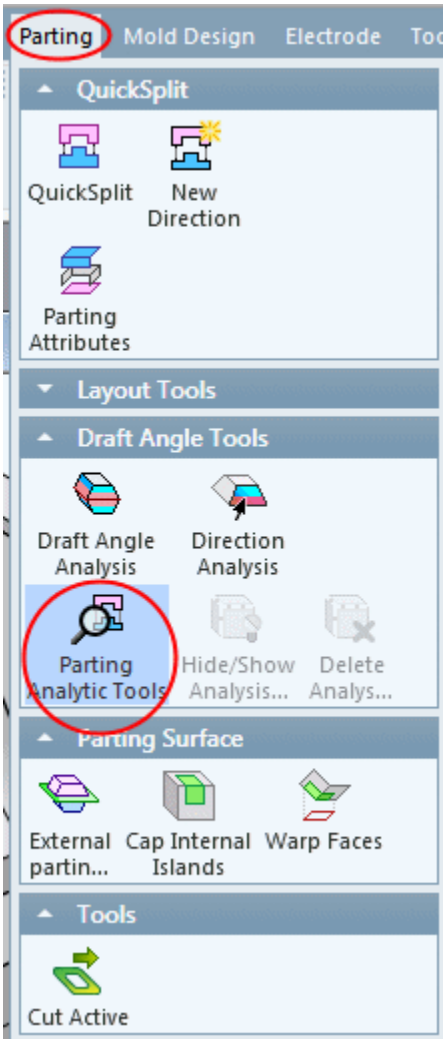


18. **Add the three indicated edges.**



Analysis Tools – Parting Analytic Tools – Analyze Free Edges

1. From the **Parting** menu, select the **Draft Angle Tools > Parting Analytic Tools** option.

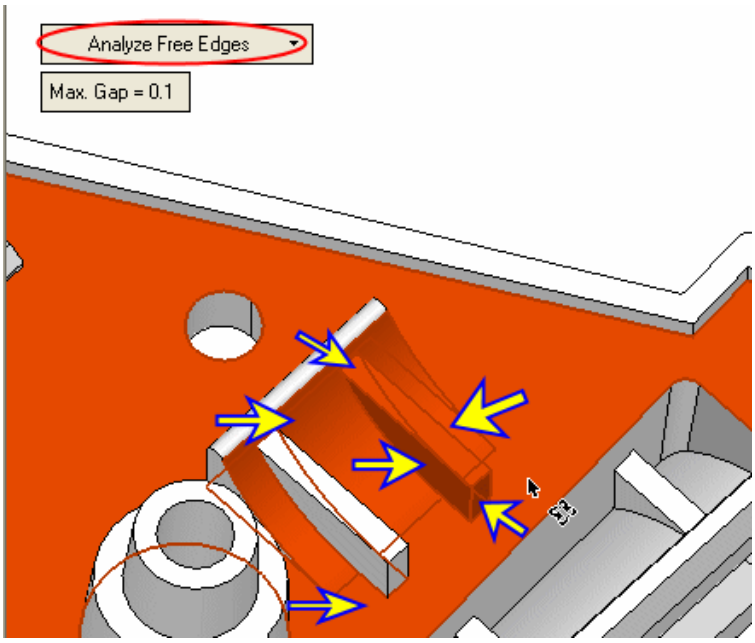


The **Parting Analysis Tools** function includes several analysis options for the parting process:

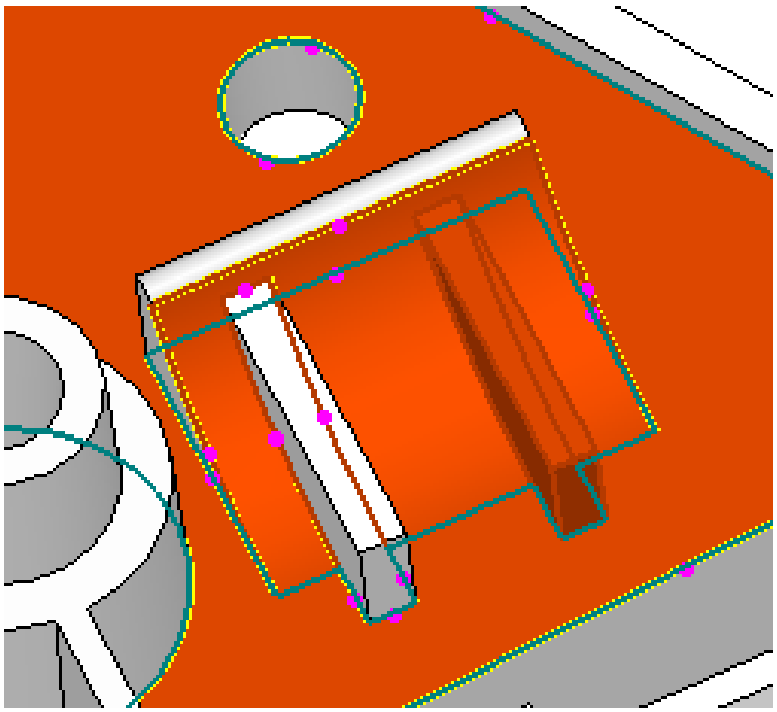
- Evaluation and Mending of Imported Parts
- Analysis of Parting Surfaces and Active Faces

2. Select the option **Analyze Free Edges** (the old **Analyze By Selection – Free Selection** option) to analyze a number of adjacent faces.

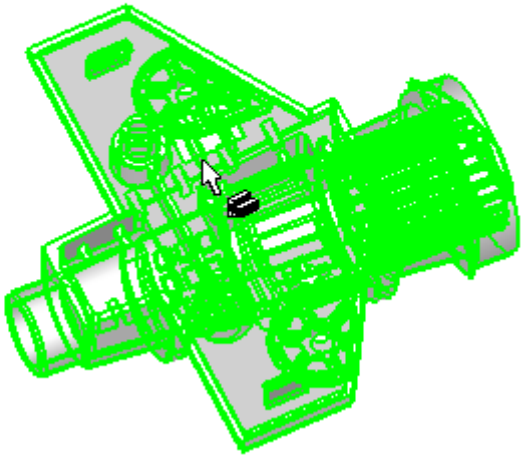
   Generally, it is recommended to use this option for analyzing local areas (of active faces and/or parting faces).



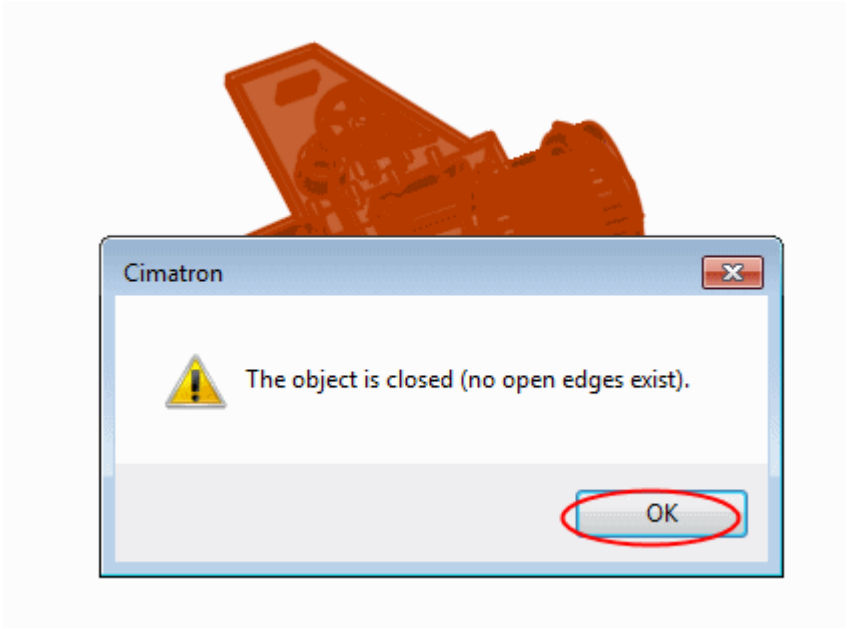
3. Notice that the free edges detected in the previous steps (file) no longer exist:



4. Stitch the entire faces. Examine if there are any open edges in the object.



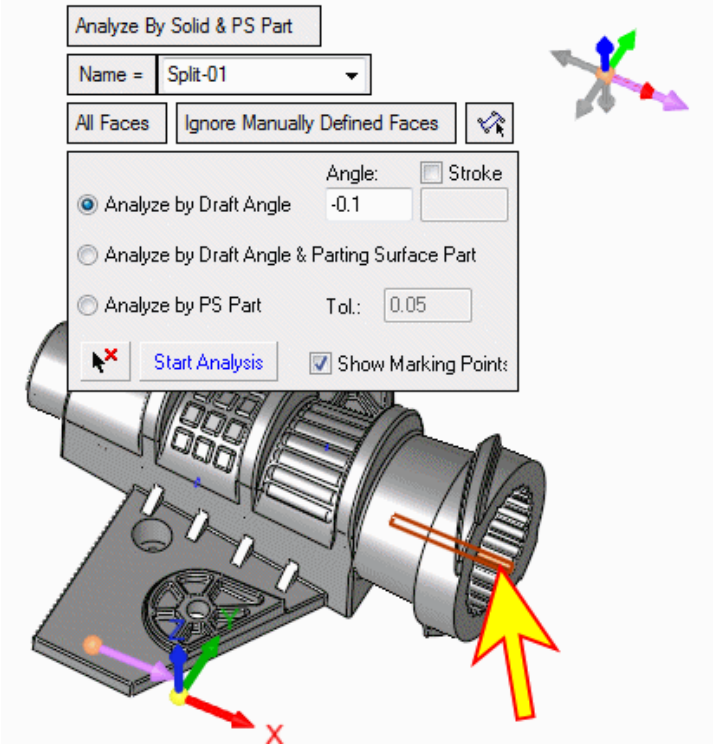
5. The object is closed. Press the OK button.



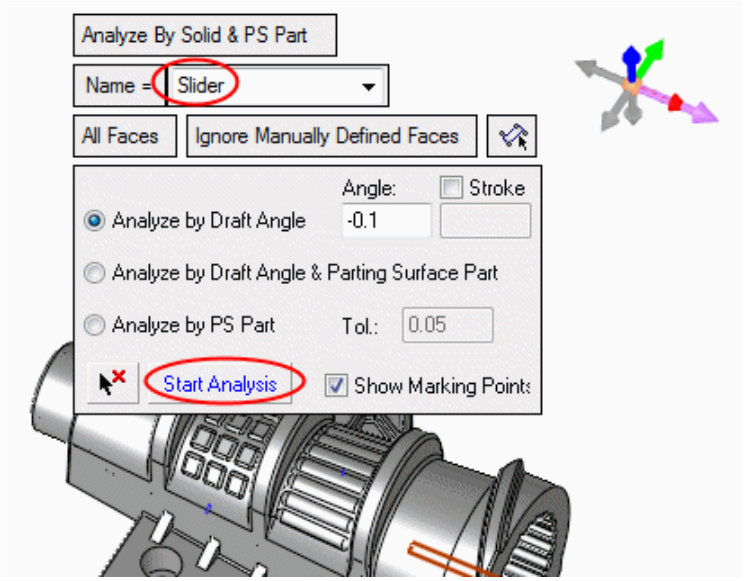
Quick Split Improvements – Predefine Name for New Direction

1. Create a new direction via the **Parting** menu in the Guide.

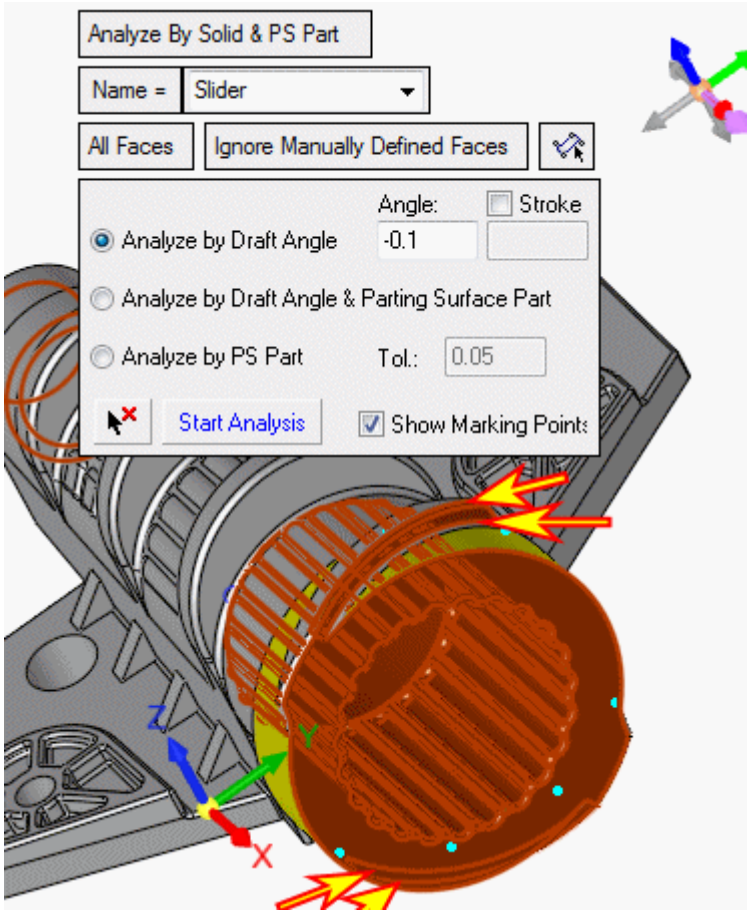
2. Select the **Along X axis** direction. Select the face as indicated below:



3. Set the name of the direction by typing **Slider**. Press the **Start Analysis** button.

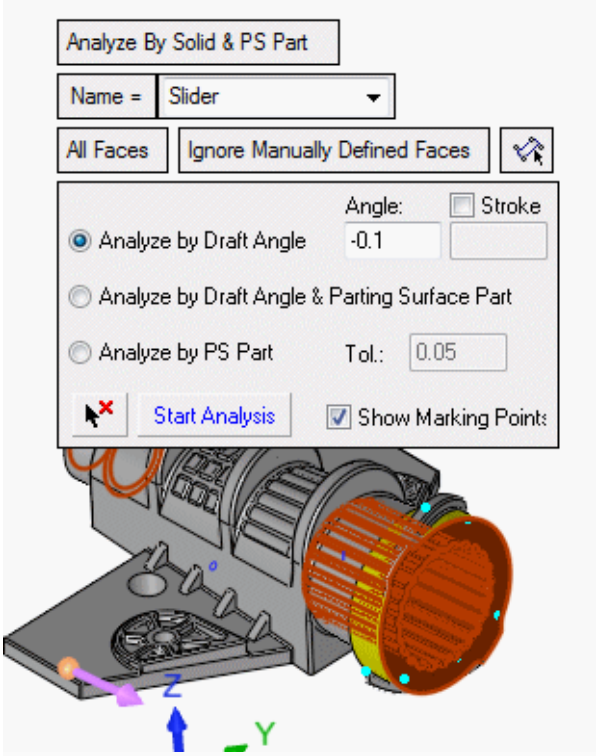


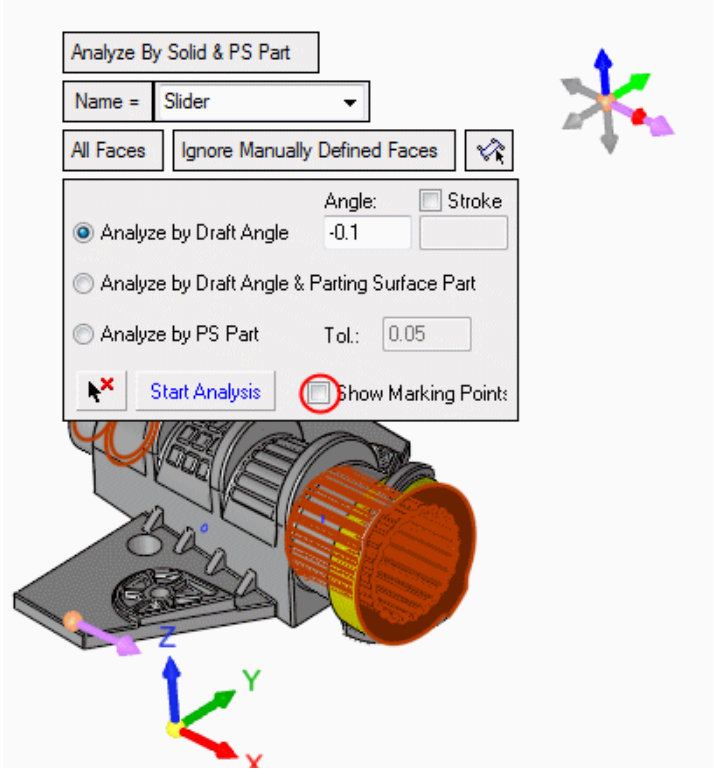
4. Unselect the four faces shown below so that only the inner faces and the front face will remain in the set:



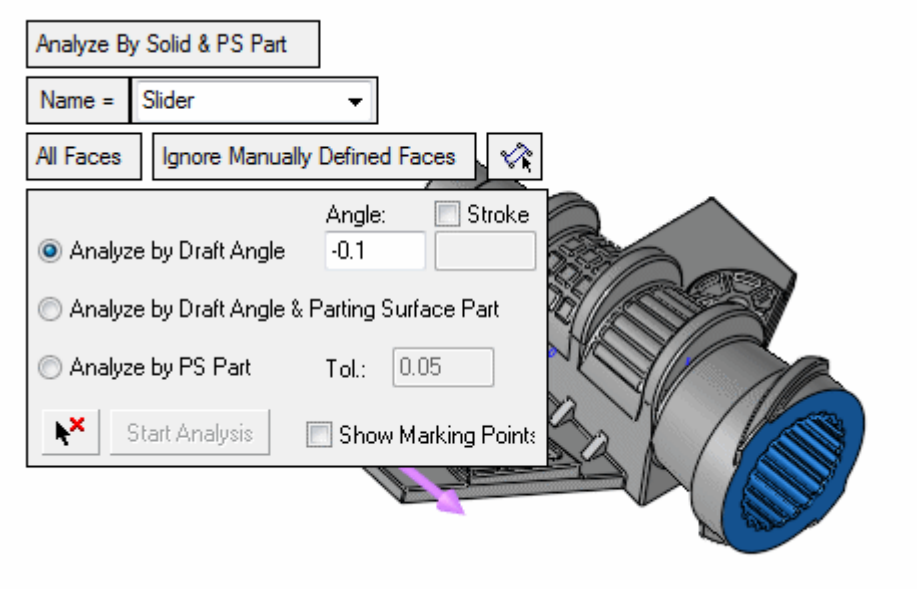
Quick Split Improvements – Show/Hide Marking Points

1. You can **Show/Hide Marking Points** to view undercuts areas. See relevant checkbox in the dialog:

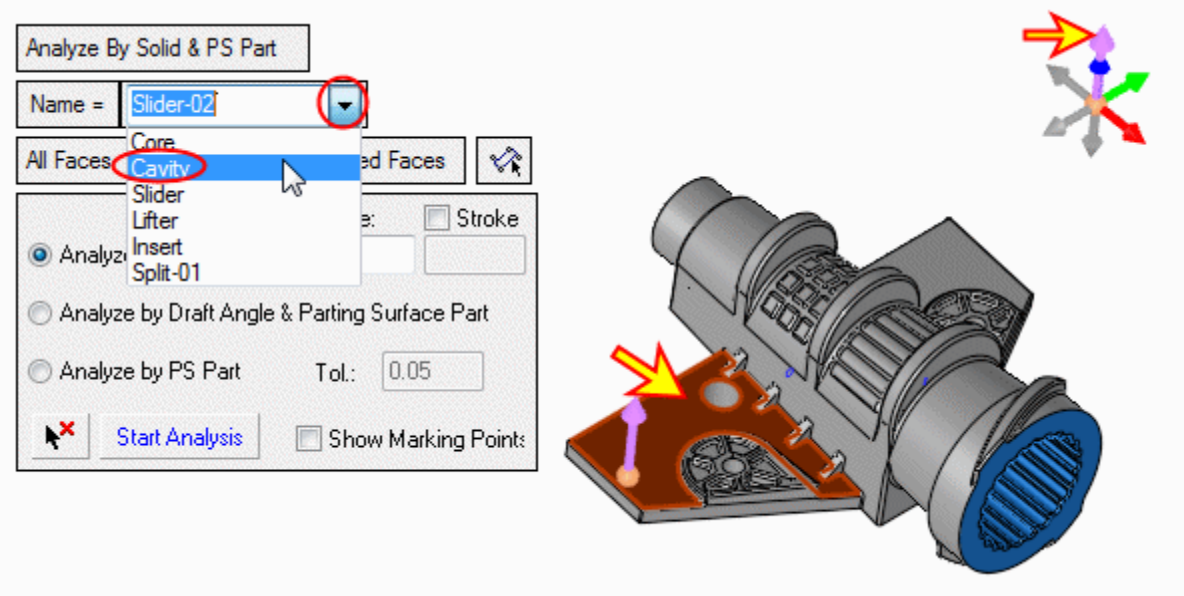




2. Press **Apply**.

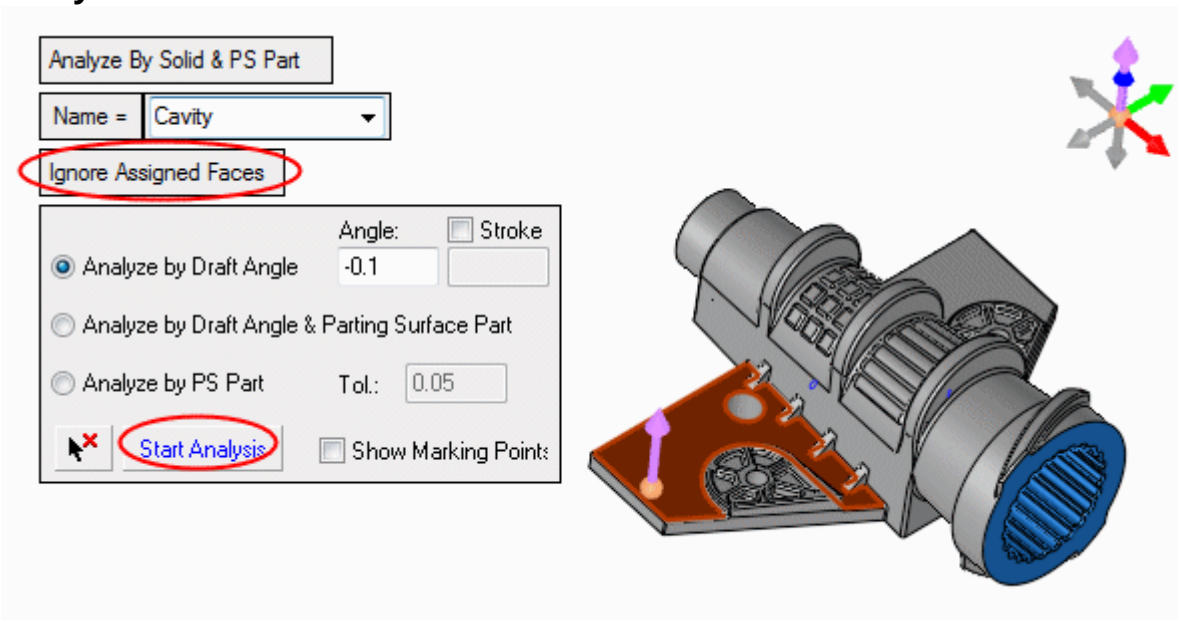


3. Change the direction arrow to **Along Y**. Select a split direction **Name** from the dropdown menu. Select, **Cavity**.

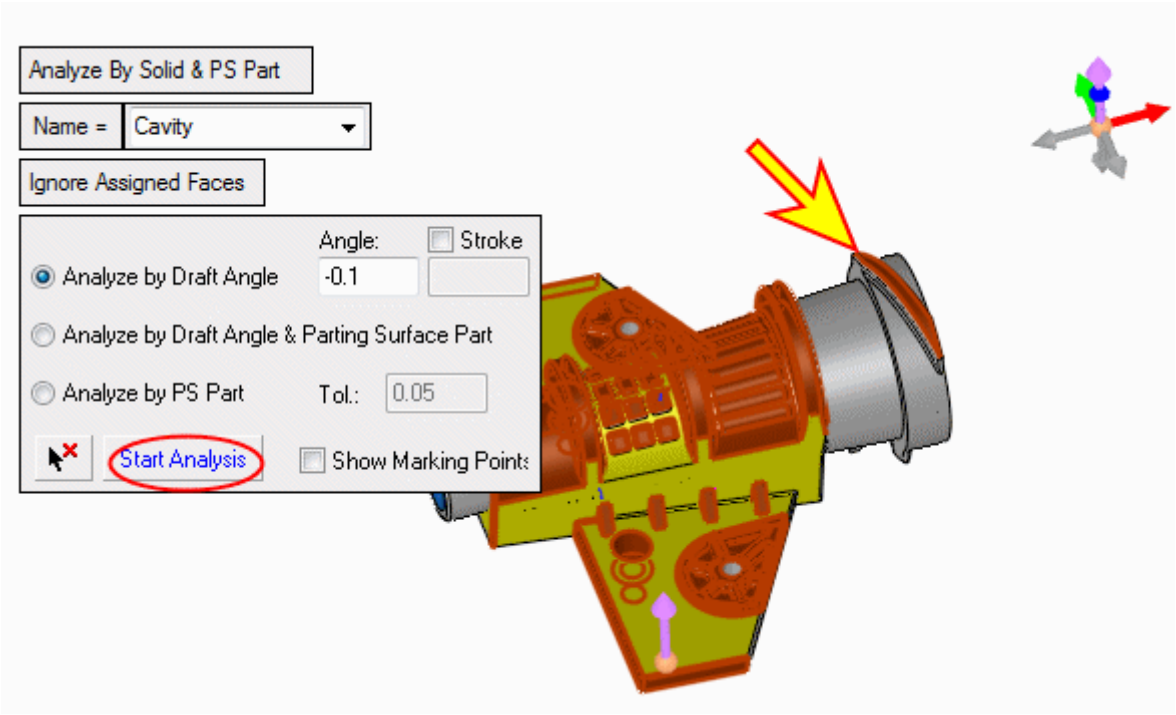


Note: The list of default Set names can be edited in the **Preferences**.

4. Ensure that the **Ignore Assign Faces** option is displayed. Press the **Start Analysis** button.

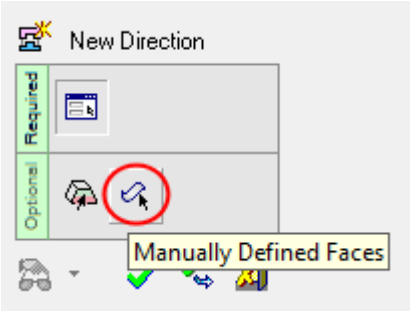


5. Select another area by selecting the face indicated by the arrow below and press the **Start Analysis** button.

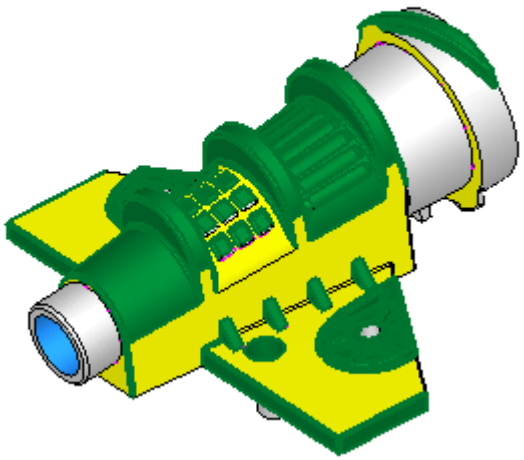


Quick Split Improvements – Manually Defined Faces

1. Enter the second Optional stage, **Manually Defined Faces**.



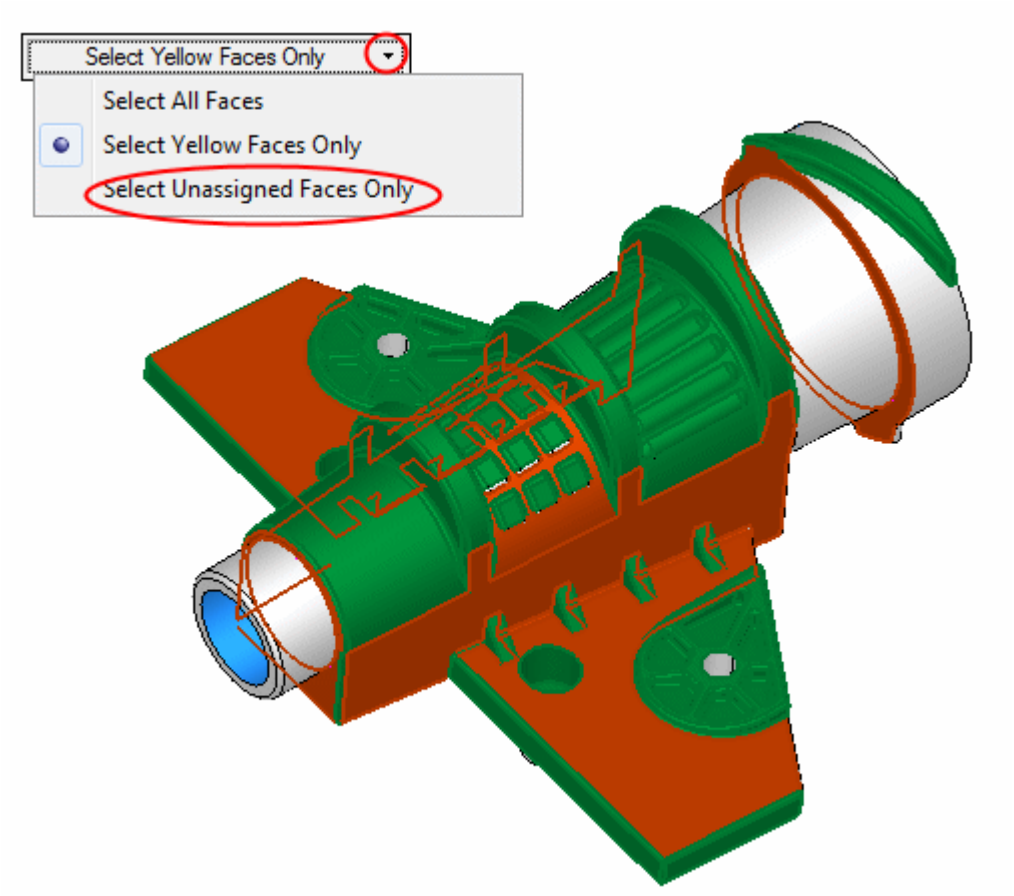
2. Accept the default option, **Select Yellow Faces Only**, and press the **Select All** button  .



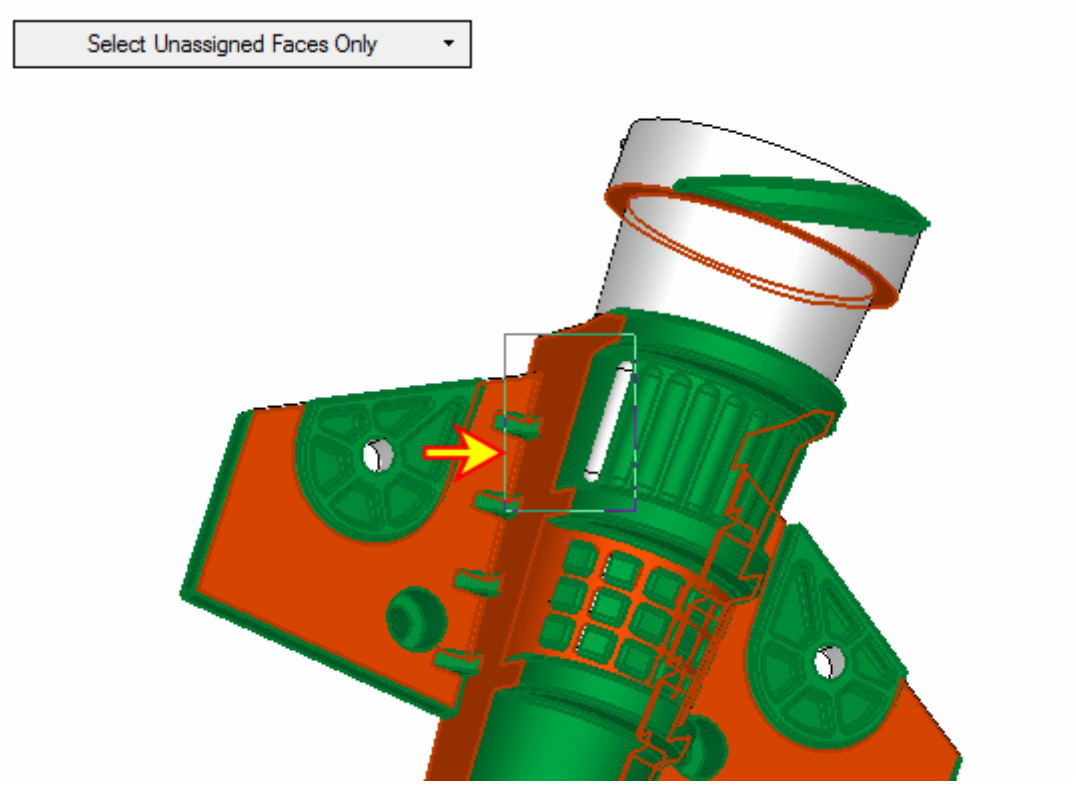
3. Press the dropdown menu. Notice there are three types of selections:

    - **Select All Faces**
    - **Select Yellow Faces Only**
    - **Select Unassigned Faces Only**

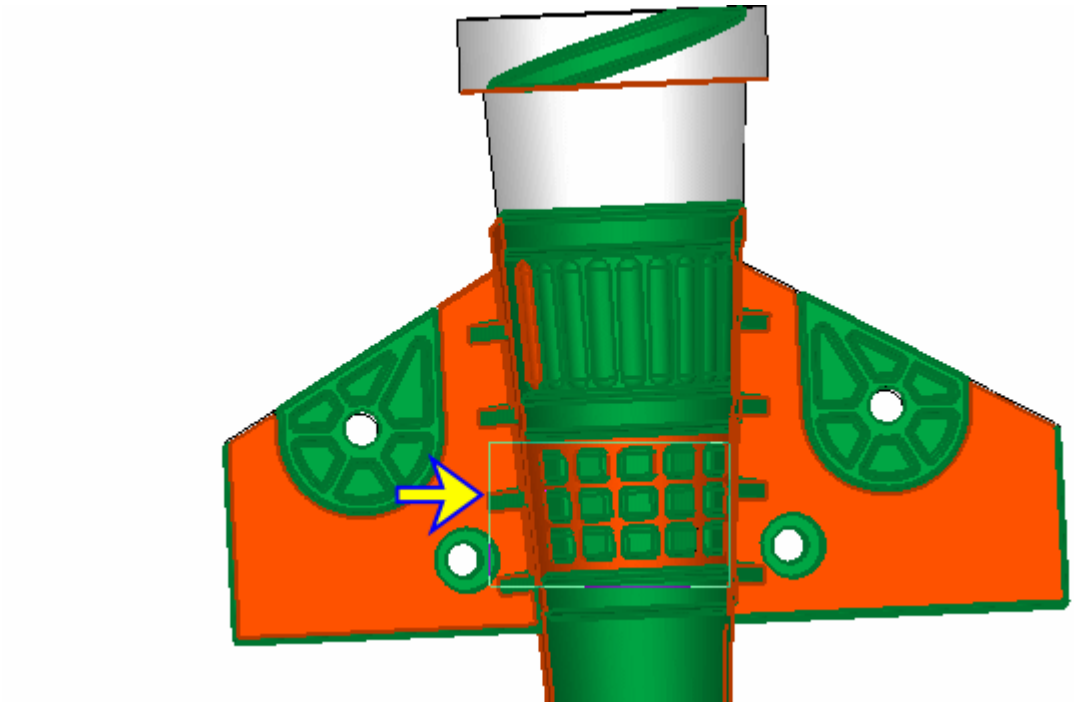
    Use these options to easily select the appropriate desired faces. Select the **Select Unassigned Faces Only** option.



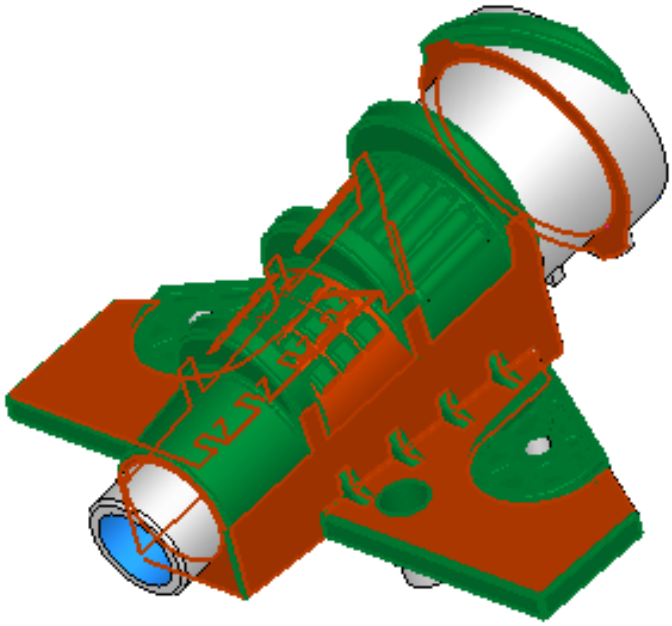
4. Use the **Visible Faces Only** mode and select **by box** the faces indicated by the arrow below:



5. Select more faces **by box**:

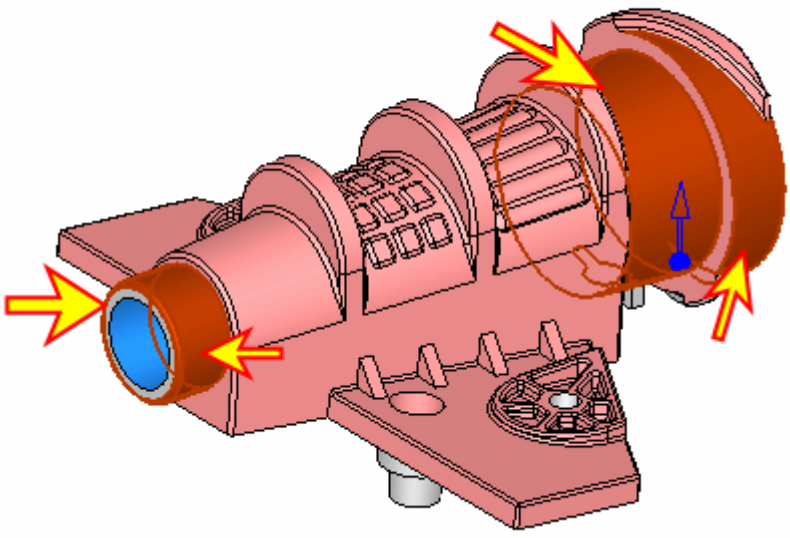


6. Press **OK**.

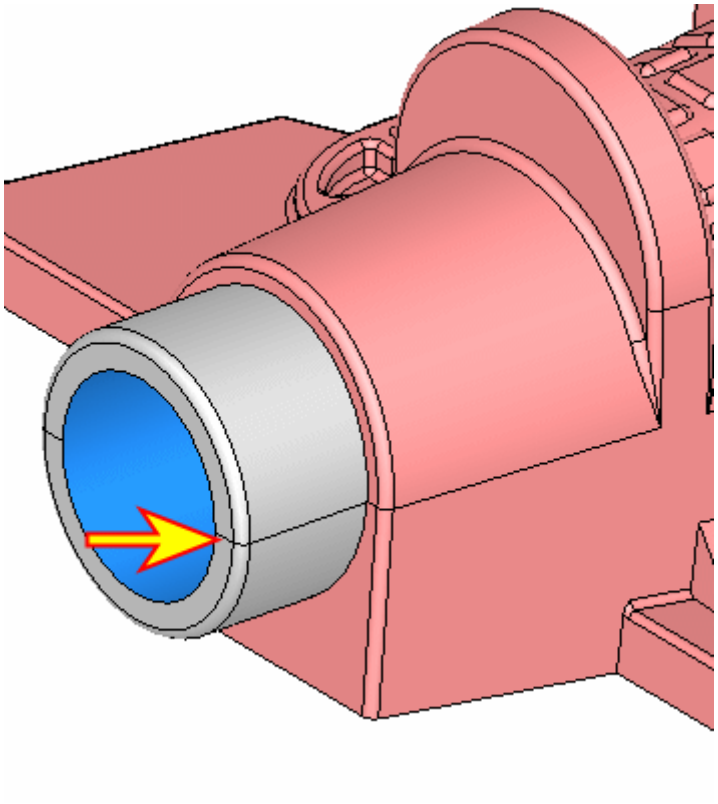


To demonstrate the next new improvement, we will first fix some undercuts.

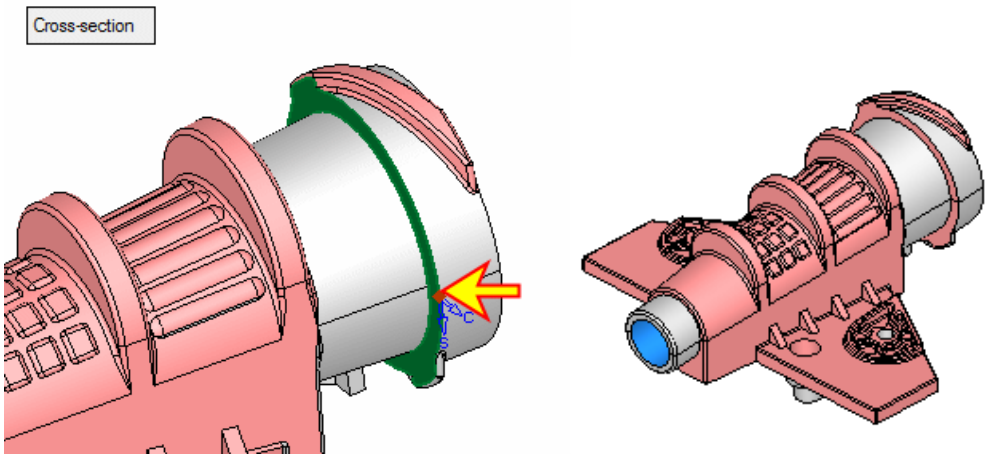
7. Split silhouette (select **Faces > Modify Faces > Split Silhouette > Along Z**) the faces indicated by the arrows below:



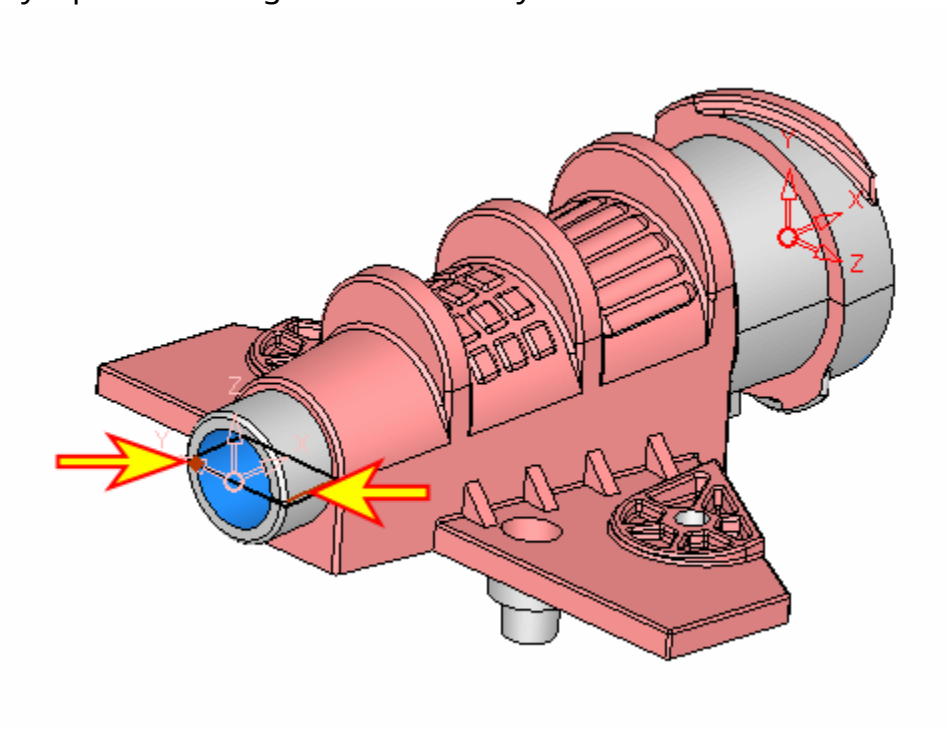
8. Split the face indicated by the arrow below:



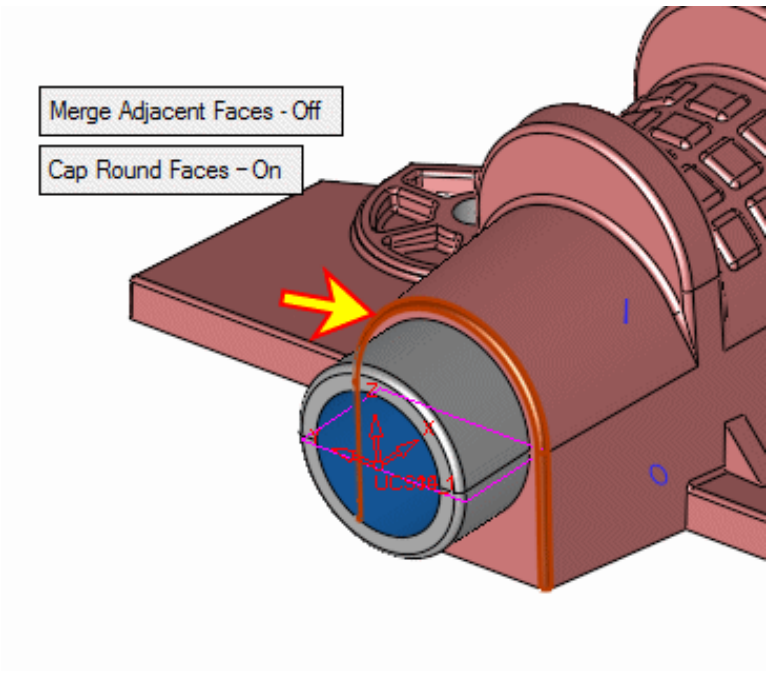
9. Split another indicated by the arrow below:



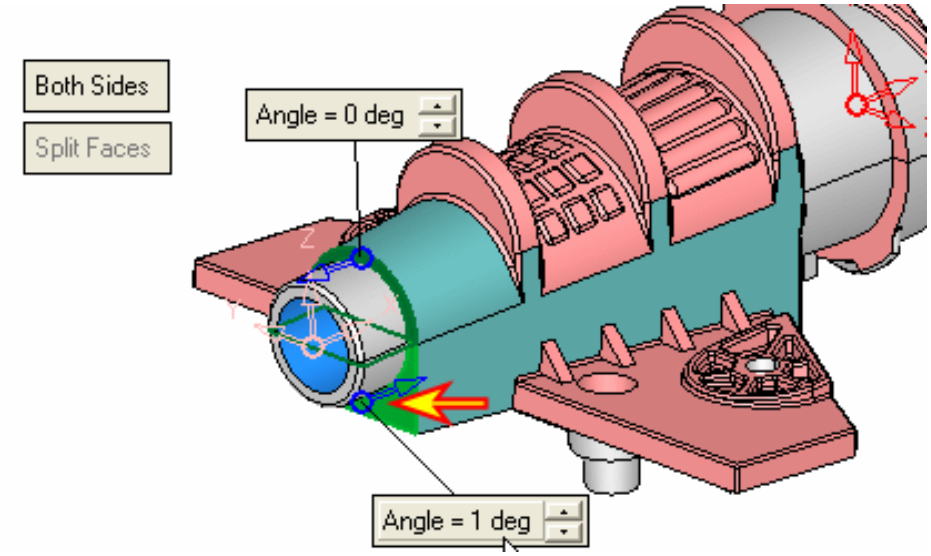
10. We will now add a taper to the front face. Create a datum plane defined by a point and edge as indicated by the arrows below:



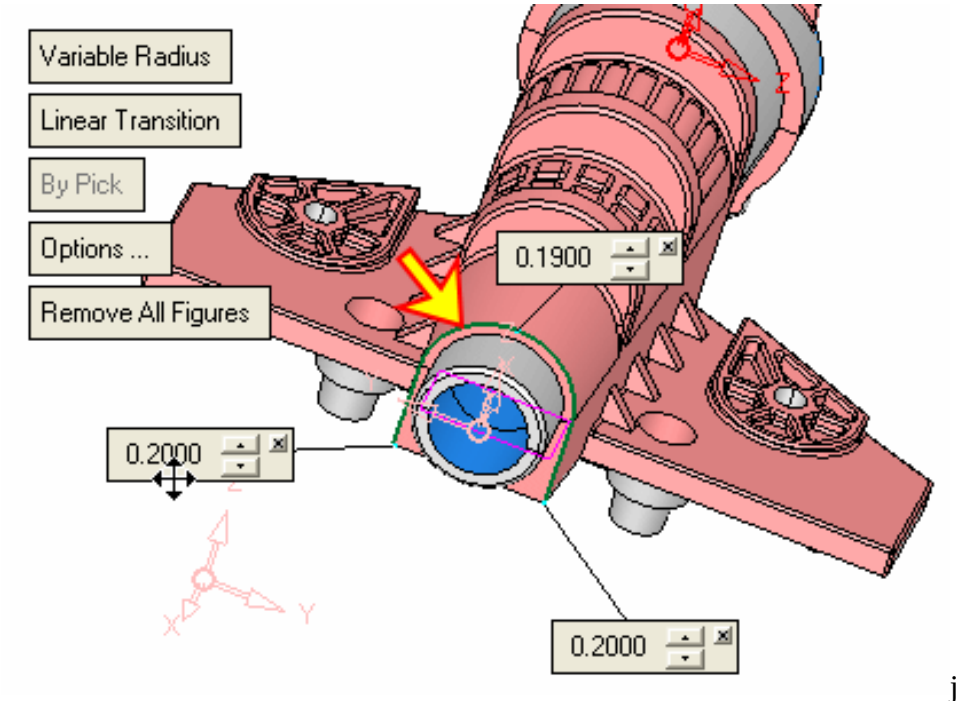
11. Remove and extend the rounded faces.



12. Taper the face indicated by the arrow below. Use the previously defined plane as reference. Split both sides with a different draft angle.



13. Create a variable round as shown below:

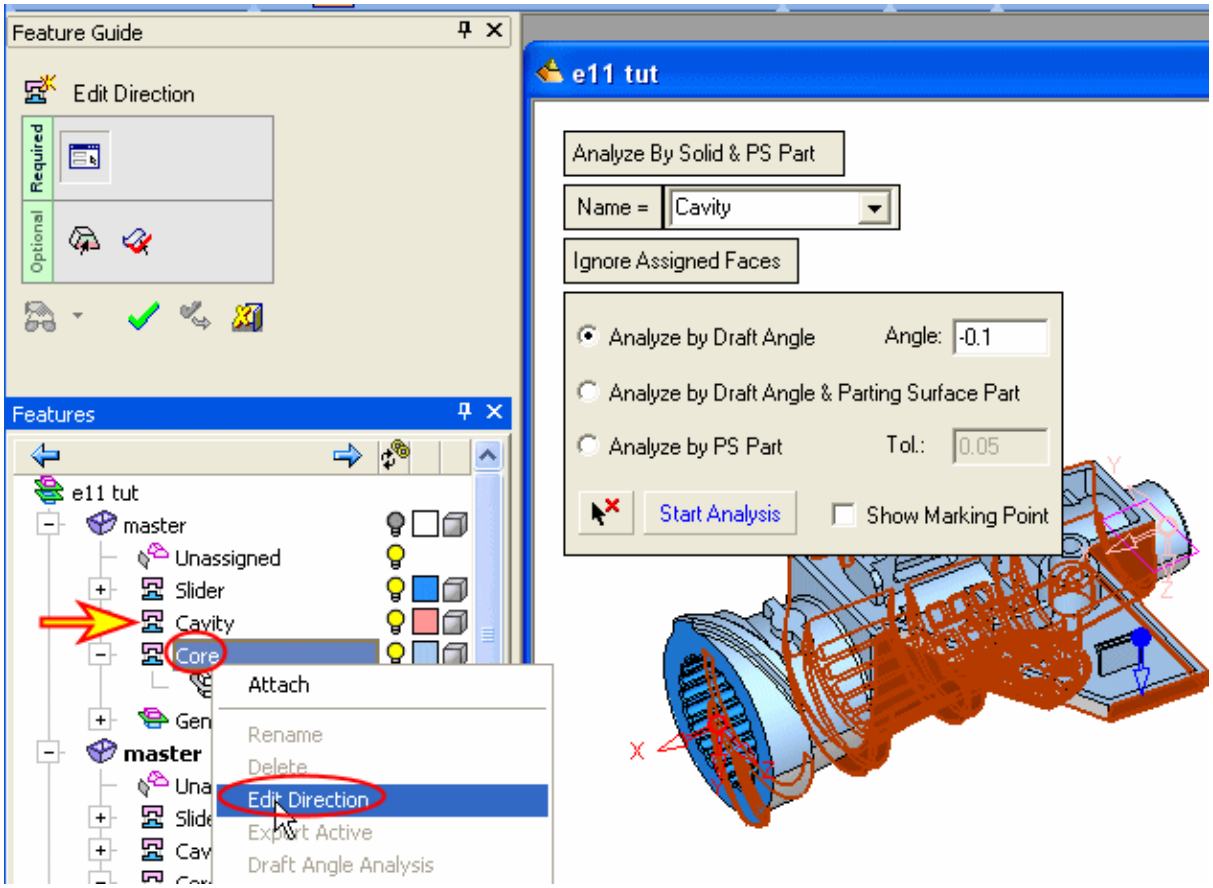


j

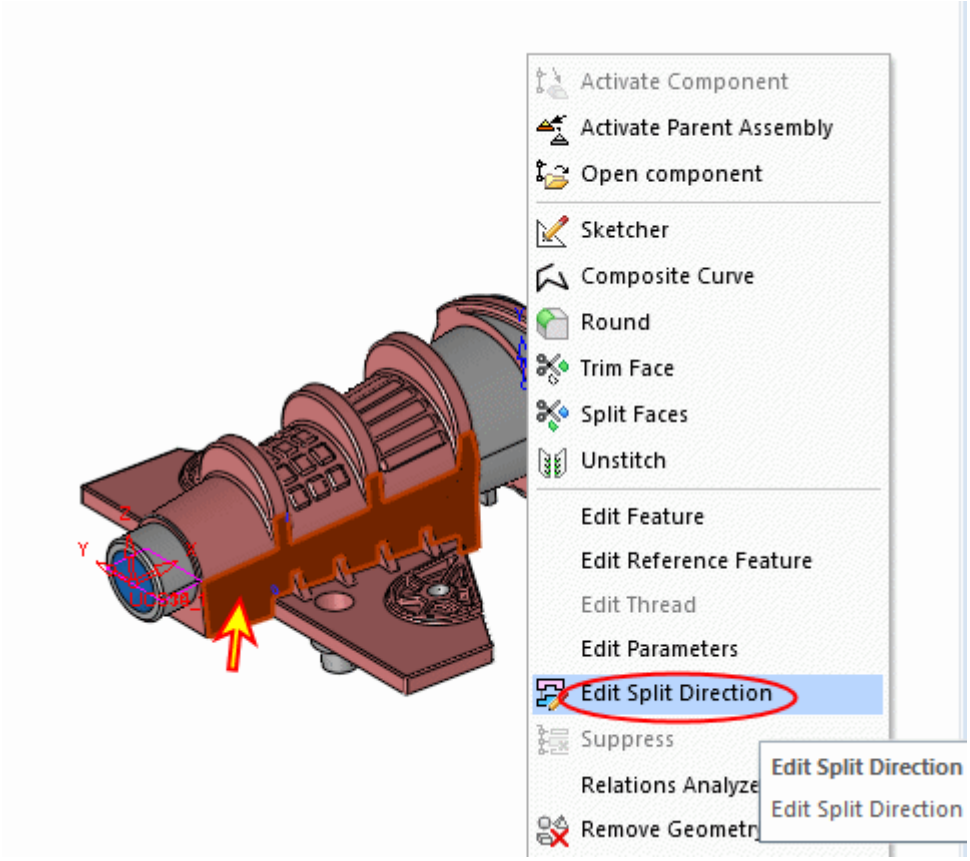
Quick Split Improvements – Edit Direction

Note: Edit Direction is available from the context menu on a face.

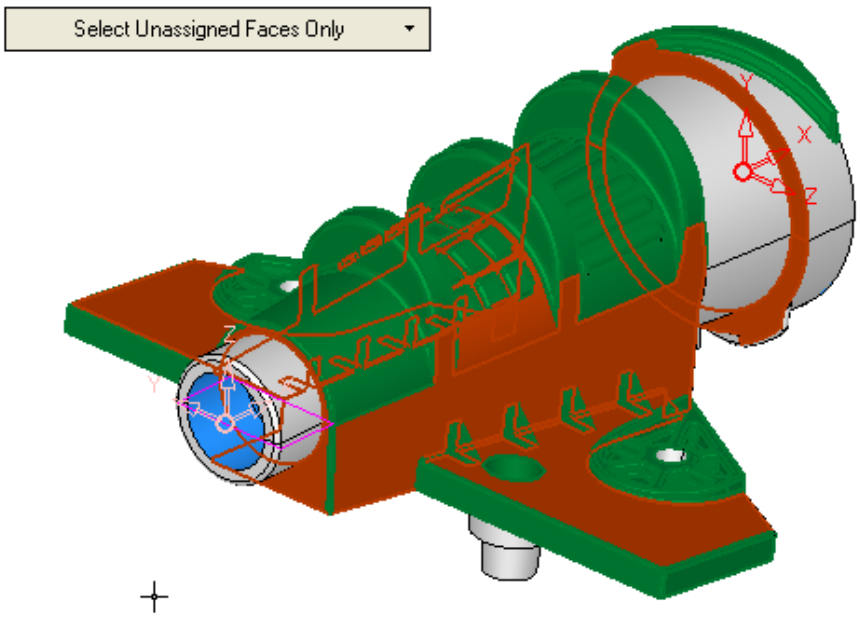
Accessing the **Edit Direction** menu does not require activating the work part. You may also access the Edit Direction option while being in the edit direction stage of another split direction (see the image below). (You may edit the core set while the cavity set is being edited.)



1. **Right-click** the face indicated by the arrow below and select **Edit Direction**.



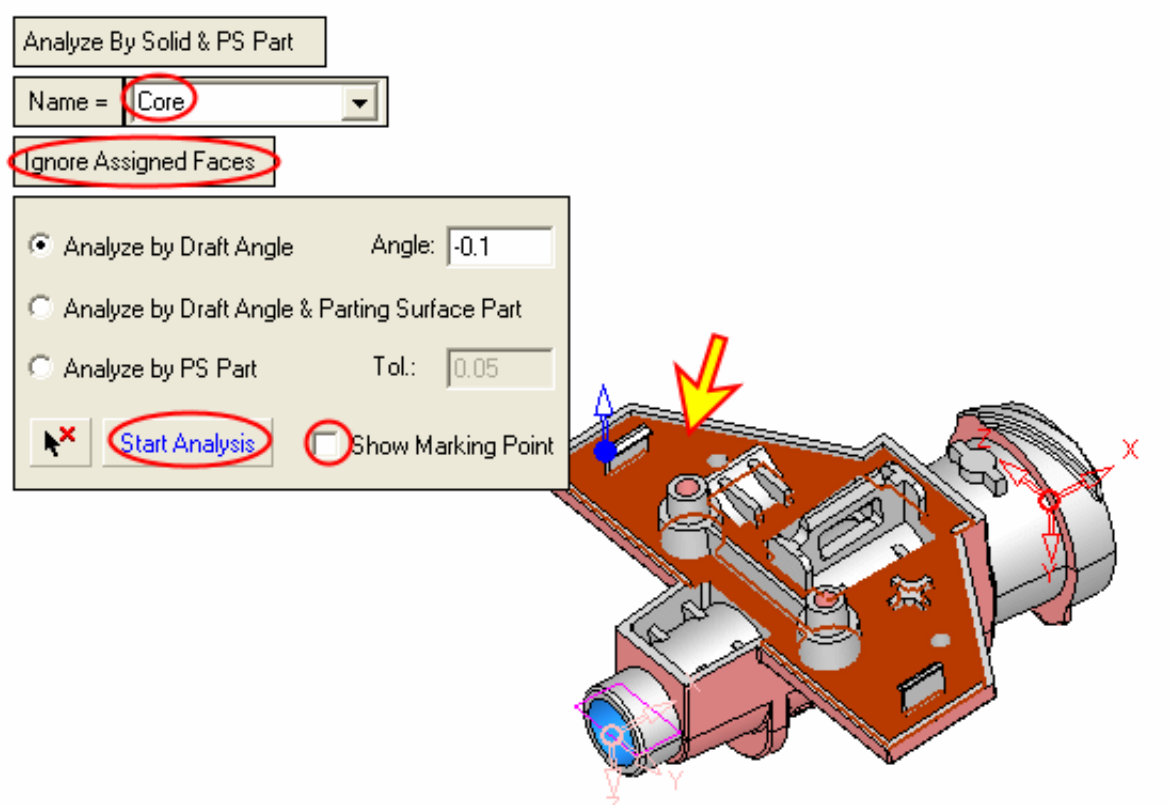
2. Enter the **Manually Defined Faces** step and note the **RED** faces that were manually selected.



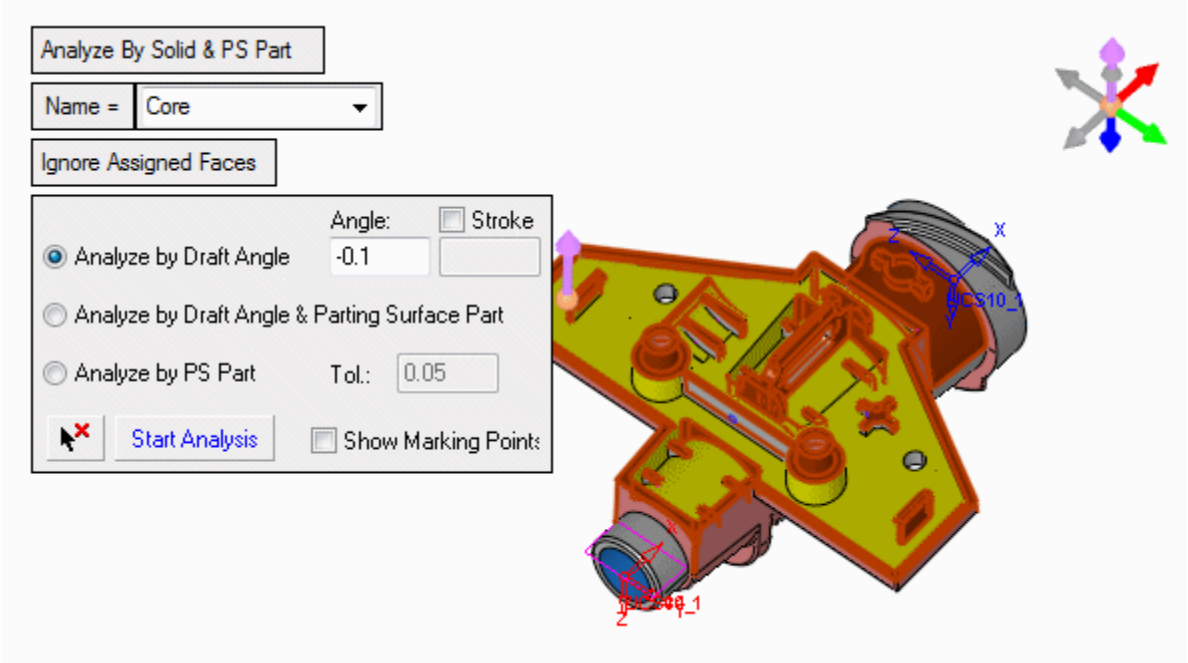
Quick Split Improvements – Allow Overwrite Manually Defined Faces

1. Create a new split direction, **Along -Z**.

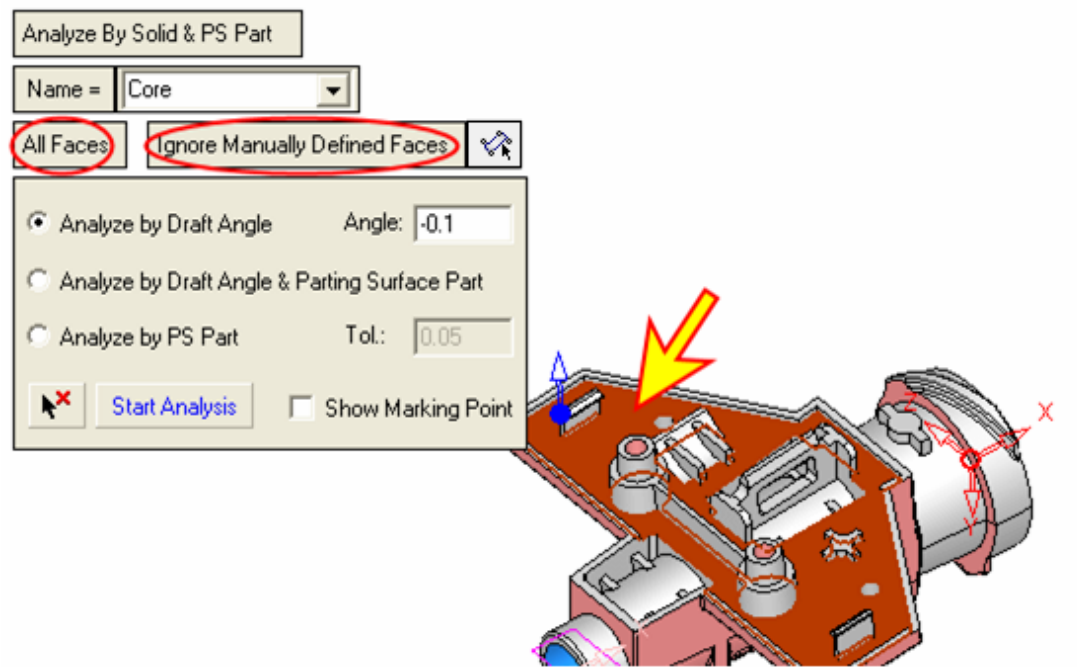
2. Name the new direction, **Core**. Ensure that the **Ignore Assigned Faces** option is selected. Uncheck the **Show Marking Point** option and select the face indicated by the arrow below. Press **the Start Analysis** button.



3. Examine the result.



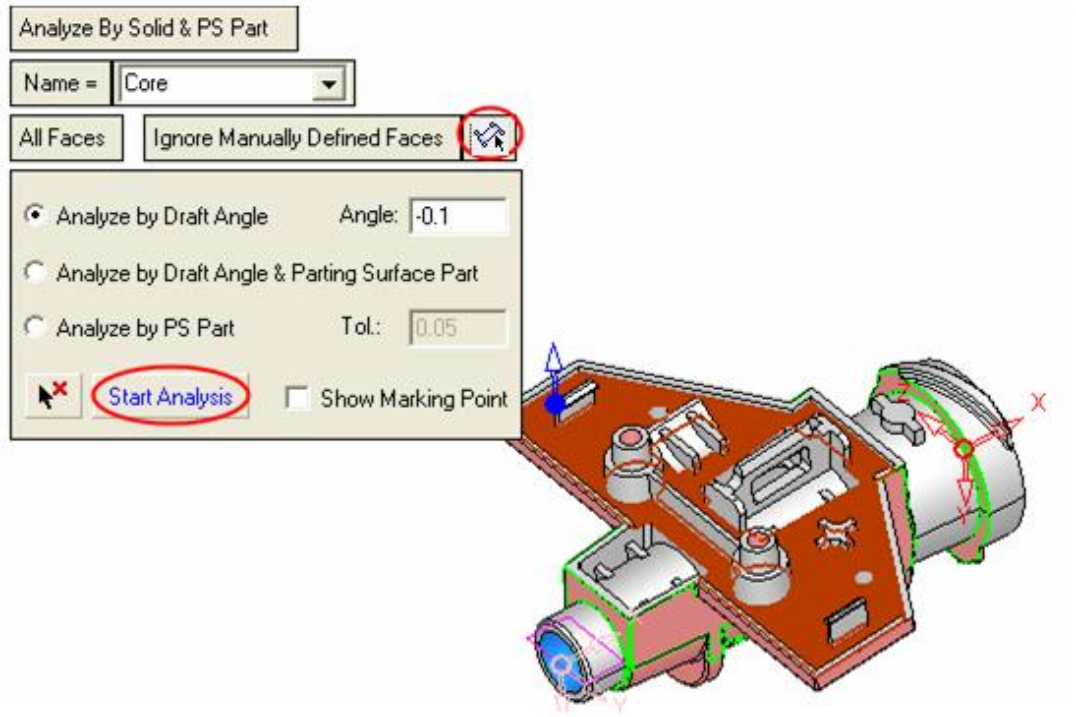
4. Unselect all faces. Toggle the **Ignore Assign Faces** option to **All Faces**. Accept the default **Ignore Manually Defined Faces** option. Select the face indicated by the arrow below:



5. Turn on the **Face** icon marked in **RED** circle below.

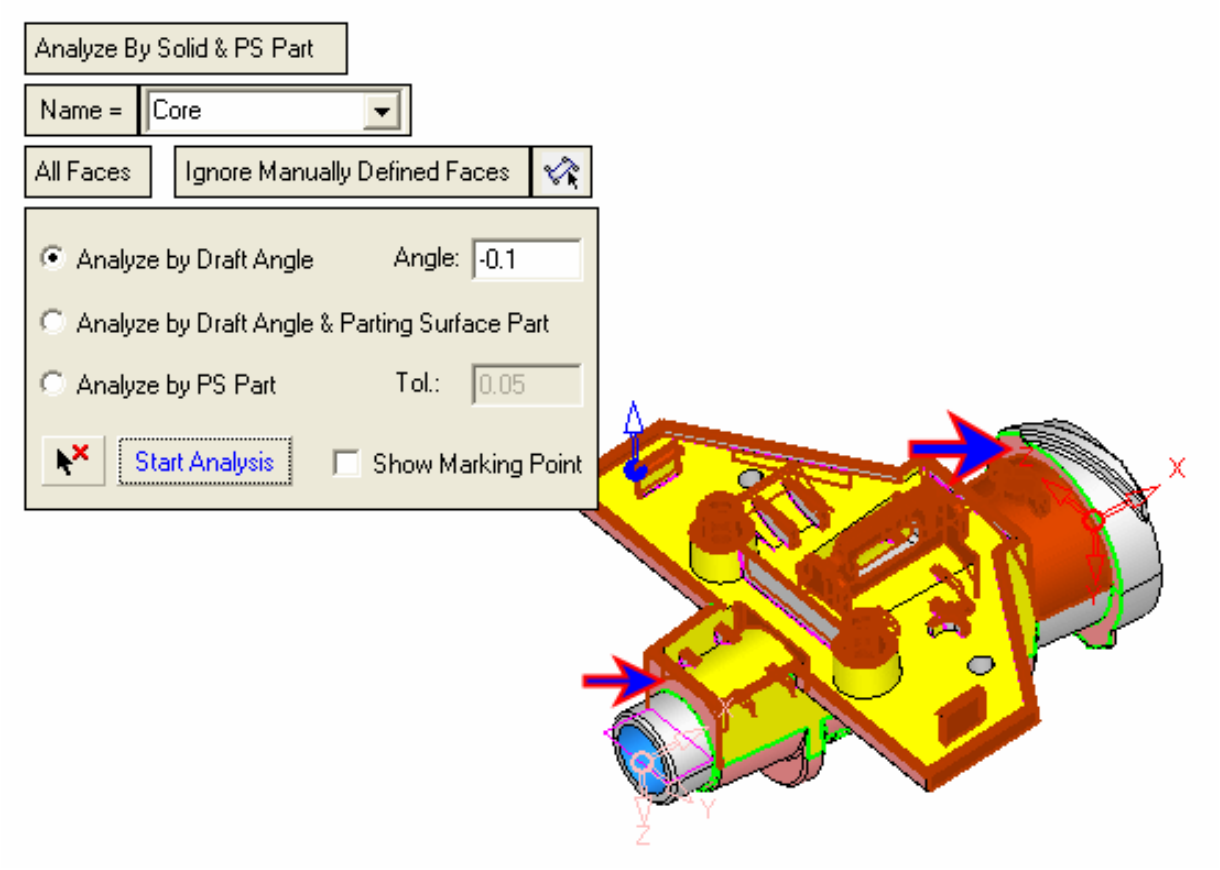
   Notice the highlighted faces (**GREEN** boundaries). These are the manually selected faces of other split direction sets. As in previous versions, although selecting the **All Faces** option, these manually selected faces will not be selected (the analysis will not overwrite these faces) when pressing the **Start Analysis** button.

6. Press the **Start Analysis** button.

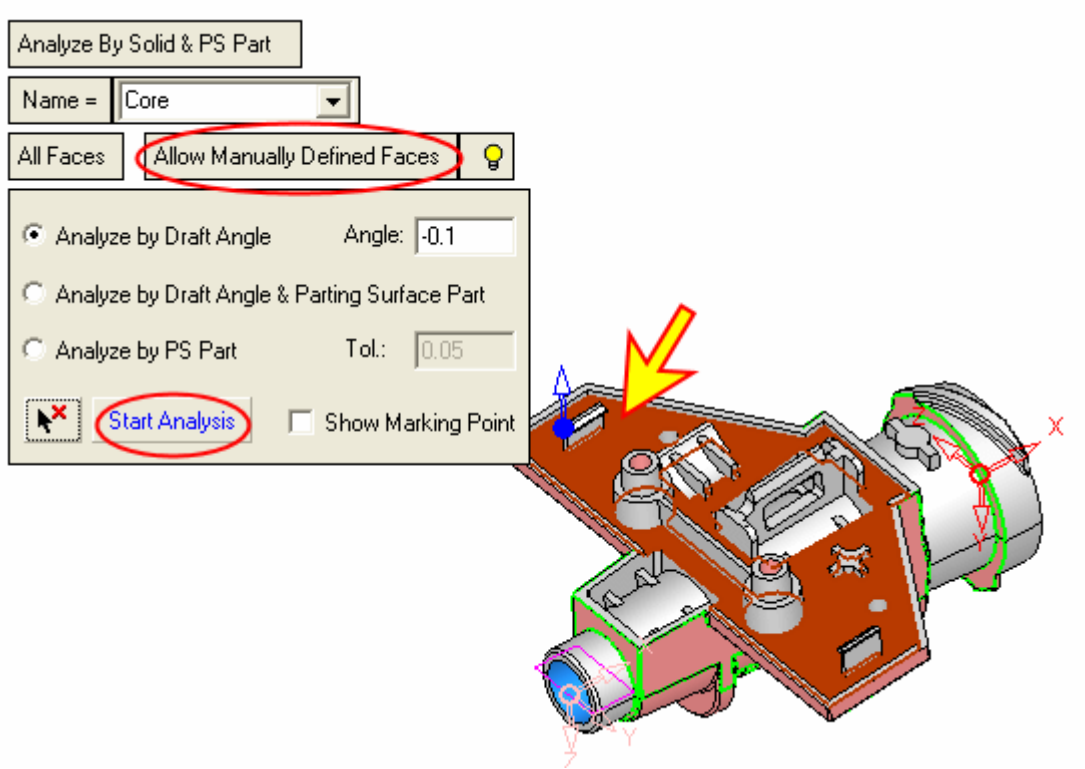


Note: The two faces indicated by the arrows in the following image belong to the cavity (manually selected faces). These faces were not selected in the current analysis due to the use of the **Ignore Manually Selected Faces** option, although they are suitable for the core split direction.

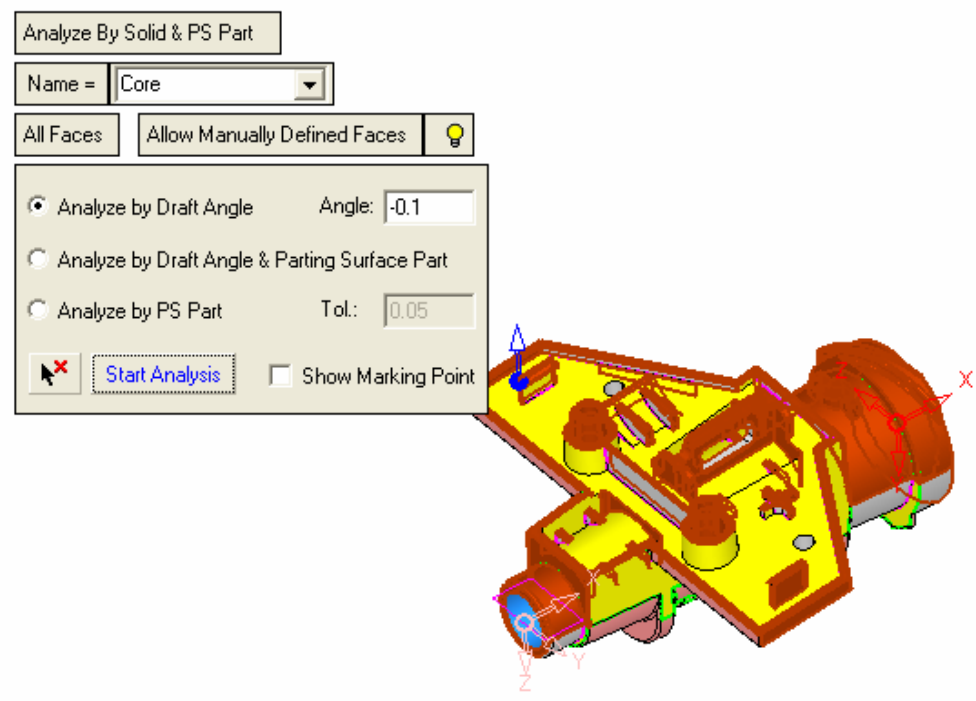
Note: The system does show **YELLOW** faces, even if were selected manually by other split sets.



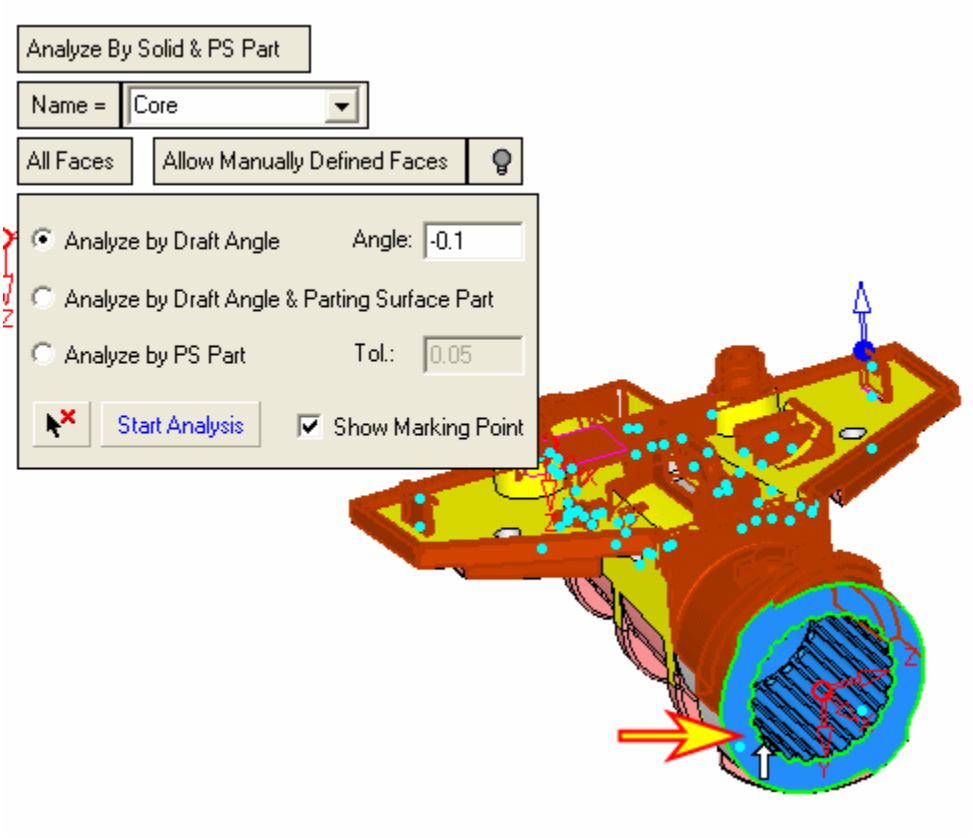
7. Unselect all faces. Toggle the **Ignore Manually Defined Faces** option to **Allow Manually Defined Faces**. Select the **Start Analysis** button.

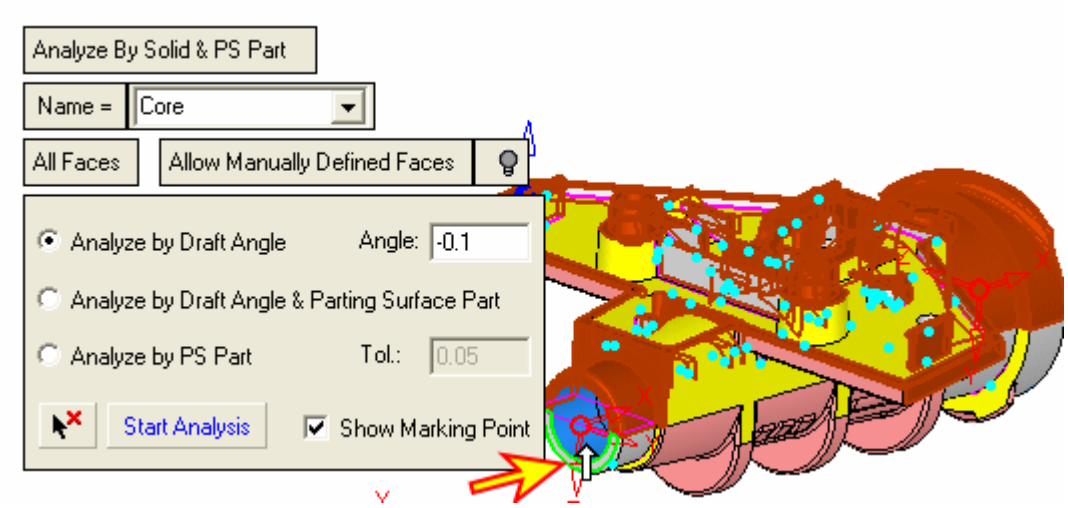


8. Note that this time the system does overwrite manually defined faces of other sets.

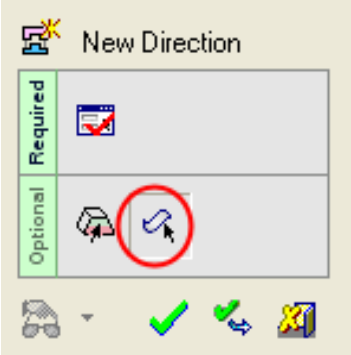


9. Unselect the faces indicated by the arrows below:

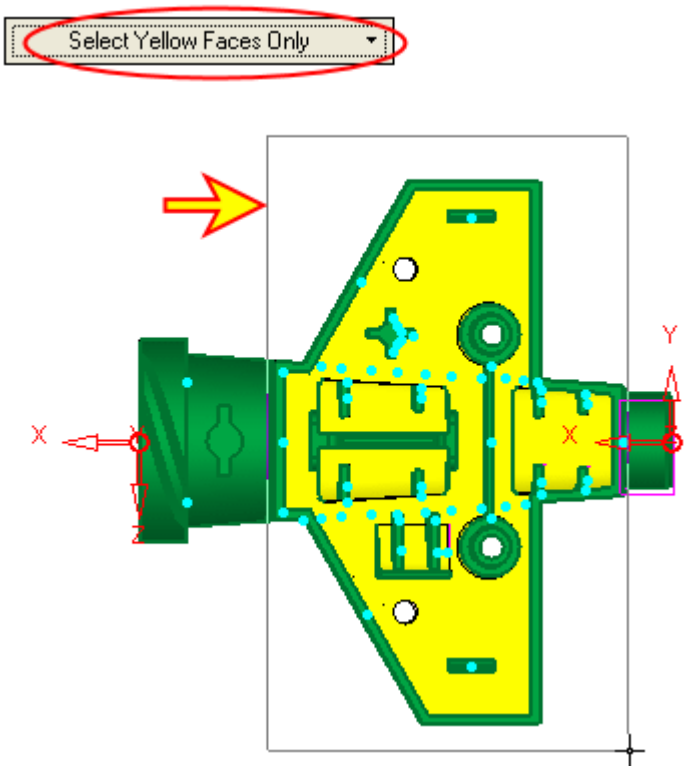




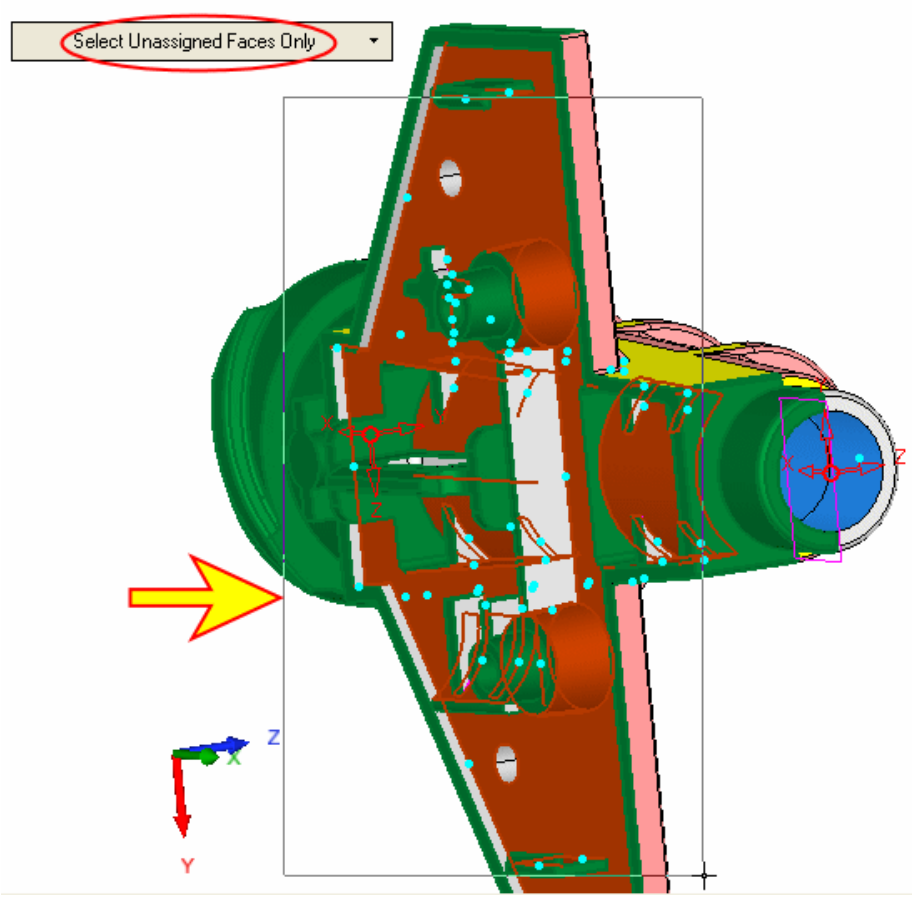
10. Enter the **Manually Defined Faces** stage to select other faces to be included in the split direction.



11. **Select Yellow Faces Only** by box (visible only mode).



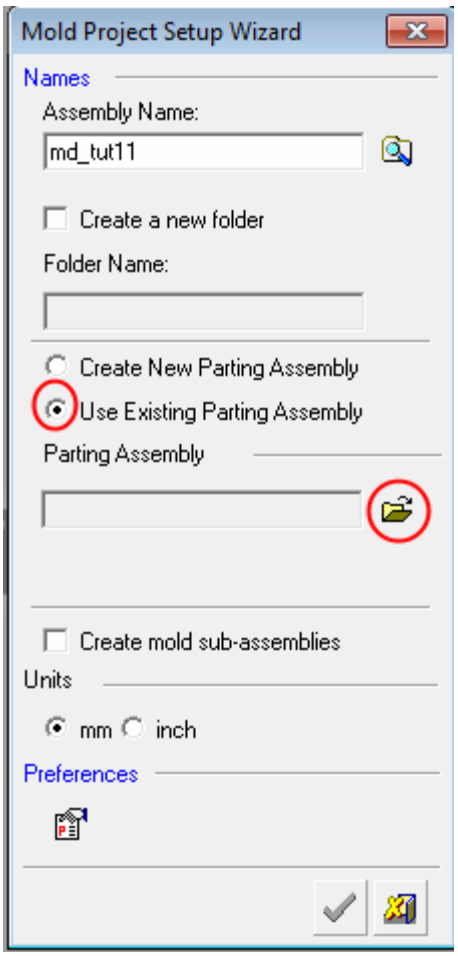
12. **Select Unassigned Faces Only** by box. Press **OK** and **save** and **close** the assembly.



Mold Project Setup – Use Existing Parting Assembly

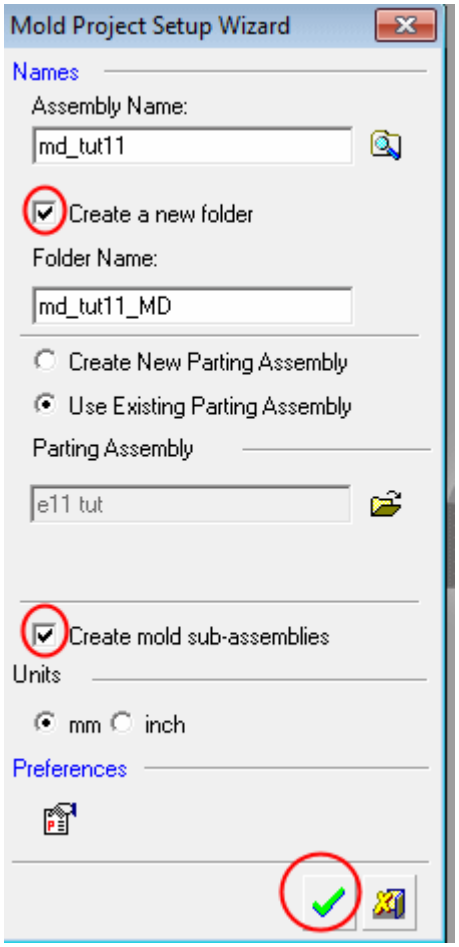
1. Open the **Mold Project Setup Wizard**. Check the **Use Existing Parting Assembly** option and load the previously saved parting assembly file.

Note: The **Use Existing Parting Assembly** option enables a better solution for dividing the work between the person who does the parting job and the mold designer.

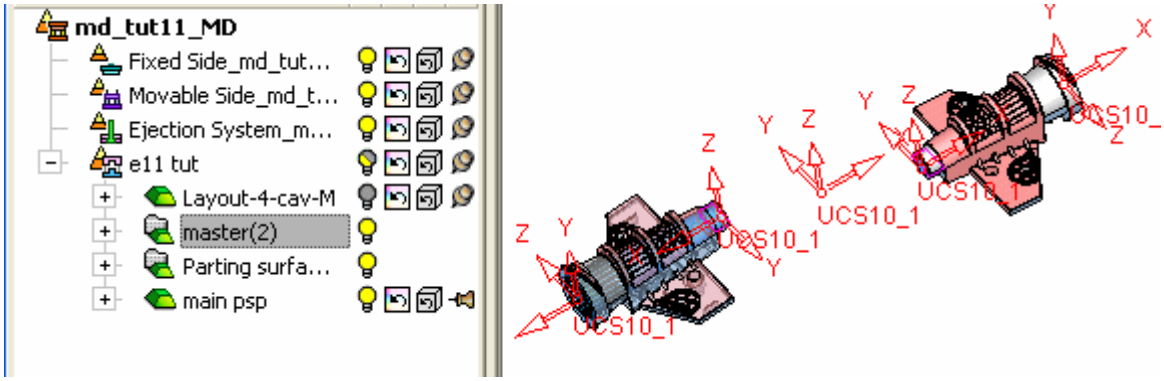


2. Check the **Create a New folder** and **Create Mold Sub-assemblies** options.

Note: By unchecking the **Create Mold Sub-assemblies** check box, the Mold assembly will be loaded with the parting assembly only (without the Fixed, Movable, and Ejection system assemblies).



3. The Mold assembly was loaded with the ready parting assembly. **Close the file without saving.**

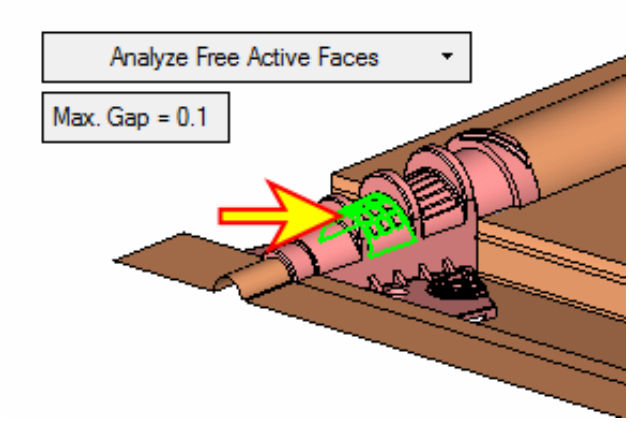


Analysis Tools – Parting Analytic Tools - Analyze Free Active Faces

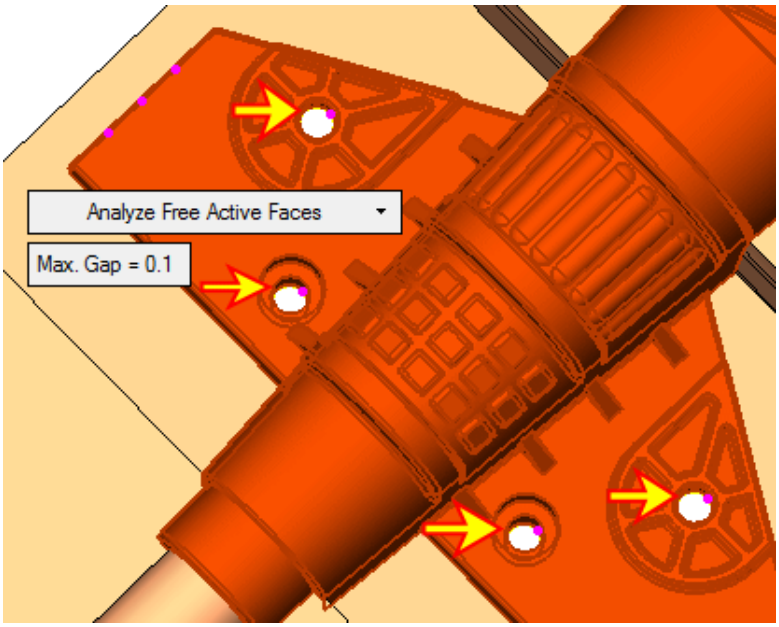
1. Unpack the **mold_MD_after parting** file. This assembly already includes parting faces (without attributes).

   The **Analyze Free Active Faces** option is a quick analysis tool to detect areas of free active faces which do not have adjacent parting faces.

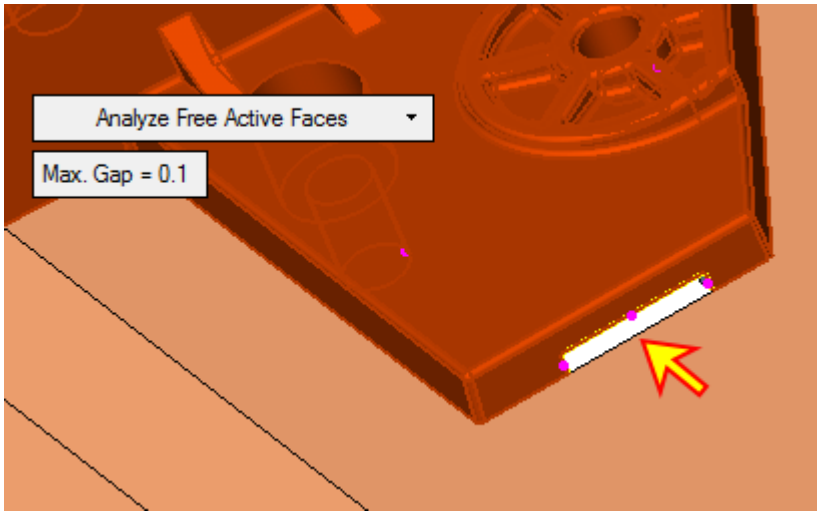
2. Enter the **Parting** menu, select **Parting Analysis Tool > Analyze Free Active Faces**. Select the face indicated by the arrow below (Split 2 direction).



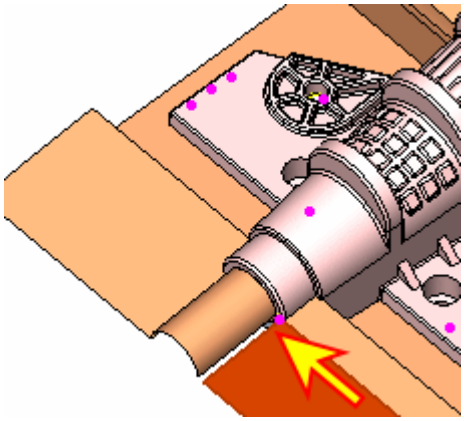
3. Press the **Preview** button. Notice the four internal islands that require parting faces:



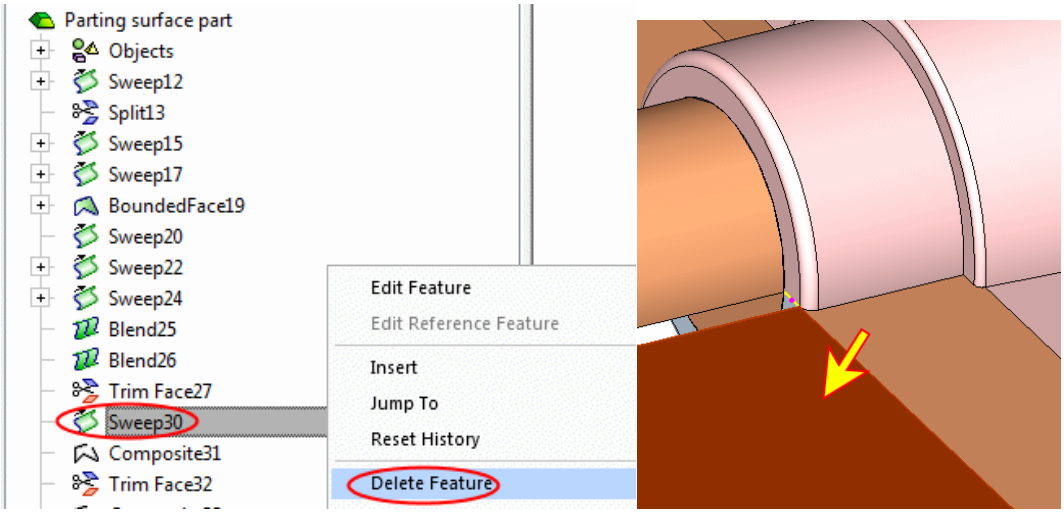
4. Another free active area indicated by the arrow below:



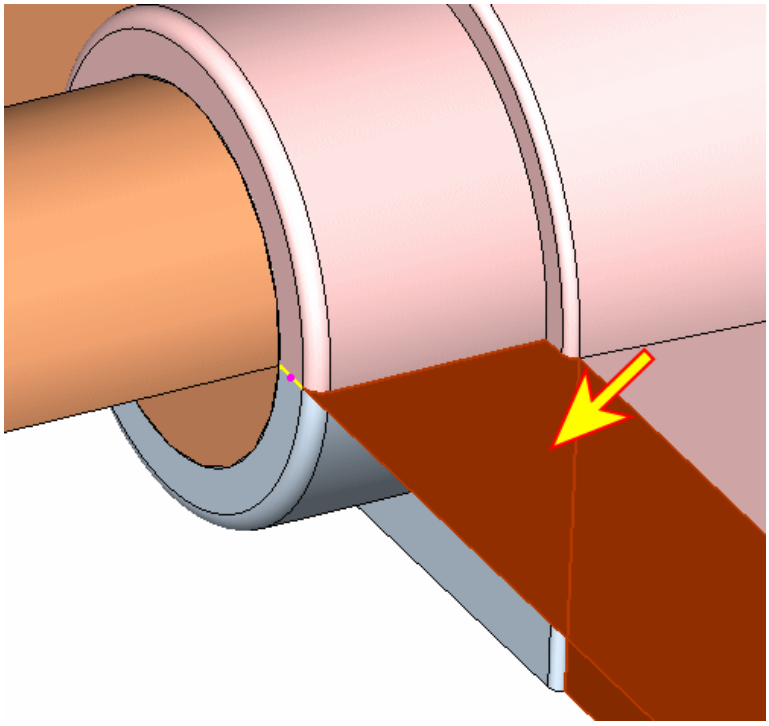
5. Notice the gap indicated by the arrow below. Press **OK** to exit the analysis tool.



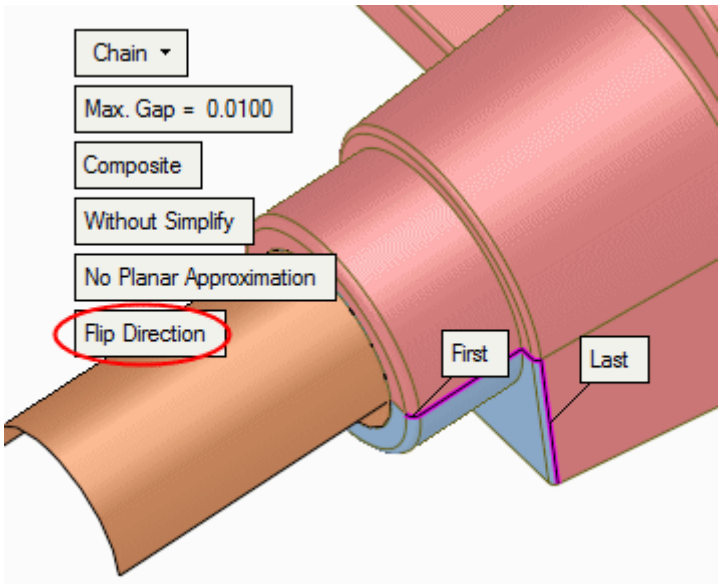
6. To fix the last-mentioned gap, first delete the sweep face (Sweep 30).



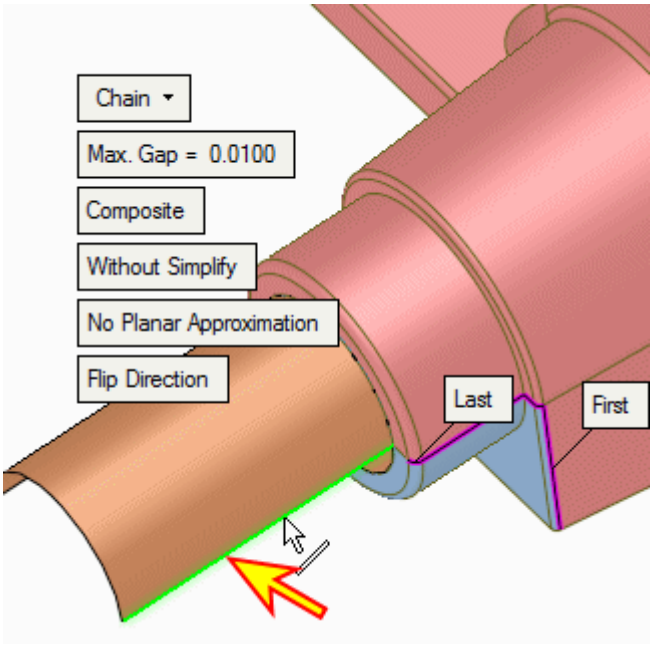
Note: We will now edit the other sweep face (Sweep 22).



7. Edit the sweep face composite by selecting the **Flip Direction** parameter.

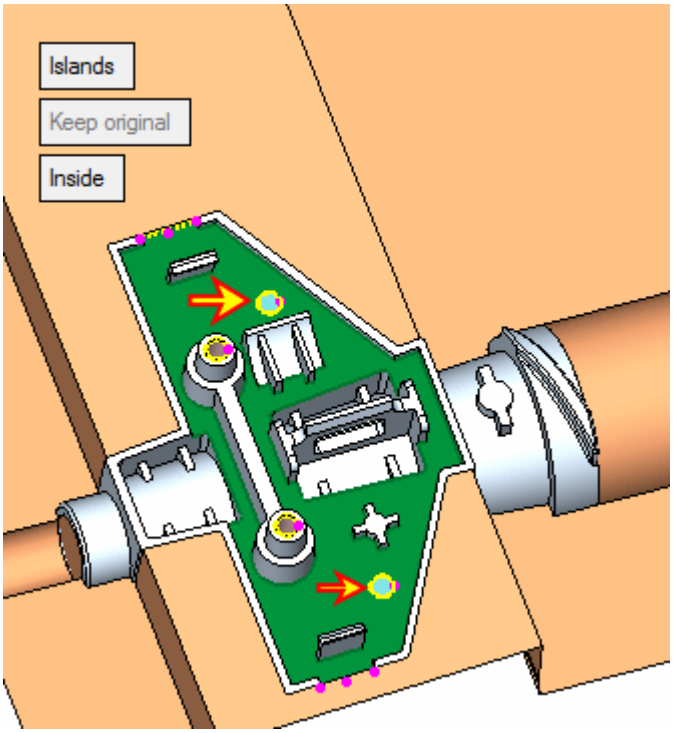


8. Select the edge indicated by the arrow below as the last curve in the composite.

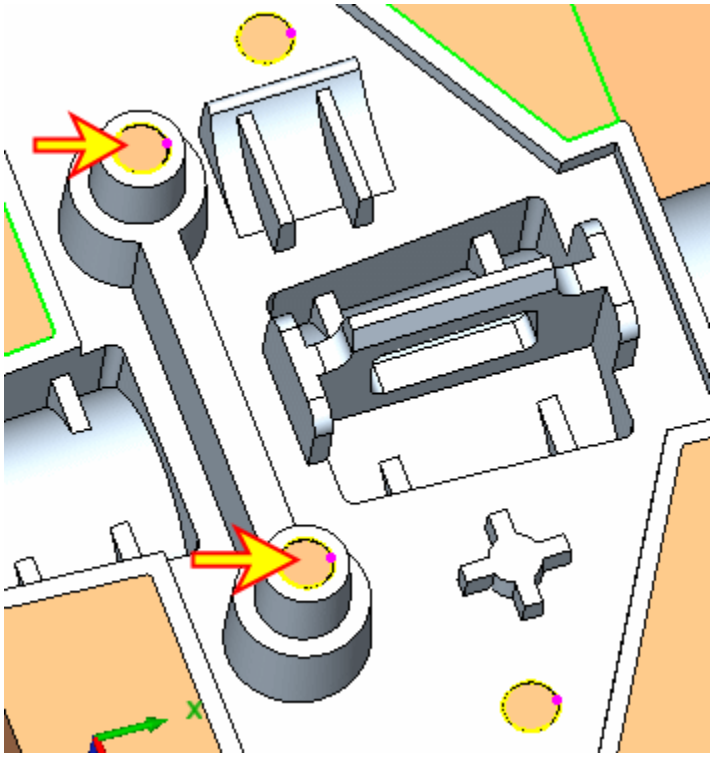


9. Press **OK** to recreate the sweep face.

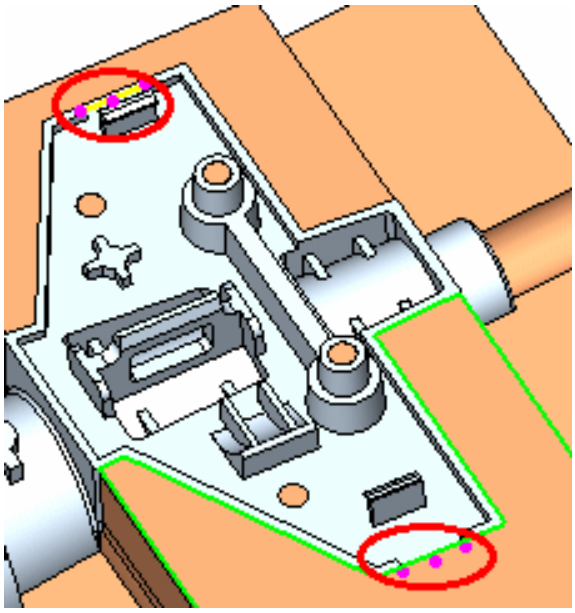
10. Create a boundary face (Islands), leave only the two contours (as indicated by the arrows below) and press **OK**.



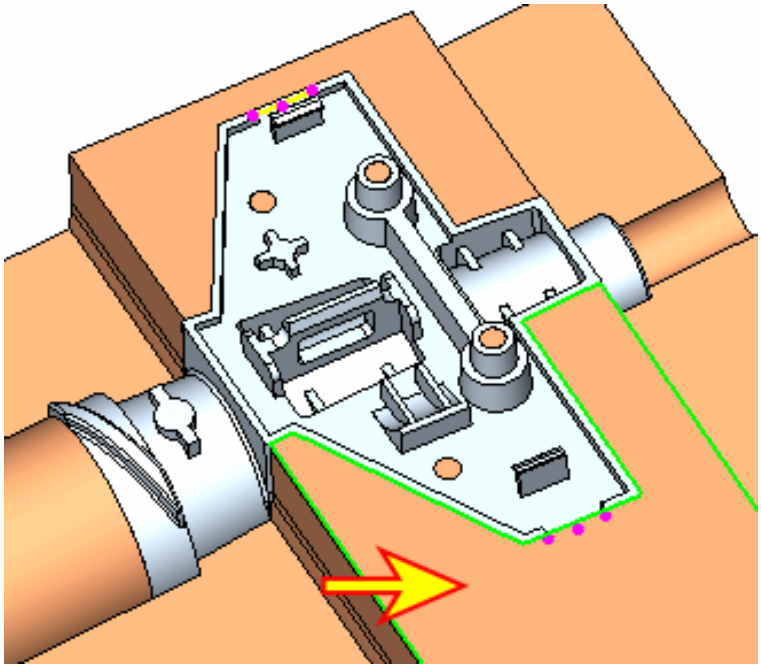
11. Create two more boundary faces:



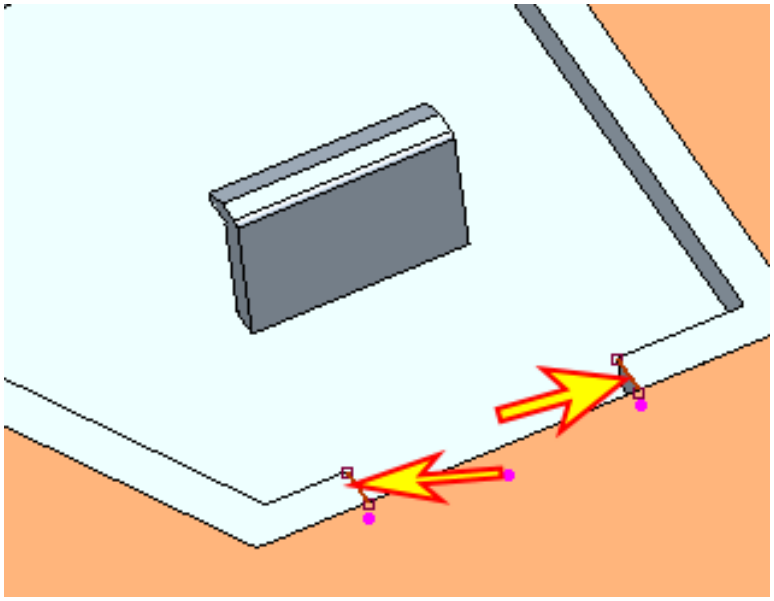
12. Delete all marks of areas that were treated. Note that only two areas were left to deal with:



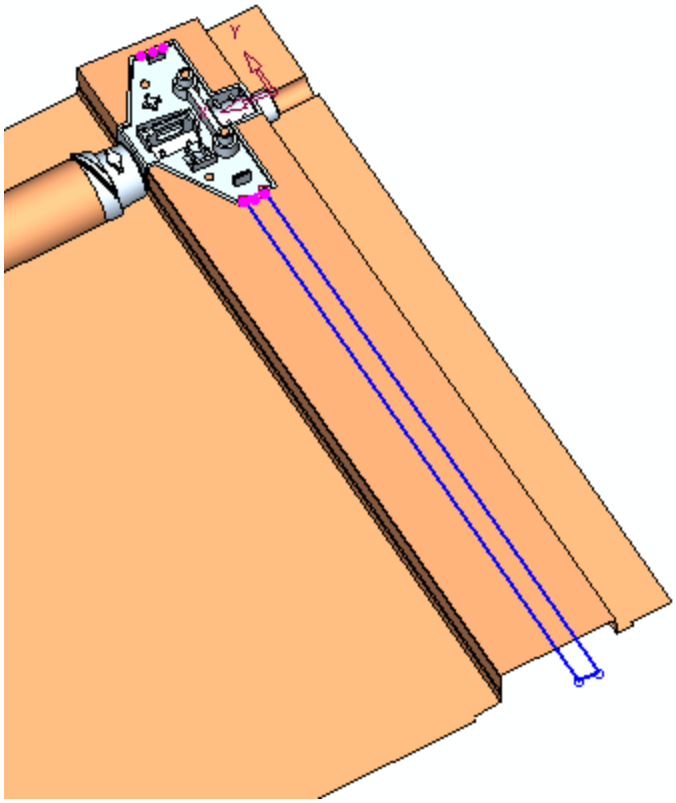
13. Enter the **Sketcher**. Select the face indicated by the arrow below as the sketcher plane:



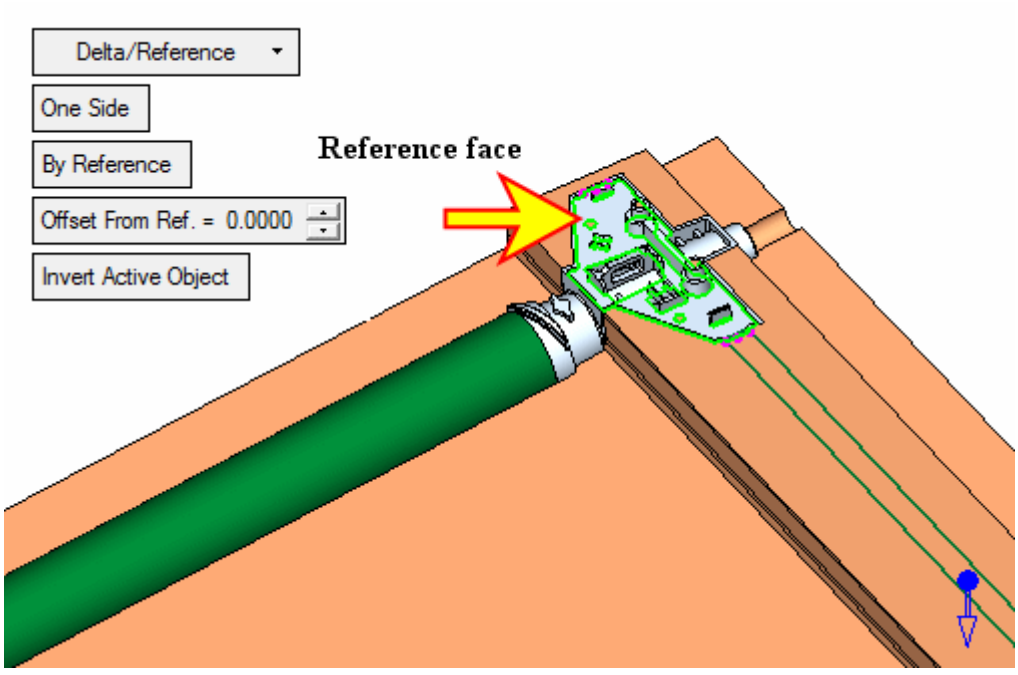
14. Add reference edges:



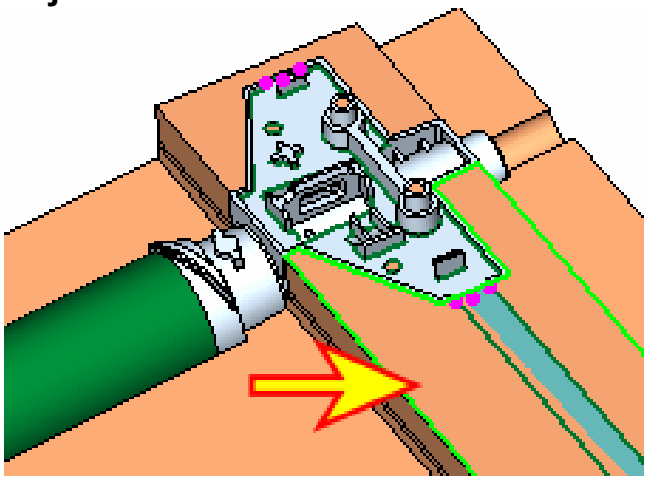
15. Create the sketch shown below, using the references edges previously selected.



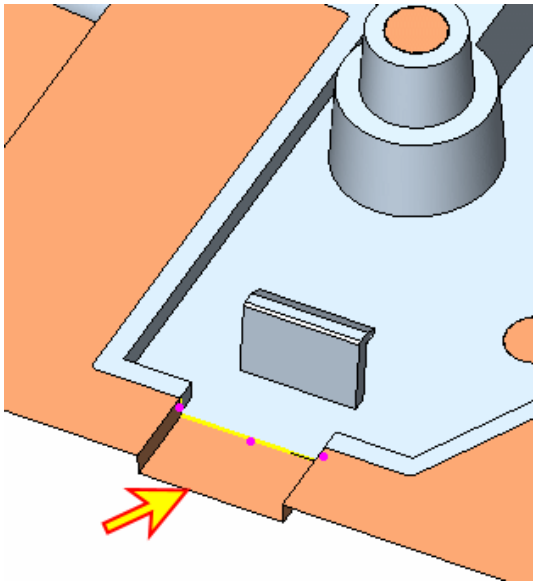
16. Add Extrude the sketch **To Reference**. Select the face indicated by the arrow below as the **Reference Face**:

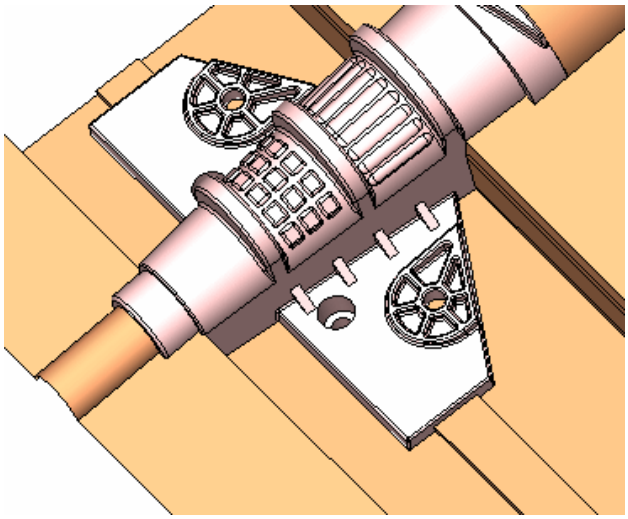


17. Select the reference face indicated by the arrow below as the **Active Object**.

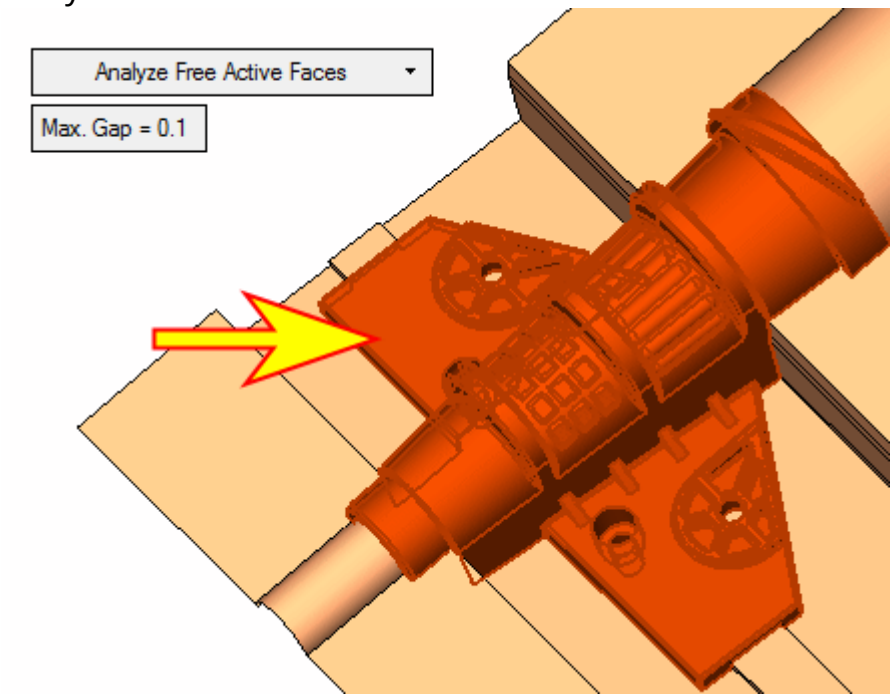


18. Repeat the same operations in the other free active faces side:

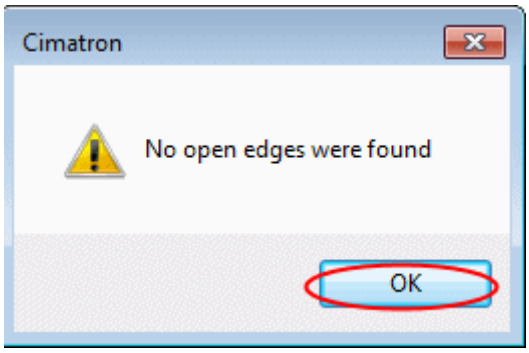




19. Delete al marks of treated areas. Re-perform the free active faces analysis.

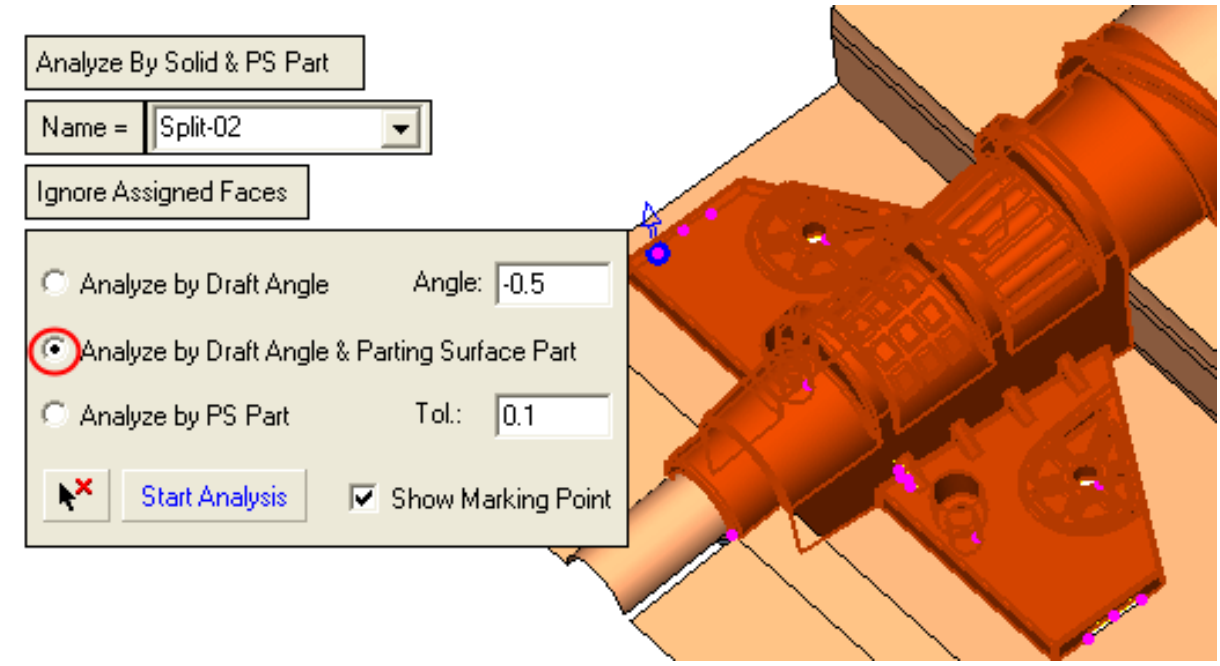


20. No open edges were found.

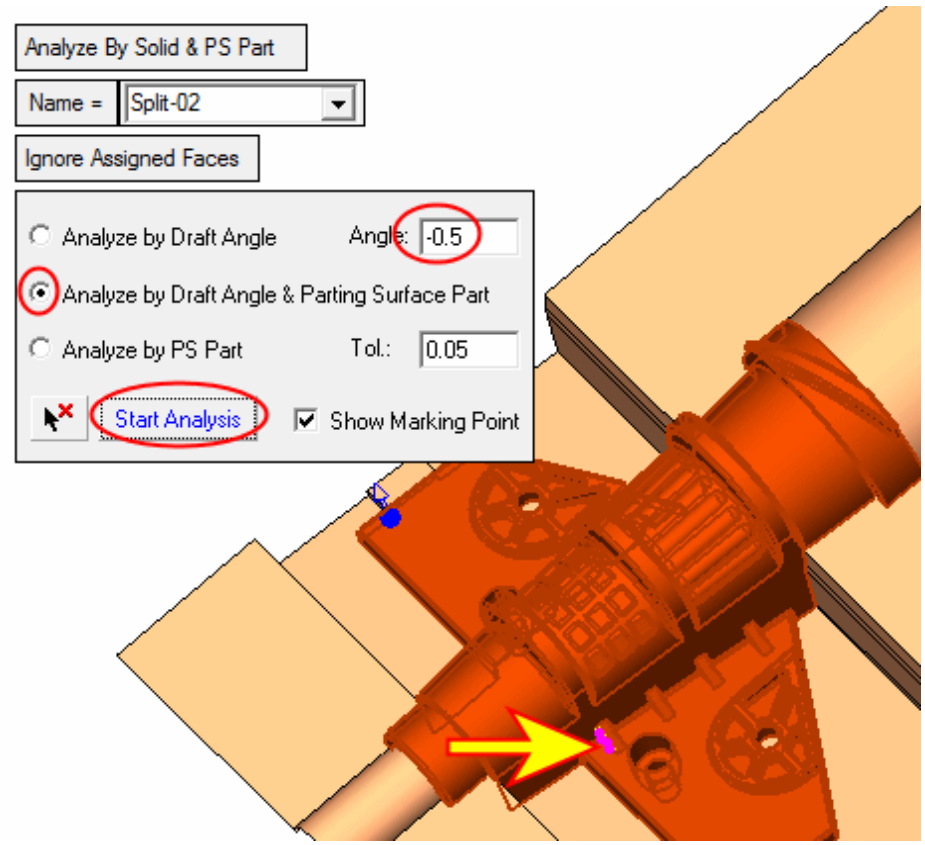


Note: The **Free Active Faces** analysis is almost similar to the **Edit Direction – Analysis By Solid & PS Part – Analyze by Draft Angle & Parting Surface Part** option. It is the same analysis algorithm; however, when performing the analysis through the edit mode, the system looks also for undercuts and can expand the split direction set to other unassigned faces while the **Free active faces** analysis checks only the selected split direction faces (does not expand any farther) and does not look for undercuts.

The advantage of using the **Free Active Faces** analysis is that it is an external fast analysis (it does not require entering the edit direction mode).

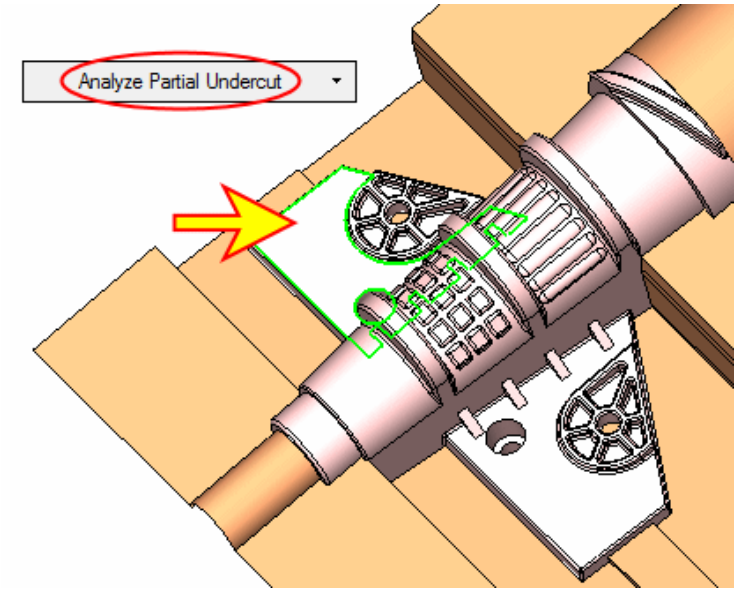


Note: The undercuts detected in the **Edit Direction analysis** (indicated by the arrow below) are due to using **Analysis by Draft** angle in addition to **Analysis by Parting Surface Part**. You may also detect these undercuts using the *external* **Analyze Partial Undercut** tool (detailed in the next step).

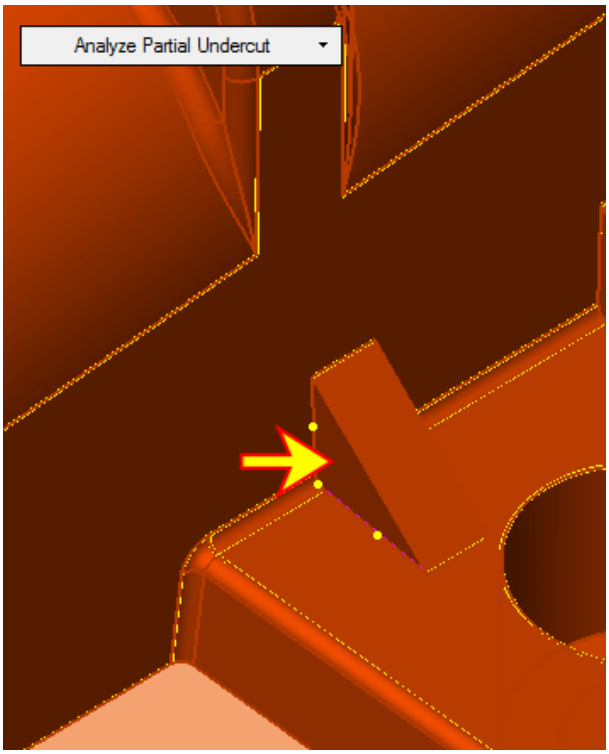


Analysis Tools – Parting Analytic Tools – Analyze Partial Undercut

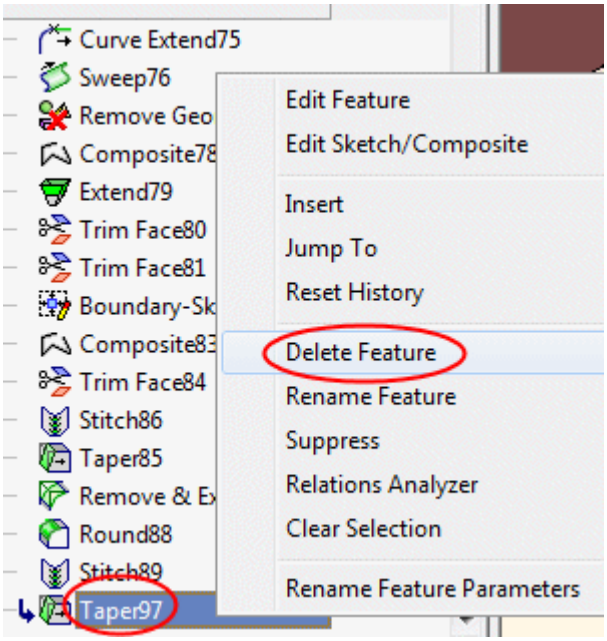
1. Enter the **Parting Analysis** tool. Choose the **Analyze Partial Undercut** option and select a face of the core split direction as indicated by the arrow below:



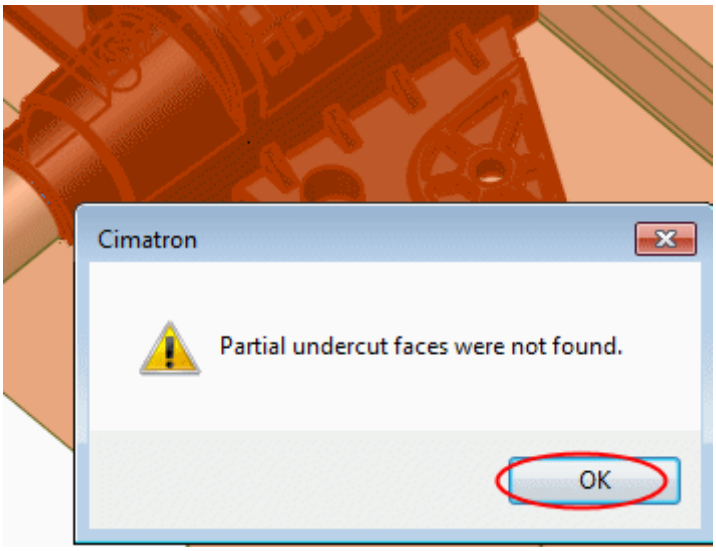
2. The system detects an undercut (highlighted in **YELLOW** dots).



3. Fix the undercut by deleting the Taper feature in the work part (last feature).



4. Recheck the part for undercuts. No undercut faces were found.



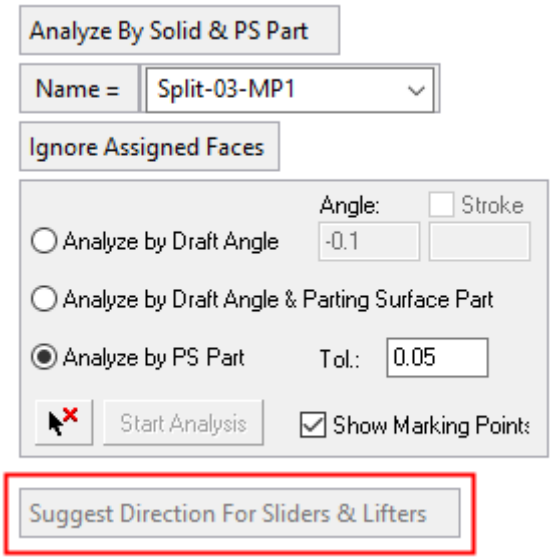
Find the Best Extraction Orientation for Sliders and Lifters

Description

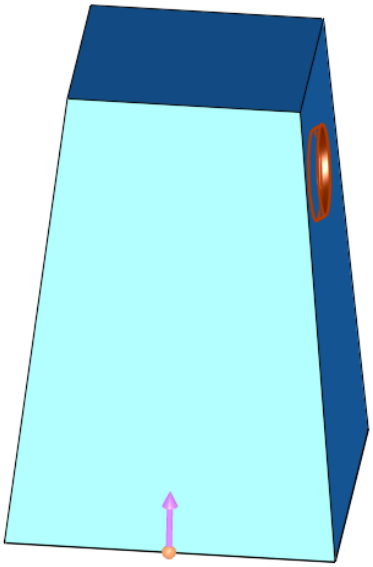
An option in the Quick Split tool allows finding the best opening direction of selected Slider and Lifter faces. This considers all selected faces and finds the direction with the least number of undercuts. This is especially useful when all the faces of a slider or lifter have draft angles and the opening direction is very hard to define for them.

Exercise

1. Load the part, **Suggest Direction 2.elt**.
2. Enter **Parting > Quick Split**.
3. Click **New Direction**.
4. The option, **Suggest Direction For Sliders & Lifters**, is displayed with the parameters. Set the parameters as shown below:

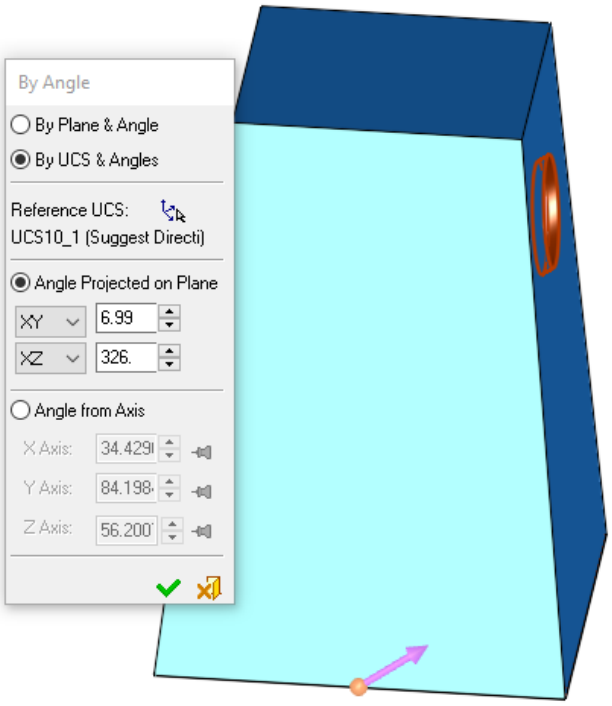


5. Select the faces as shown below:



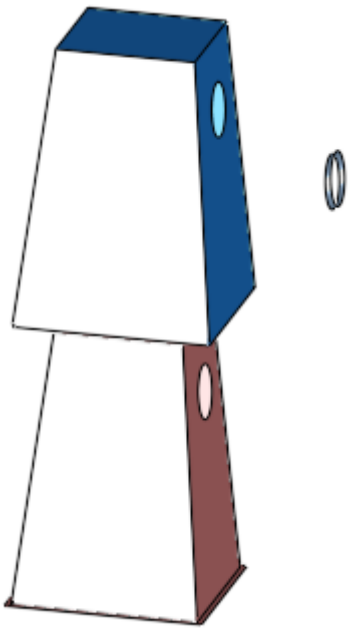
The option becomes available, **Suggest Direction For Sliders & Lifters**.

6. Click this option. This opens the **By Angle** tool of the direction area allowing the user to fine tune the angle and round it.



7. Set **XY** angle to **7** and **XZ** to **325**.

8. Select **OK**.



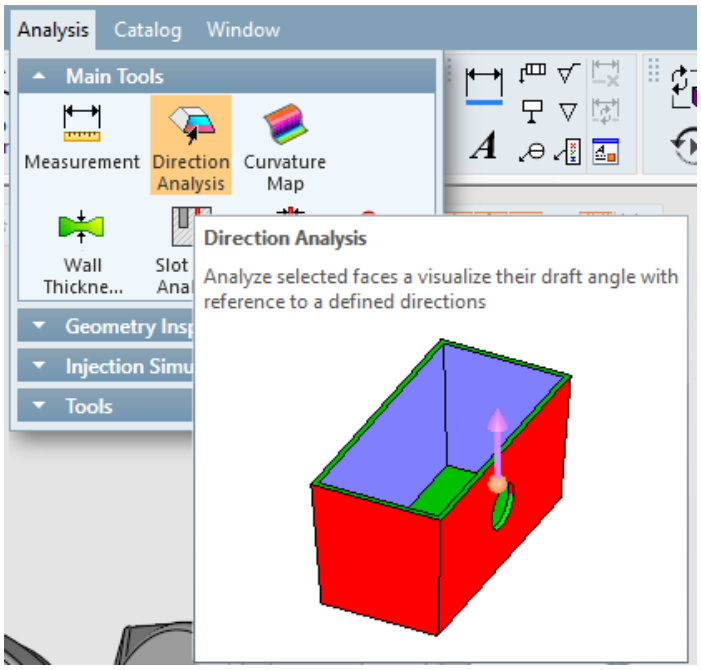
Direction Analysis Improvement

Users get more information from the **Direction Analysis** tool: two main directions and all the areas that are undercuts, as well as the ability to set part orientation for complex parts.

Exercise

1. Load the part, **Direction Analysis.elt**.

2. Select the **Direction Analysis** tool.



3. Select the object and **Exit** via the **Middle Mouse Button**.

   The direction analysis tool has two Opening directions modes:

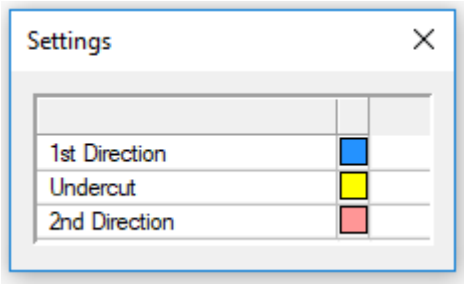
   Old (the existing one) – 1 opening direction.
   New – 2 opening directions.

Note: The 2 Opening Directions mode is not available in an NC Mode in an NC document. The button will be set to 1 and grayed out. The default in all CAD environments except for a Die project is 2. This mode is available for users with a QSPLIT license.

4. Set the opening direction **along Y** (using the **Direction Arrow**) or select **Suggest Direction** and set 2 Opening Directions.

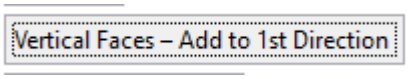
   This mode divides the part into 3 distinct colors:

   - Selected Direction
   - Opposite Direction
   - Under Cut

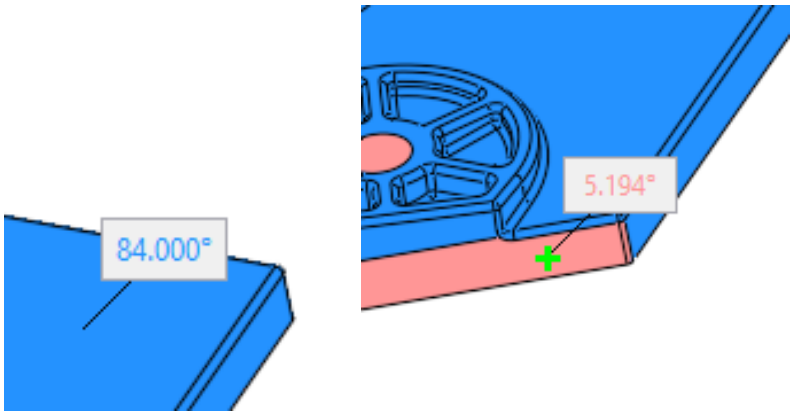


   * Vertical faces go with one of the 2 first directions. The displayed color changes are only temporary; once you exit the function, the original colors of the objects are displayed. Vertical faces may be assigned to either the 1st or 2nd opening directions.

5. Set **Vertical faces – Add to 1st Direction**.



   Notice that the text color changes to that of the direction:



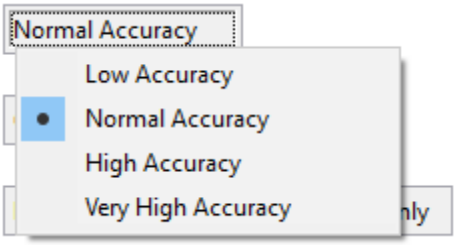
6. **Accuracy**: A dropdown screen parameter (SP) with the following options:

   - **Low Accuracy**
   - **Normal Accuracy**
   - **High Accuracy**
   - **Very High Accuracy**

   The corresponding faceting tolerances are:

   - **0.5**
   - **0.1**
   - **0.02**
   - **0.01**

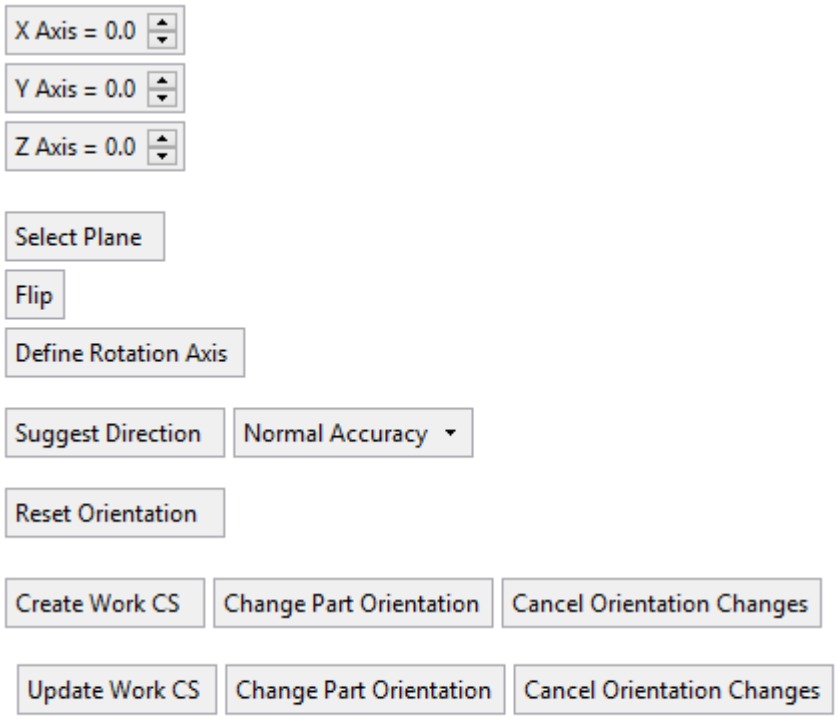
   The default is **Normal Accuracy**.



7. **Orientate Part**: This option allows the user to find the correct orientation in the mold. The prompt says: “Rotate part to review opening direction (on Z) and undercuts, and <exit>.” After any rotation in any method, the analysis updates.

   Note: Rotate the part with just the **Left Mouse Button**.

8. Select **Orientate Part**. New screen parameters appear:



More Options:

Define Rotation Axis: Pick 2 points, a linear edge or curve or a circular face, edge, or curve. It is possible to define the axis by any of the inputs mentioned. When in this mode, if selecting a valid input, the model will only rotate around the defined axis. Clicking the **Esc** key or **Middle Mouse Button** leaves the mode without any change.

Suggest Direction: Finds the minimal undercut solution.

9. Select **Suggest Direction** with normal accuracy and examine the result.

10. Select **Reset Orientation** to go back to the previous orientation.

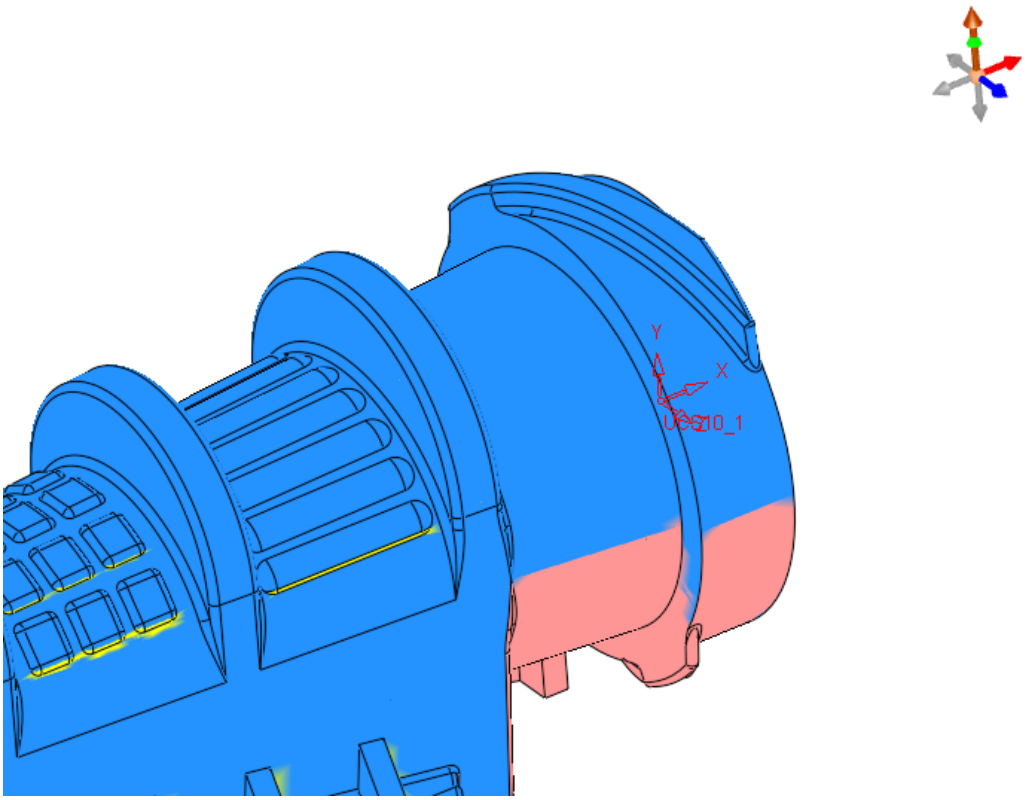
Reset Orientation: Goes back to the orientation that was there when orientation mode was invoked.

Create/Update Work CS: This is not a toggle screen parameter. The button will show one of these options.

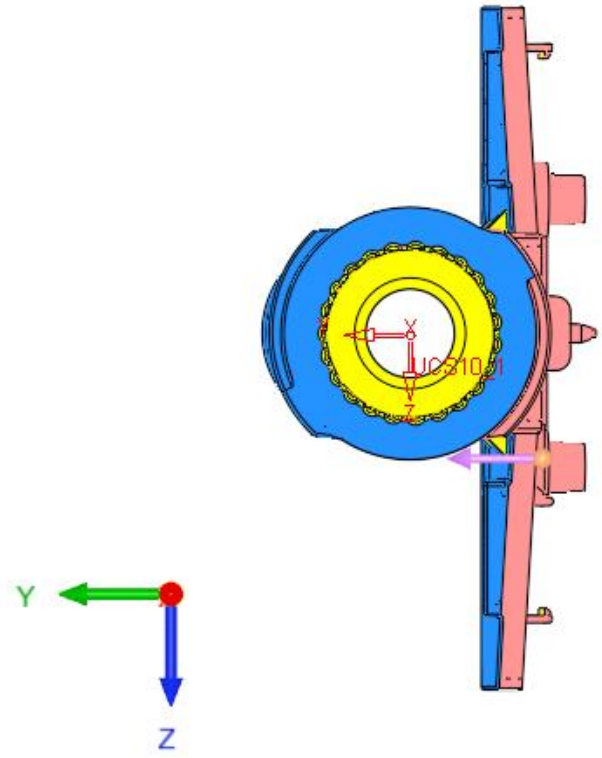
If a CS already exists in the part, we will show **Update Work CS**. If it doesn't, we will show **Create Work CS**.

Clicking this button creates a Work CS feature in the direction of the reposition but when we exit this mode the part will go back to its original position. If a work CS already exists, we change its orientation accordingly.

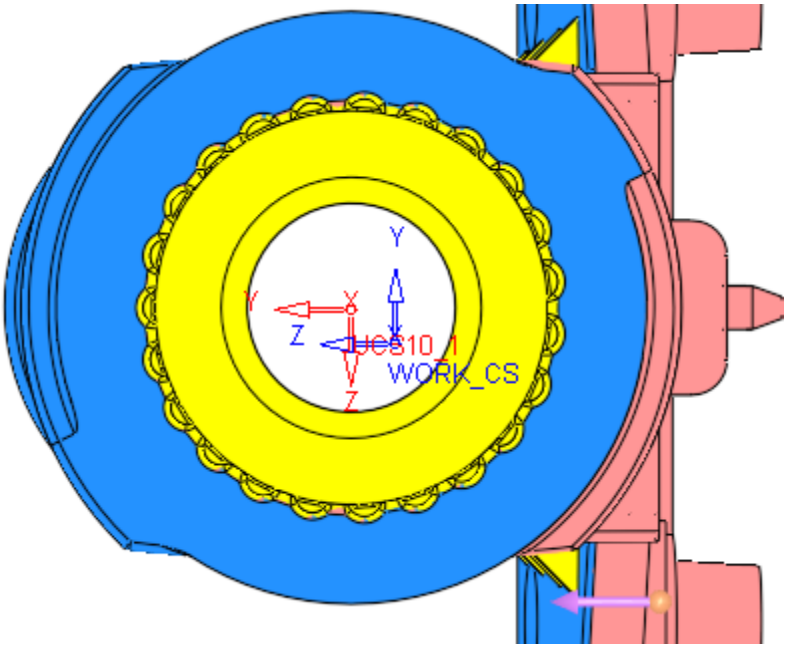
11. Set the opening direction **along Y**. **Exit** to go back to the previous dialog.



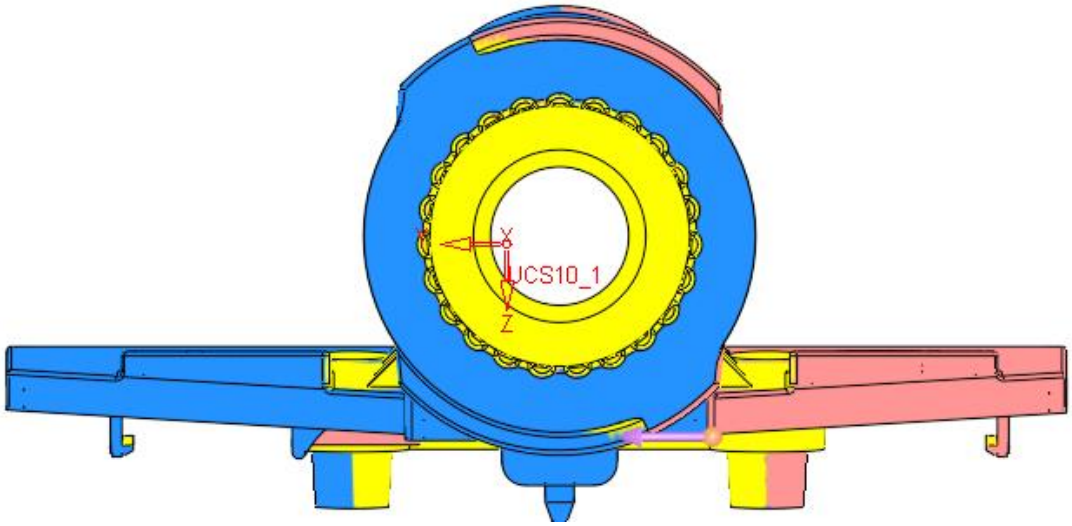
12. Rotate the part to the orientation shown below:



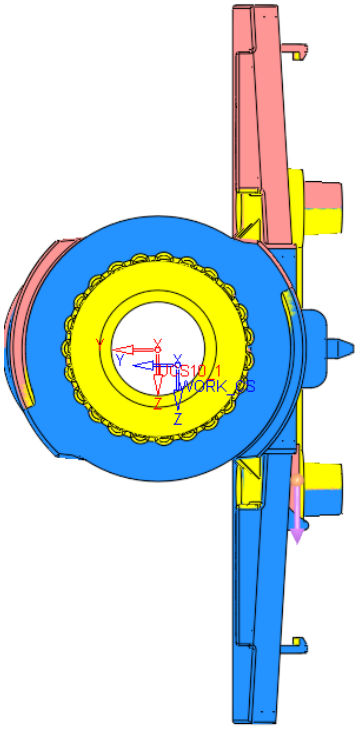
13. Select **Orientate Part**. If we create a UCS in this case, the Z direction will be along Y. **Don't create the UCS!**



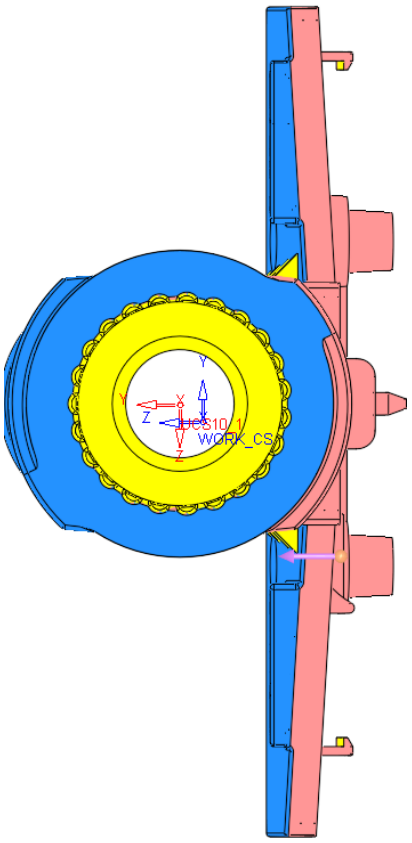
14. Rotate the part by setting **X axis** = **-90**.



15. Select **Create Work CS**. See the part return to its original position.

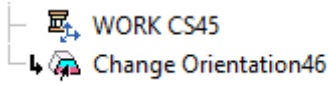


16. Select **Orientate Part** again, rotate the part **X** =**90** and select **Update Work CS**. Now the result Z is along Y.



Change Part Orientation: This will move all the entities in their parts to fit the new position as we exit out of orientation mode.

17. Enter **Orientate Part**, rotate the part, and select **Change Part Orientation**. See a new feature is created in the tree:



This feature can't be edited.

Cancel Orientation Changes: Reset orientation and exit out of orientation mode

Preferences

The Preferences for **Modeling > General**: