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Solids Import Tutorials

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SOLIDS IMPORT TUTORIALS

About the Tutorials

These tutorials provide instruction for extracting geometry from solids and sheet models in preparation for machining on a basic mill or lathe without solid machining capabilities. Users should be familiar with creating geometry in GibbsCAM. For more information on basic concepts, these are discussed in more detail in the Advanced Milling User guide. These exercises do not contain any machining instructions.

EXERCISE 1: GEAR.X_T

This exercise introduces procedures for extracting geometry.

Opening the Part

1. In the Open dialog choose Parasolid (*.x_t, *.xmt) from the Files of Type menu.
2. Open the file Gear.x_t.

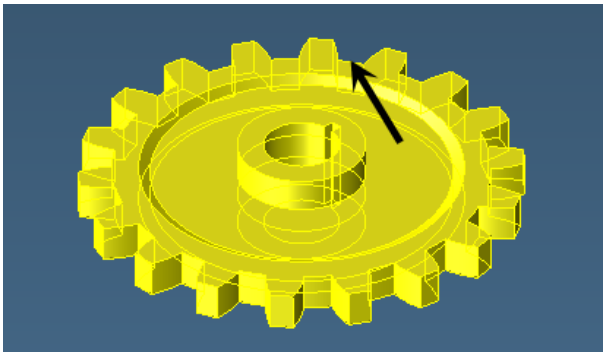
   This file is located in the part files of the installation CD in the solid tutorial files section.

Extracting the Geometry

Extracting Edges

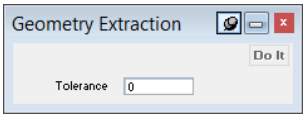
1. Turn on Edge Selection.

2. Double-click the edge shown to select the entire contour.



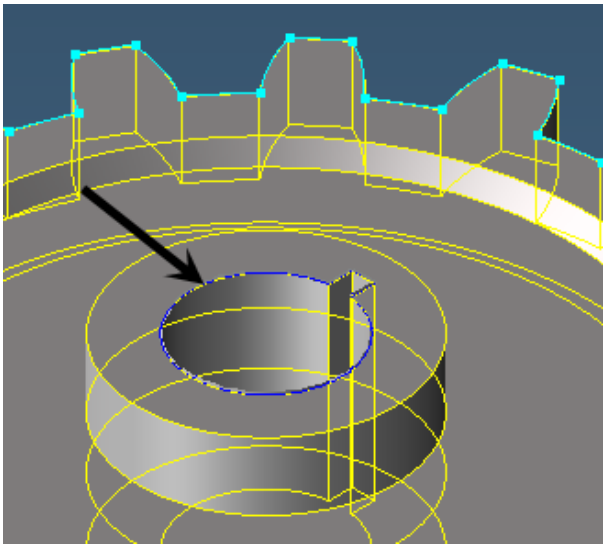
3. Right-click.
4. Select Extract Edges.

5. Extract the geometry.

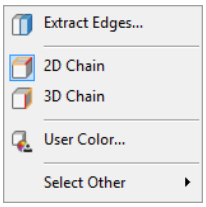


We will now extract the geometry for the keyhole slot.

6. Zoom in to the area shown.



7. `Right-click` any edge and ensure that edge selection is in the 2D Chain mode.

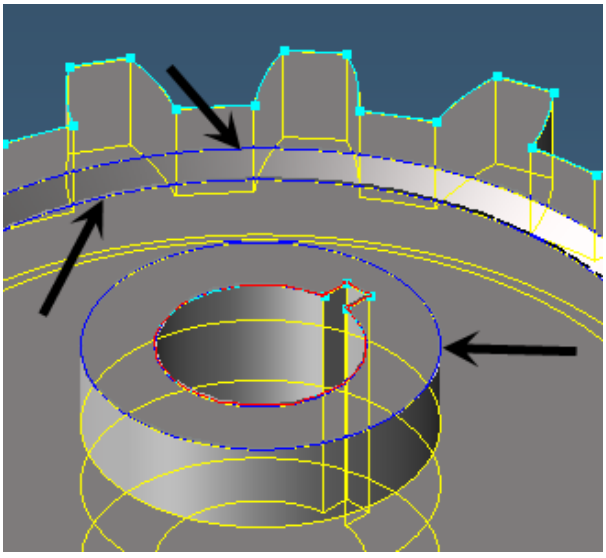


8. `Double-click` the edge indicated by the black arrow.

9. Extract the geometry.

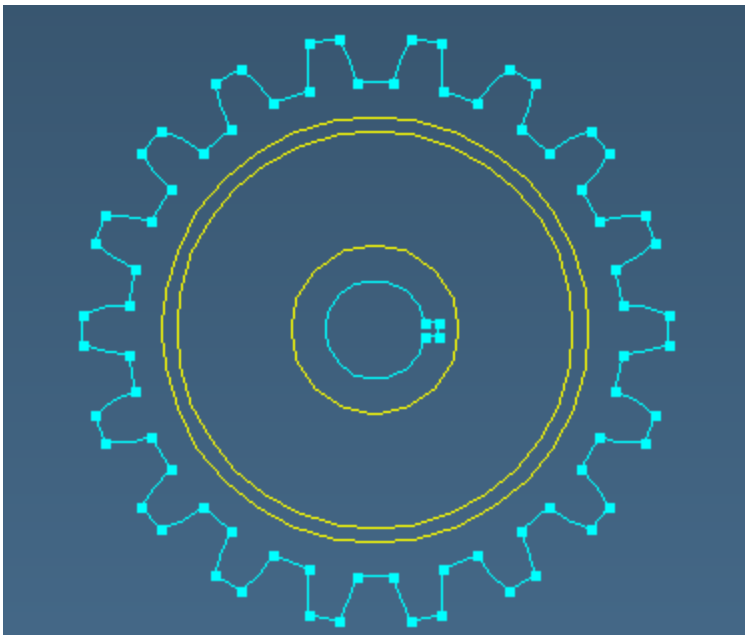
Our last step will be to extract the geometry we would need to define the recess of the gear.

10. (`Ctrl+click`) the three edges shown.



11. Extract the geometry.

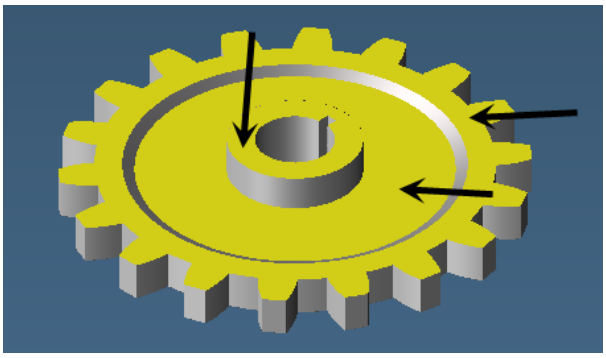
We now have enough geometry to machine. But let's take a look at an even faster method.



12. Delete all of the geometry.

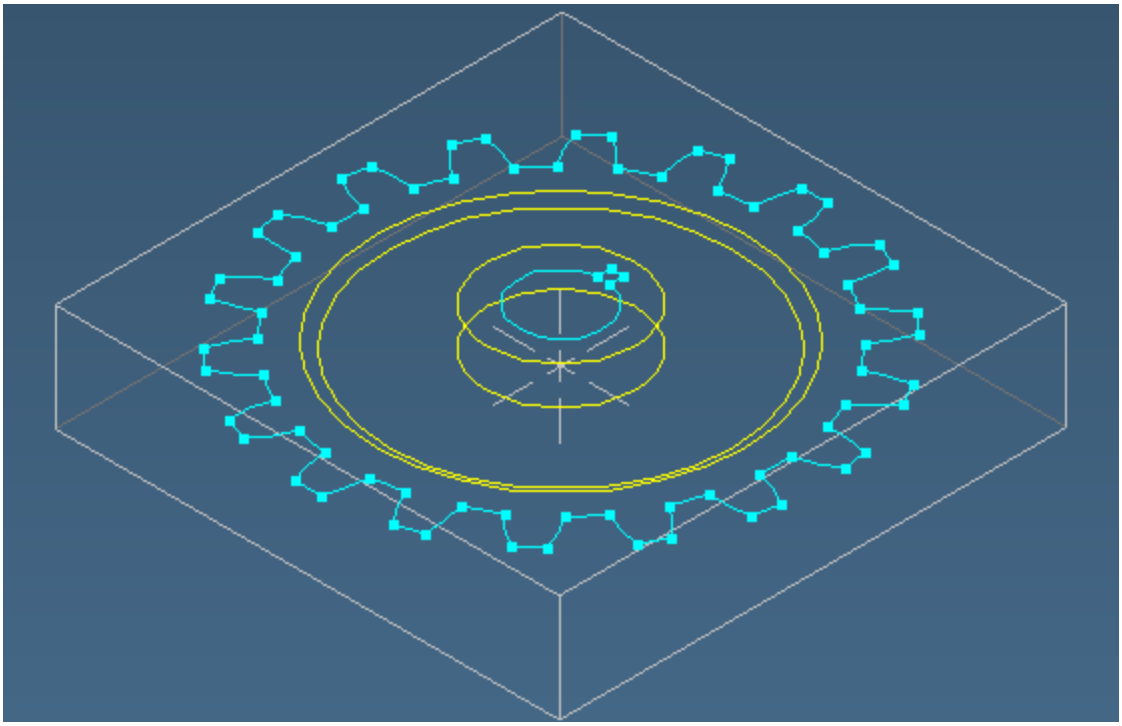
Extracting Face Boundaries

1. Turn off Edge Selection.
2. Turn on Face selection.
3. Select the faces shown.



4. Right-click and choose Extract edges.

   With this method you actually get more geometry. Face selections will extract the boundary edges of the face.



EXERCISE 2: FLANGE.X_T

This exercise will introduce the concept of slicing a solid using a coordinate system. Once the body is sliced we will extract geometry to create a lathe part.

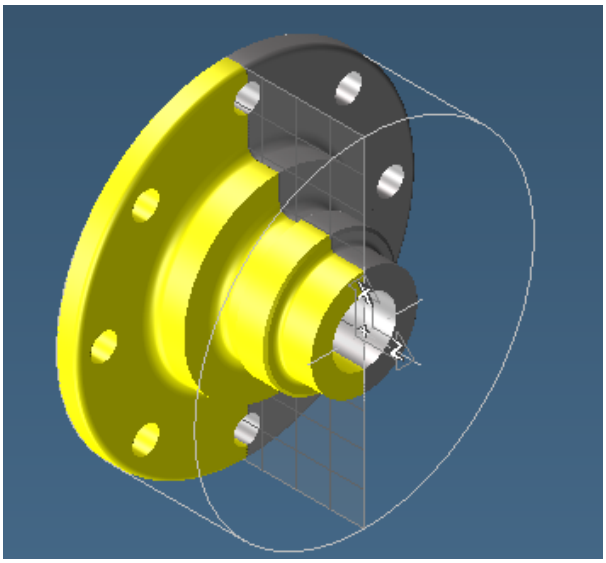
Opening the Part

1. In the Open dialog choose Parasolid (*.x_t, *.xmt) from the Files of Type menu.
2. Open the file flange.x_t.

   This file is located in the part files of the installation CD in the solid tutorial files section.

   The first thing we will do to this part is slice it to extract the geometry of the flange's profile. We will use the ZX plane as a slicing plane.

1. Open the Solid Modeling palette.



2. Click the Slice button.

   The part is now separated into two distinct bodies.

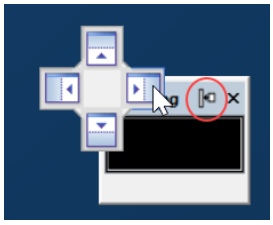
Extracting the Geometry

Placing a Body in the Body Bag

1. Click the Body Bag command.

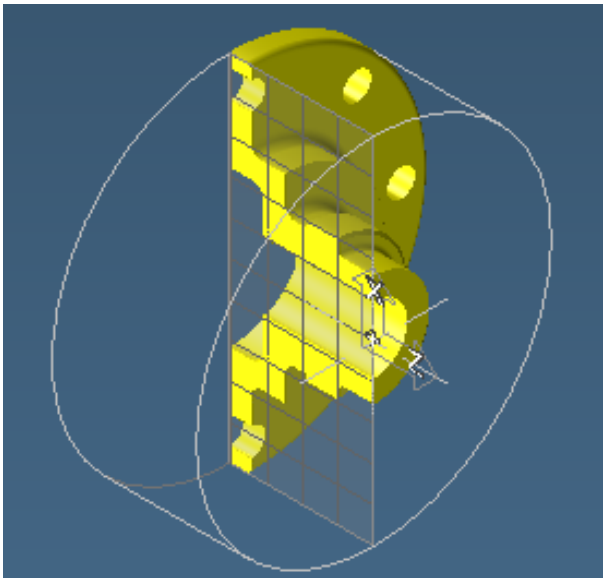
As we will be using this dialog extensively, we will attach the Body Bag dialog to the right side of your screen.

2. Click the docking icon on the body bag dialog (circled below). Now click and drag the body bag title bar and place the arrow on the grid selection as shown below. The dialog will snap to the outside right of the Gibbs display, leaving the workspace clear.



3. From the isometric view, `Double-click` the selected half in Y+ side. This places the front half in the Body bag as shown

We now have a clear view of the part's profile. Note that the body is still selected.

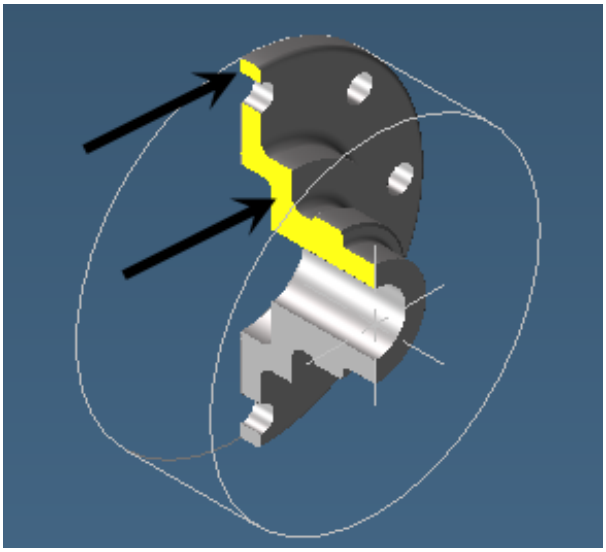


We will now extract geometry from the exposed profile. You may find it helpful to turn off the Coordinate System grid for this.

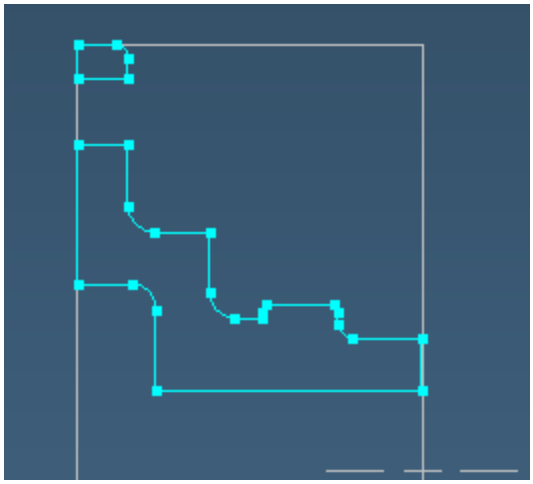
4. Turn on Face selection.

Use the `Ctrl` key while selecting multiple faces.

5. Extract the geometry from the faces shown.



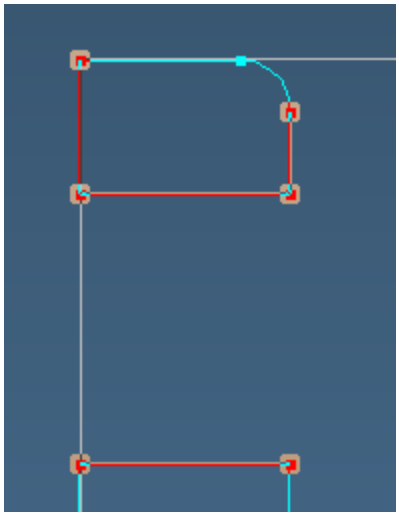
Connecting Geometry



The profile geometry is in multiple pieces. For this part to be machined we will need to fix this.

1. Place the other half of the body in the Body Bag. (`Double-click`)

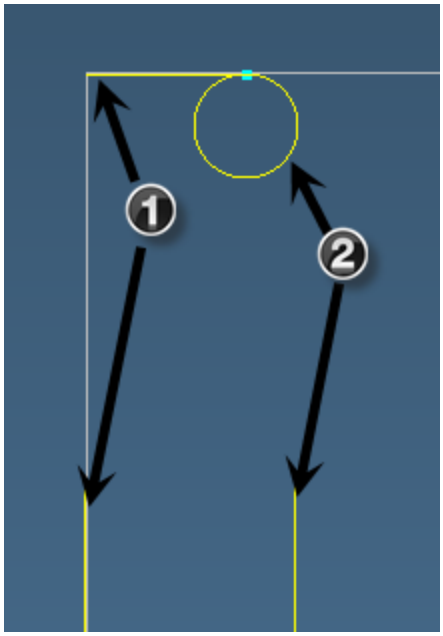
2. Delete the points and lines shown.



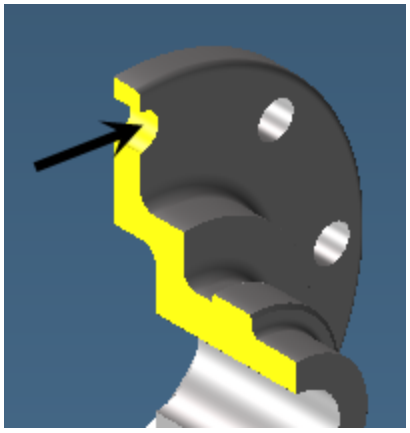
We will now connect the geometry.

3. Connect the lines shown (1).

4. Connect the line and the circle shown (2).



We now have a fully connected profile of the flange that is ready to be machined.



An alternate method for this extraction would be to select the hole's interior face as well, extracting all geometry. This would create splines that could be forced to a depth of Y0, and then changed to fall in the XYZ rather than HVD. However, using splines for the geometry may be inappropriate for some uses.

EXERCISE 3: ANC101.X_T

This exercise is designed to give you an understanding of creating coordinate systems from solids. In addition to creating the coordinate systems they will be modified to be of greater use. You should be familiar with the Advanced Milling module and its exercises before starting this exercise.

Extracting Geometry to CS

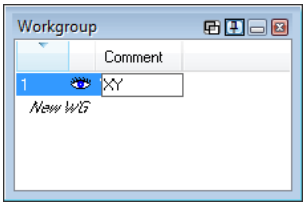
XY Hole Extraction

1. Locate and open the file named anc101.x_t

   The first thing we will do is extract the geometry that is in the XY plane. As there will be geometry in several different coordinates systems, the screen can get cluttered very quickly. Workgroups can help us keep things organized. We will need to be sure to rename each workgroup we create to reflect the geometry that will be in it.

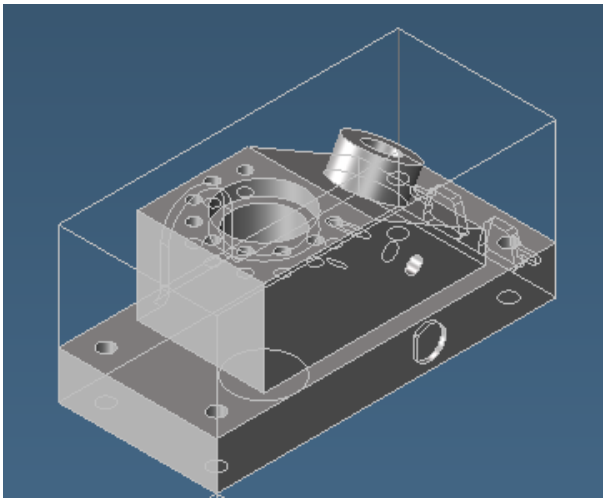
2. Open the Workgroup list.

3. Name Workgroup 1 XY.

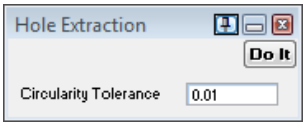


4. Open the Hole Extraction dialog located in the Geometry creation palette.

5. Select the model.



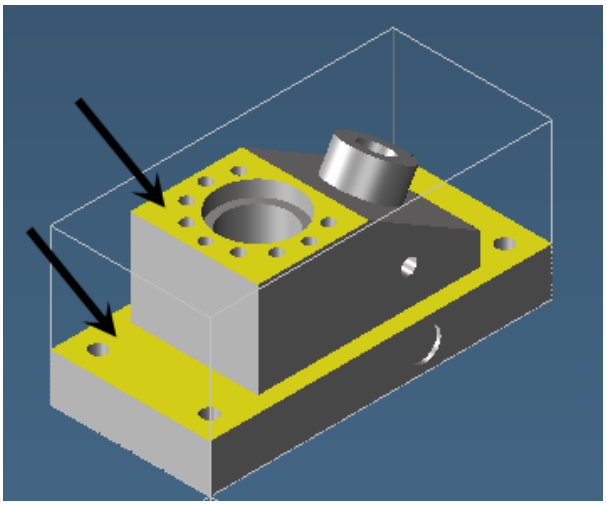
6. Extract the holes as shown.



Now we will continue to extract the XY geometry. Be sure that the Show Edges button is depressed.

7. Turn on Face selection.

8. Extract the geometry from the faces shown.



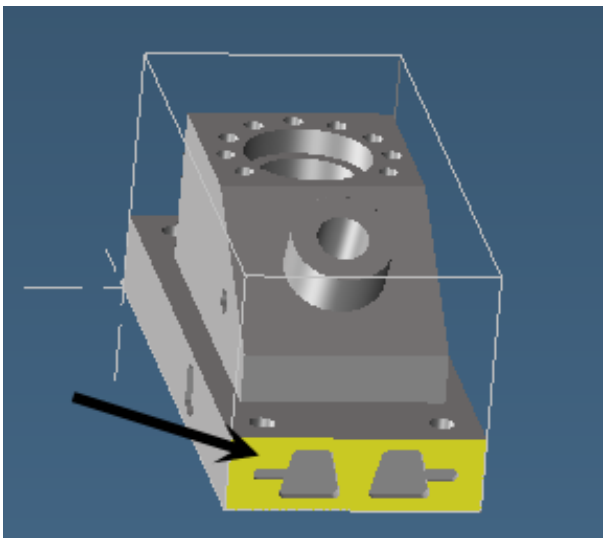
Edges could have been used to extract the geometry also.

The YZ plane

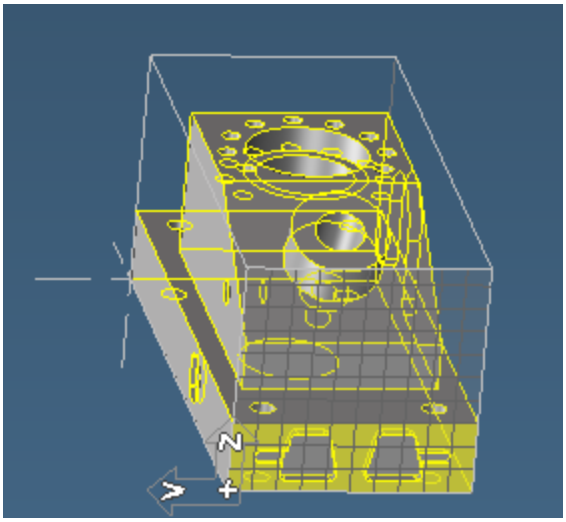
1. Create a new workgroup and YZ coordinate system.

You may wish to label these.

2. Select the face shown.



3. Right-click the Align CS Plane button and choose Align Plane Through & Move (Alt).



We need to rotate it into place.

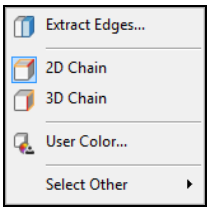
4. With the Face selected click the Align CS button 3 times.

Each click rotates the CS 90 degrees counter clockwise.

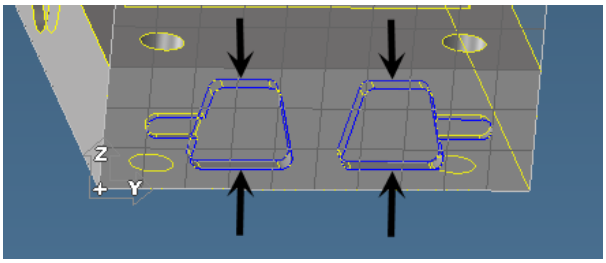
The selected face acts as a normal vector to rotate the CS.

Compound Chain Selections

1. Right-click any edge and ensure that edge selection is in the 2D Chain mode.



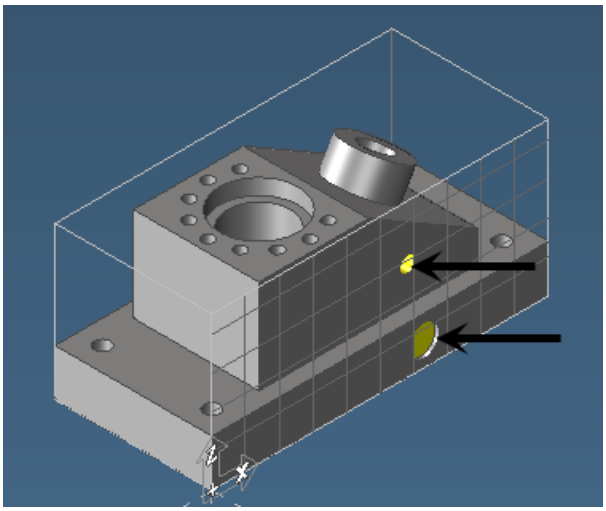
2. (Ctrl+Double-click) the edges shown.



3. Extract the geometry for these edges.

The XZ Plane

1. Switch to the XY CS.
2. Create a new workgroup and XZ CS.
3. Extract the geometry for the hole edge and pocket face shown.

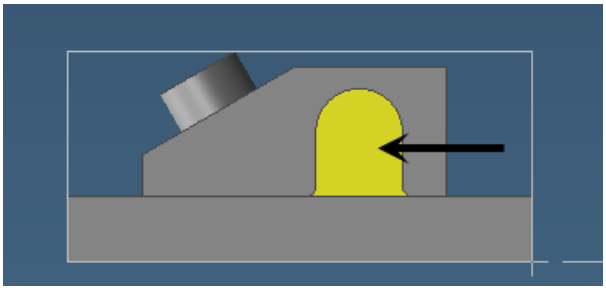


Both the edge of the hole and the pocket floor to easily obtain cut depths.

4. From XZ CS create a new workgroup and XZ CS.

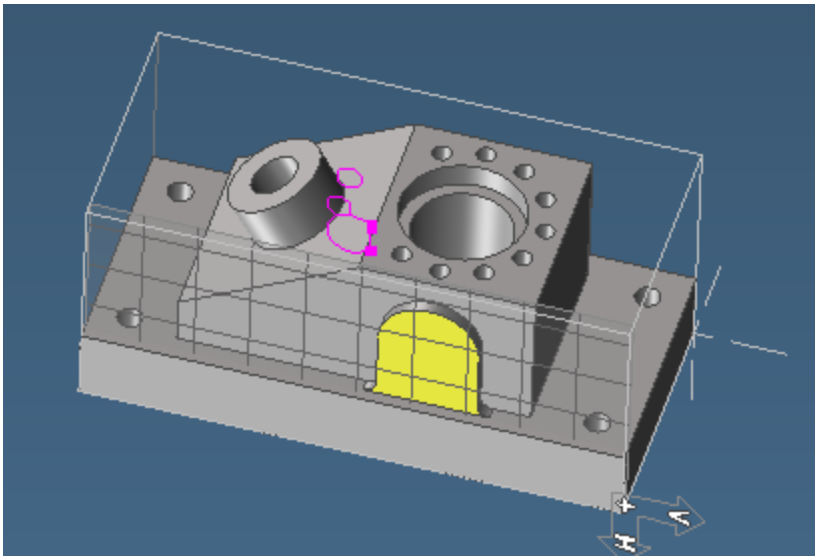
   This will be the backside XZ CS.

5. Switch to the alternate front view (`Ctrl+Alt+F`) and select the face shown.



6. `Right-click` the Align CS Plane button and choose Align Plane Through & Move (Alt).
7. `Click` the Toggle Depth button.

8. `Click` the Align CS button.



This will rotate the CS to a ZH position.

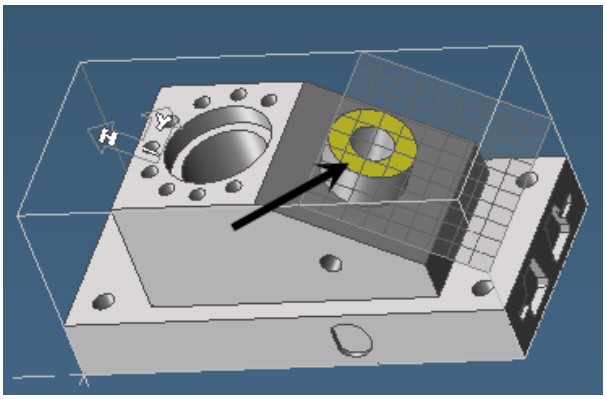
9. Extract the geometry for this face.

We will now align a CS to the face of the cylinder and extract the geometry.

The Angled Hole

1. From CS1 XY create a new Workgroup and CS.

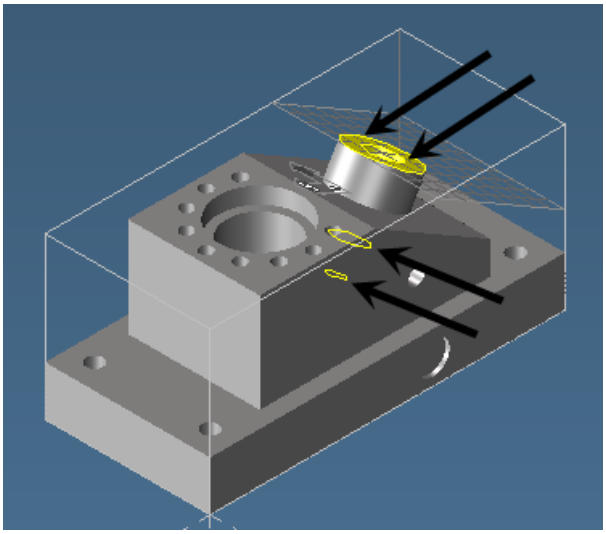
2. Select the cylinder face as shown.



3. `Right-click` the Align CS Plane button and choose Align Plane Through & Move (Alt).

4. `Click` the Toggle Depth button.

5. Extract the geometry shown.

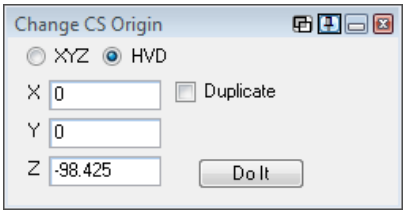


The geometry may be extracted with a combination of edge and face boundary extraction on the top of the cylinder.

Slicing the Part

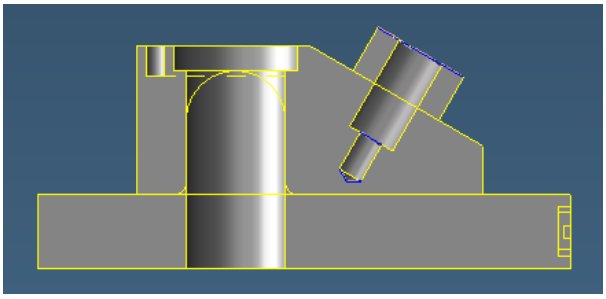
1. From CS3 XZ create a new CS.

2. Change the CS Origin as shown.



3. Open the Solid Modeling palette.

4. Slice the Model with the CS.



5. Place the selected slice in the Body Bag.

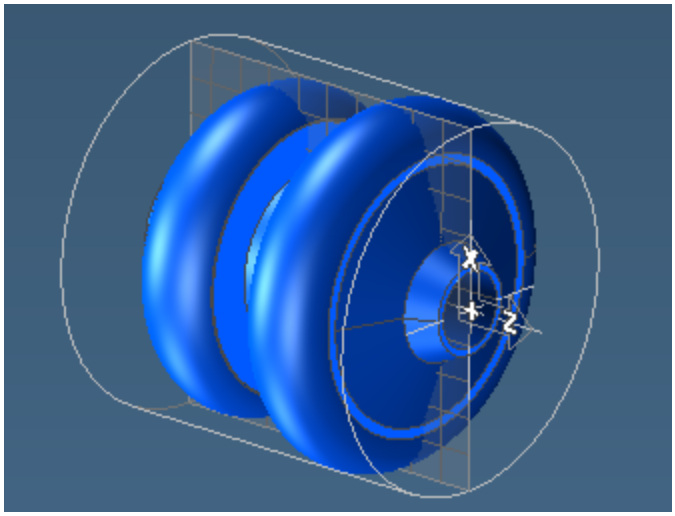
Note that at the bottom of the angled cylinder is a modeled drill tip. The lower circle indicates the depth the full diameter of a drill may go.

EXERCISE 4: REVOLVE.IGS

This exercise is designed to give you some experience in extracting geometry from sheets. The functionality of the Solids Import module for solids and sheets is identical. Sheets however can be more challenging to work with as sheets have no depth, from a certain angle you may not see a sheet. Additionally, there can easily be hundreds of sheets for what would be a single solid, making your workspace cluttered.

If you do not have a lathe MDD you can still do this exercise. The primary difference will be that you will be working in the XY plane as opposed to the ZX plane.

1. Open the Revolve.igs file.



   Ensure the Surfaces option is selected in the Features list.

   Although this part can be solidified and geometry easily extracted from a sliced face, we will assume the file can't be solidified.

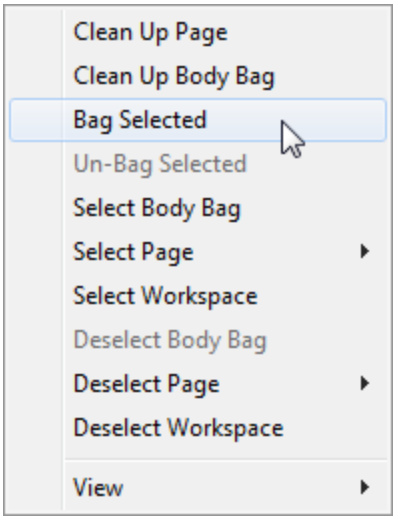
2. Choose Edit > Select > Sheets.

   Select All (Ctrl+A) will also work.

3. Slice the sheets with the CS.

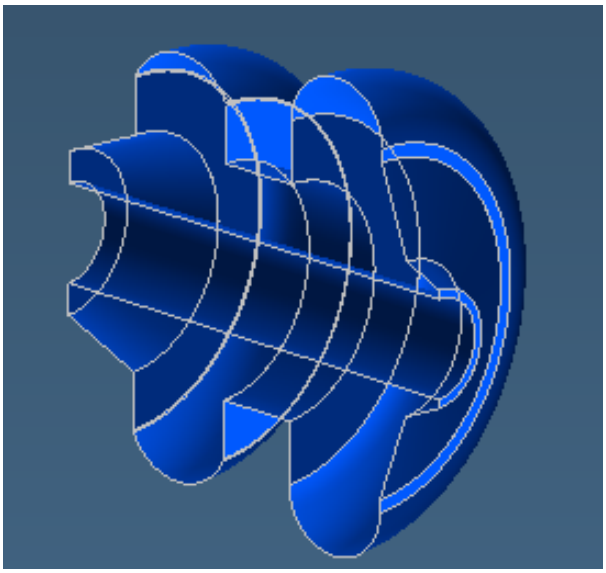
4. Open the Body Bag and Right-click the title bar and choose Bag Selected from the context menu.

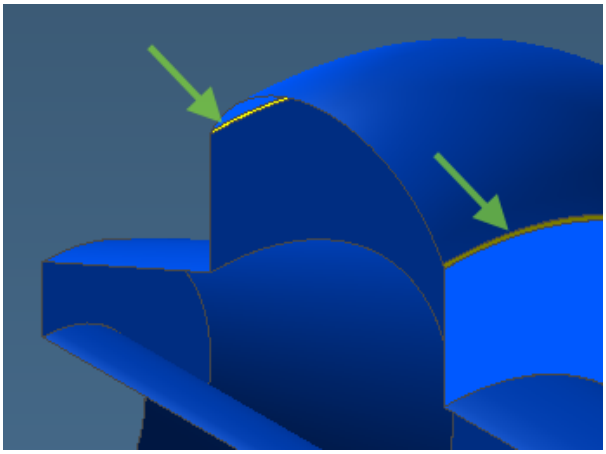
This action will send all bodies and sheets that are selected to the Body Bag. This can be very helpful when dealing with many items.



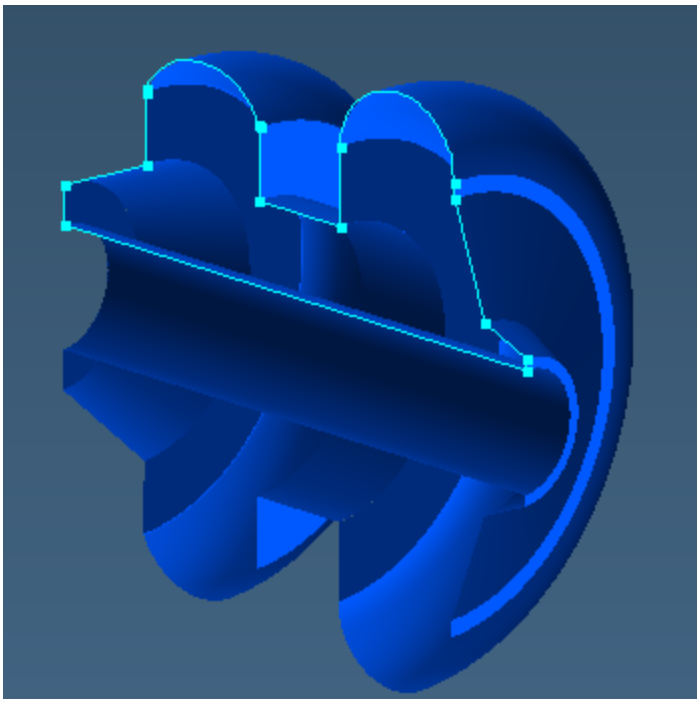
5. Turn on Show Edges so we can begin extracting geometry from these sheets.

It may be more difficult to see exactly what is being selected when working with sheets.





6. Extract the geometry that is along the sliced edges of the sheets.

   Note that the smaller “wheel” has a tiny fillet between the vertical wall and the arc. Be sure you select this as well.

When the geometry is extracted a single closed shape will appear.