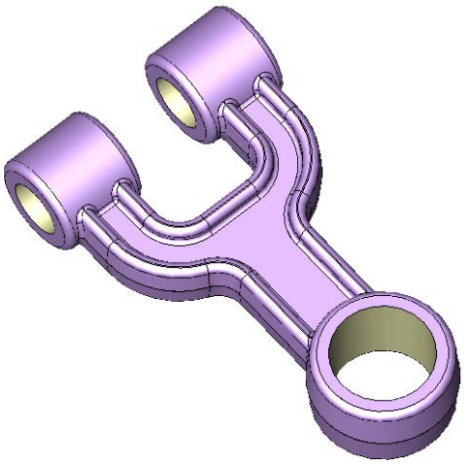
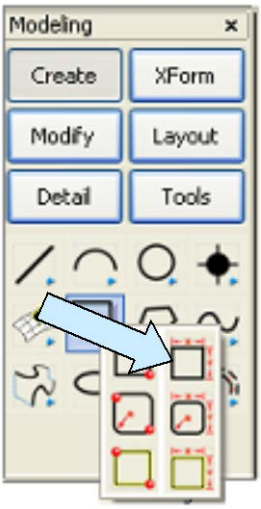
## KeyCreator Lesson KC2113

### Creating a Cast Wishbone Linkage



The cast wishbone linkage illustrated to the right is constructed with five degree draft from a midline symmetry plane. In this lesson we'll review an easy way to quickly create designs like this.

Let's start with a new file in View 1. (The Top View.)



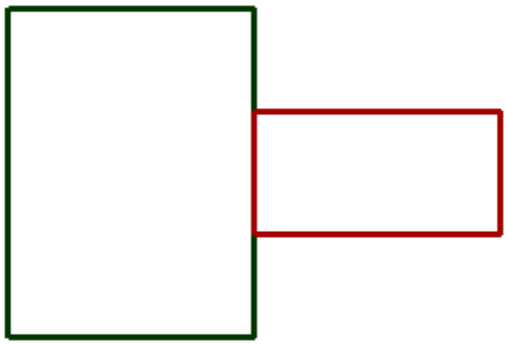

Click on the CREATE RECTANGLE BY WIDTH HEIGHT Icon.

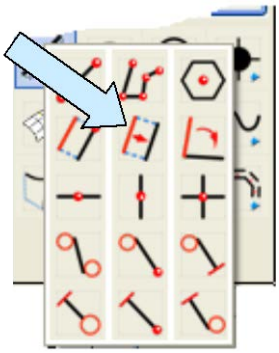
Use any anchor option. Type 3 for the Width and 4 for the Height. Using the Cursor Option, click anywhere on the screen to place the rectangle.

Next, create a second rectangle, this time using the MidLeft Anchor Option. Type 3 for the Width and 1.5 for the Height.

Using the CtrMid Option, click on the right side of the first rectangle.

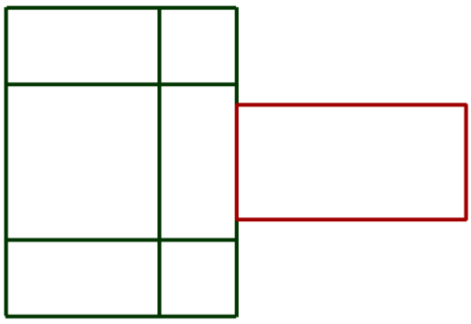
Your screen should look like this:





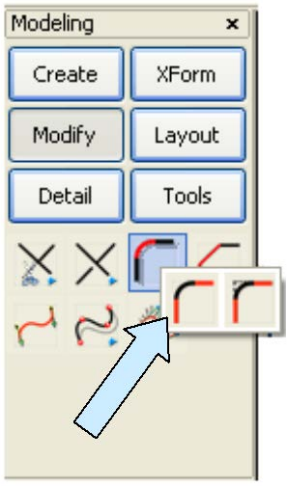
Click on the CREATE LINE PARALLEL AT A DISTANCE Icon. Type 1 for the Distance.

Click on the top of the first rectangle and then below it. Next, click on the bottom of the first rectangle and then above it. Finally, click on the right side of the first rectangle and to the left of it.



Your screen should look like this:

Delete the left side of the second rectangle.




Click on the FILLET WITH TRIM Icon.
Type 1.5 for the Radius.

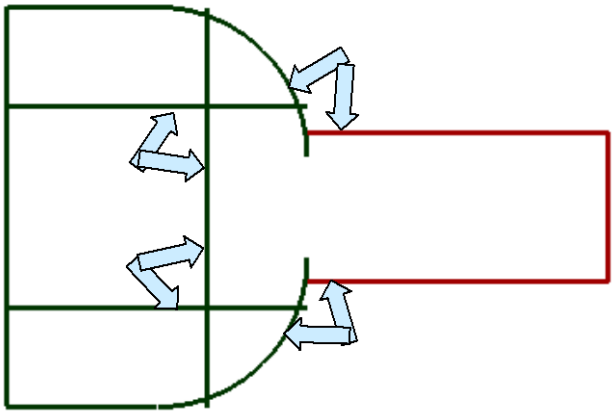
Click on the top edge of the first rectangle and then on the right side of the first rectangle.

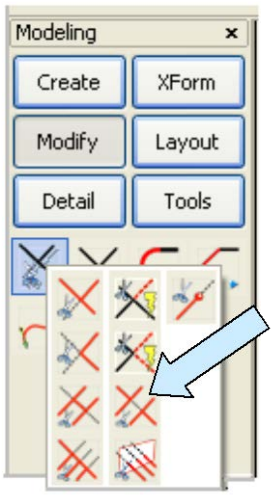
Next, click on the bottom edge of the first rectangle and then on the right side of the first rectangle.

Now, delete the remaining piece of the right side of the first rectangle.

Use the FILLET WITH TRIM Function again.
This time, type 0.5 for the Radius.

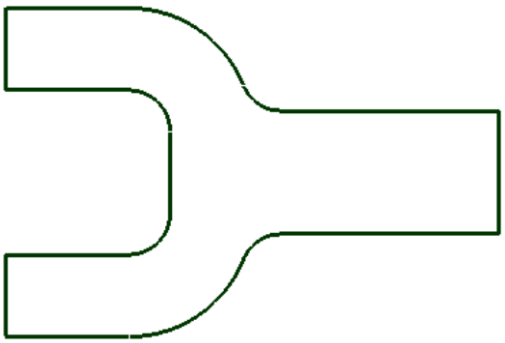
Create fillets at the four locations indicated in the illustration to the right.





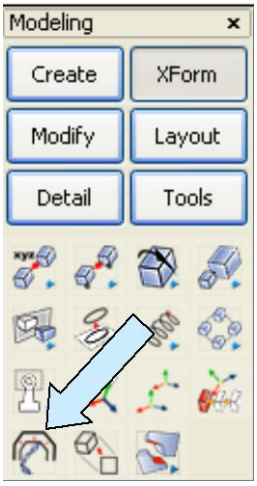

Click on the TRIM DIVIDE Icon.

Click on the left side of the what was the first rectangle and then on the second and third lines that intersect it.

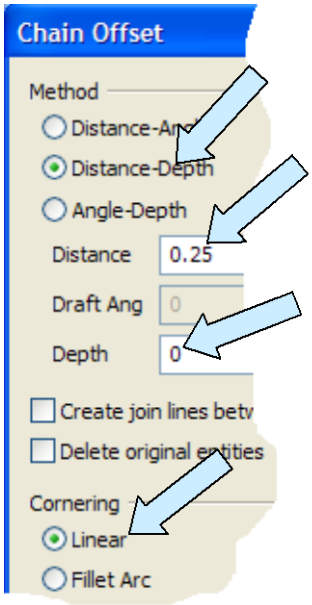


Your completed profile should look like this:

Select a new construction color.

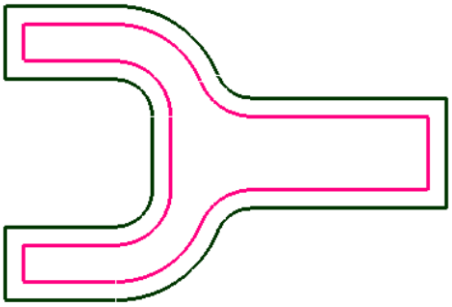



Now, click on the CHAIN OFFSET Icon.
A Dialog Box appears.

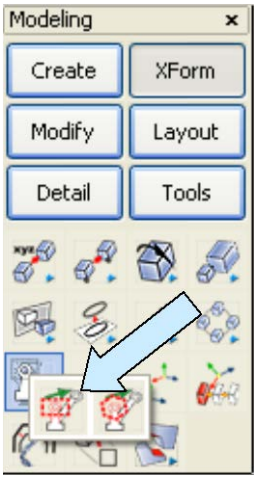



Use the Distance-Depth Option.
Type 0.25 for the Distance and 0 for the Depth.

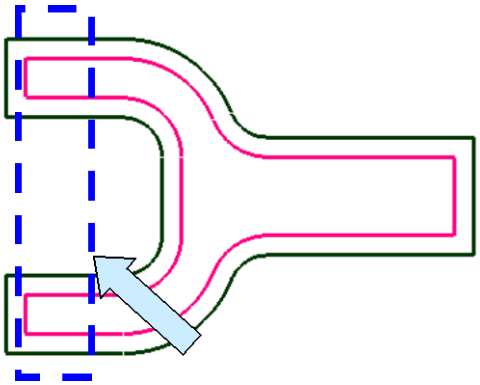
Use the Linear Cornering Option and hit the ENTER Key. (If you have the Enable Quick Chain and Restrict Chain Select to Cplane Options active in TOOLS/ OPTIONS/ SELECT, you can move your cursor over the profile on your screen and the entire profile will highlight. Click on it and then click on the inward-facing arrow.



Your screen should now look like this:




Now, click on the BOX MOVE Icon.

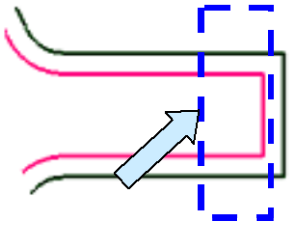


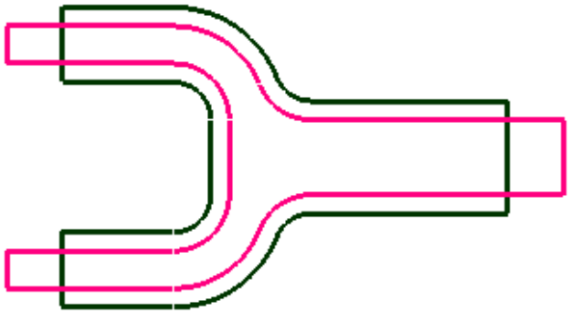
Drag a selection box around the two left ends of the smaller wishbone profile. (Do NOT include the ends of the larger profile!)

Select the XYZ Option on the Conversation Bar.
Type –1 for the X value and hit the ENTER Key three times.

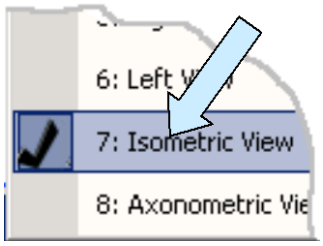
Now, drag a selection box around the right end of the smaller wishbone profile. (Do NOT include the ends of the larger profile!)

Select the XYZ Option on the Conversation Bar.
Type 1 for the X value and hit the ENTER Key three times.

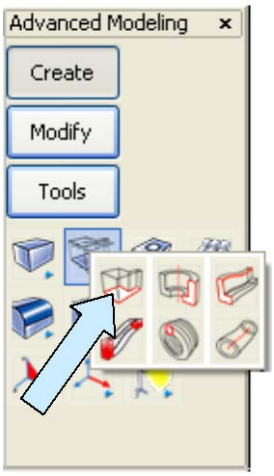




Your construction should now look like this:




Now, switch to the Isometric View. (View 7.)

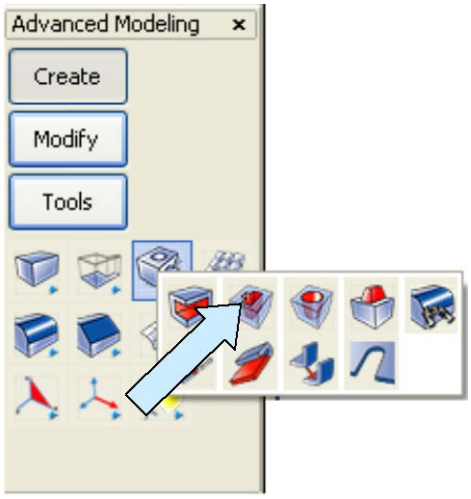



Click on the EXTRUDE Icon.

A Dialog Box appears. Type 0.5 for the Length, 5 for the Draft Angle, and select the Outward Draft Option.

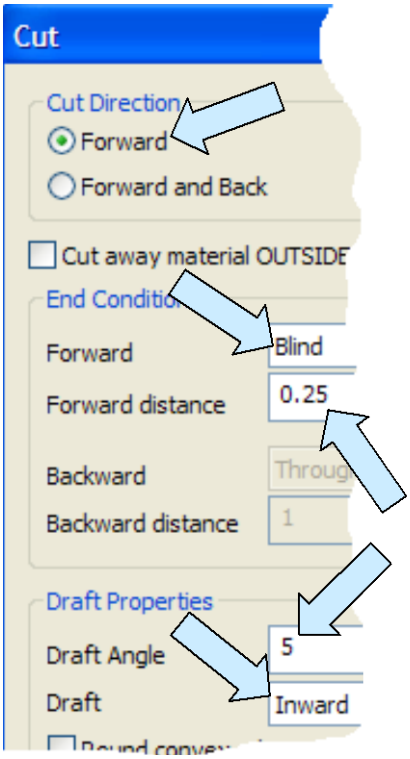
Select the larger profile and hit the ENTER Key.

Click on the downward-facing vector.




Next, click on the CUT Icon.

A Dialog Box appears.

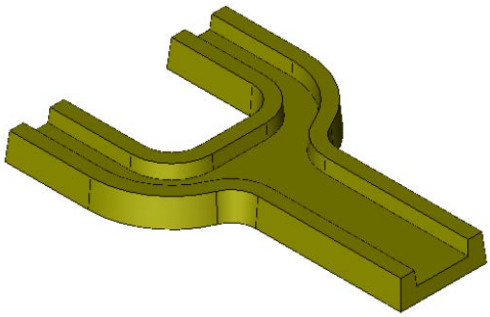



Select the Forward Option.
Use the Blind End Condition and type 0.25 for the Forward Distance.

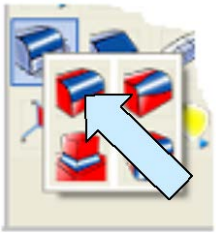
Type 5 for the Draft Angle and select the Inward Option. Hit the ENTER Key.

Select the solid. Then, select the smaller profile and hit the ENTER Key.

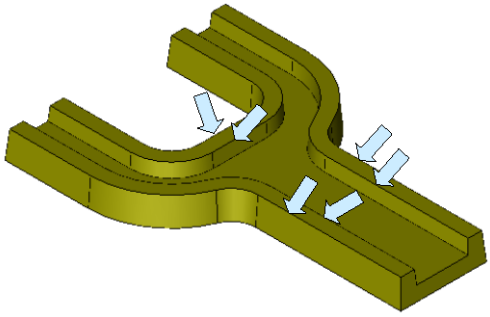
Click on the downward-facing vector.



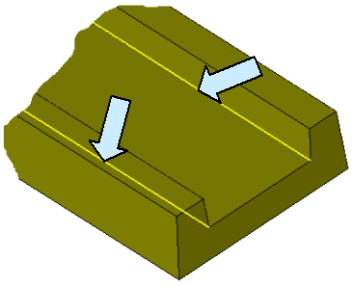
Your part should look like this:



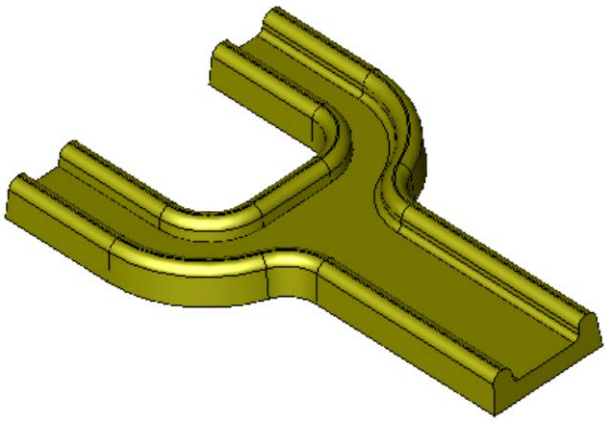
Now, click on the CONSTANT RADIUS BLEND Icon.



Create 0.125 radius blends on the top edges of the part indicated by arrows in the illustration to the left.



Then create 0.075 blends on the two inside edges indicated to the right.



Your part should now look like this:

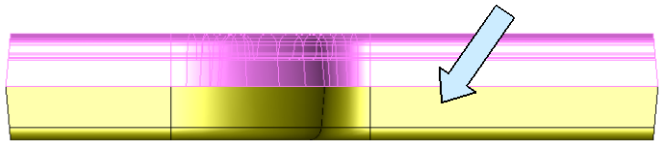

Now, switch to the Front View. (View 2.)

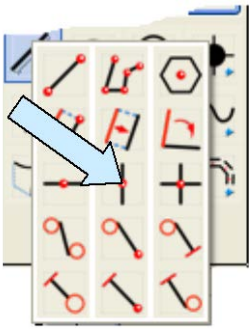

Click on the XFORM MIRROR COPY Icon.

Select the solid and hit the ENTER Key.

Click on the 1 Pos H Option and using the EndEnt Option, click on the bottom, left end of the part.

You will get a copy that looks like this:

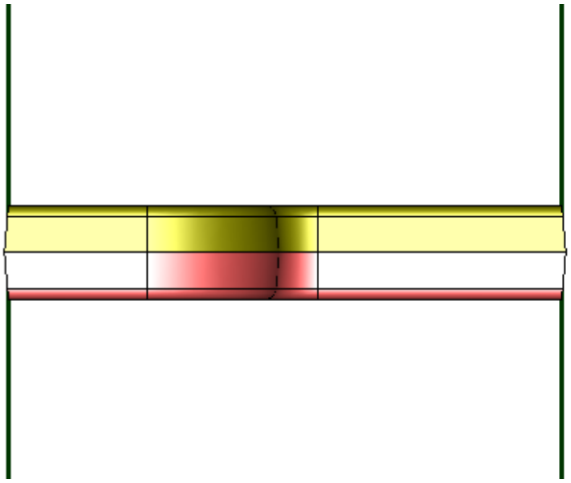


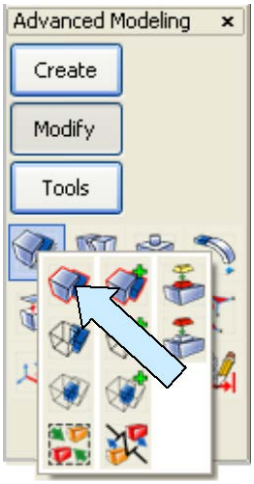


Now, click on the CREATE VERTICAL LINE Icon.

Using the EndEnt Option, click on the left end of the top edge, and the right end of the top edge.

Your screen should look like this:






Next, click on the BOOLEAN UNION Icon.

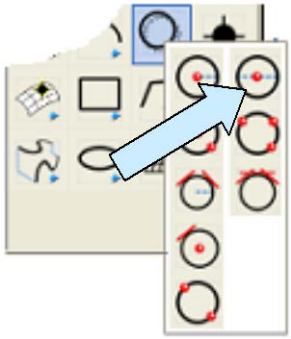
Select the two solids and hit the ENTER Key.


Click on the TRIM FIRST Icon.

Click on the middle of the solid and then on the left vertical line. Next, click on the middle of the solid and then on the right, vertical line.

Switch back to the Isometric View. (View 7.)

Click on the CONSTRUCTION PLANE Icon and type “1” for the construction plane.

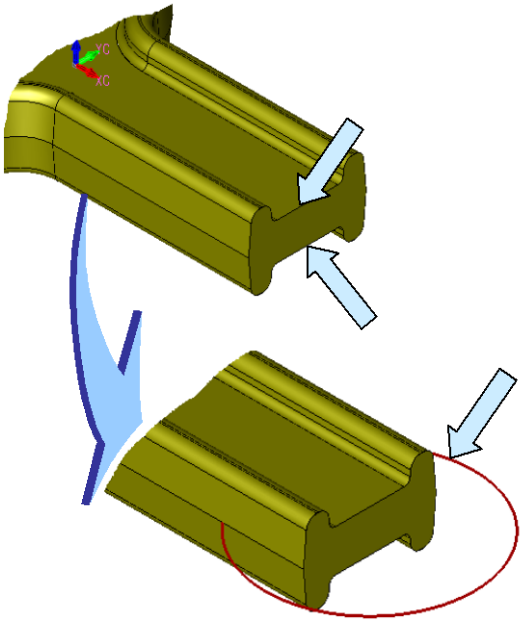


Next, click on the CREATE CIRCLE BY DIAMETER Icon.

Type 2.5 for the diameter.
Click on the TwoPos Option.

Using the CtrMid Option, click on the two horizontal edges indicated by arrows in the illustration to the right.

The resulting circle will be used to generate the large boss with draft at the right end of the part.

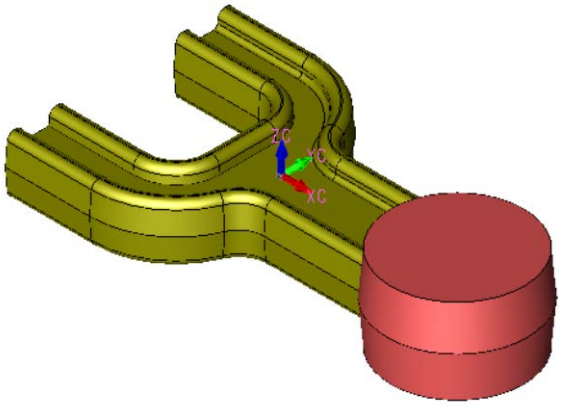



Click on the EXTRUDE Icon.

A Dialog Box appears.
Type 1.5 for the Length. Select the Midplane option.

Type 5 for the Draft Angle and select the Inward Option.

Select the 2.5 diameter circle and hit the ENTER Key

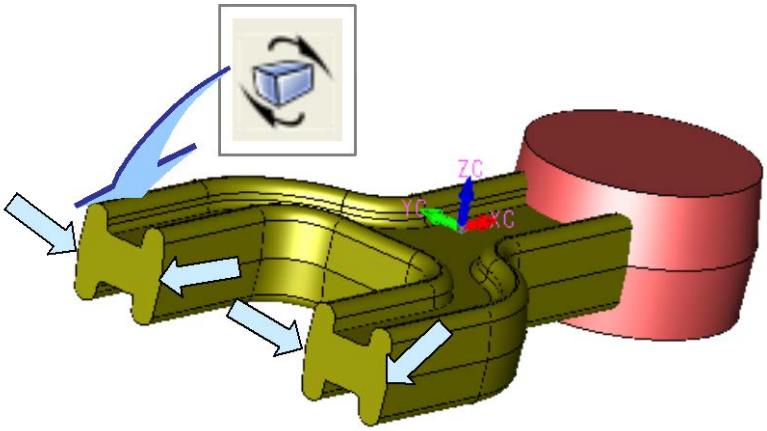


You will now have a boss with draft from the midplane at the right end of the part.

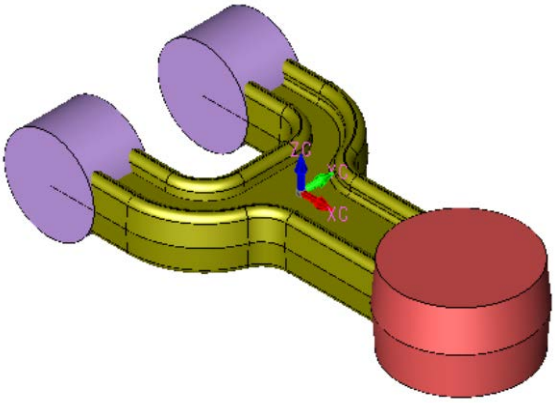

Now, click on the CYLINDER Icon.

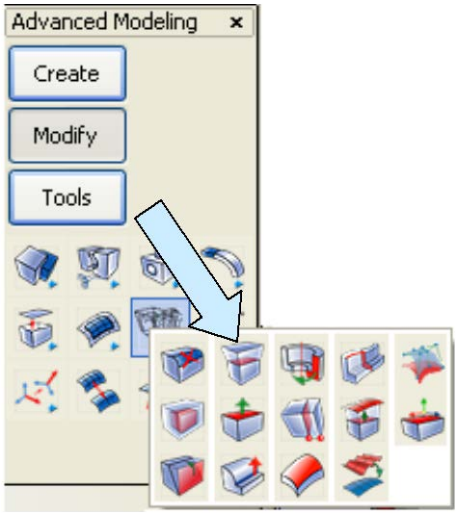
A Dialog Box appears.
Select the Two Positions Option.
Type 0.875 for the Radius and hit the ENTER Key.

Using the EndEnt Option, click on the two spots indicated by arrows on each leg of the left side of the part to place a cylinder on each leg. (Note: I rotated the part around in this view so you can clearly see the locations.)



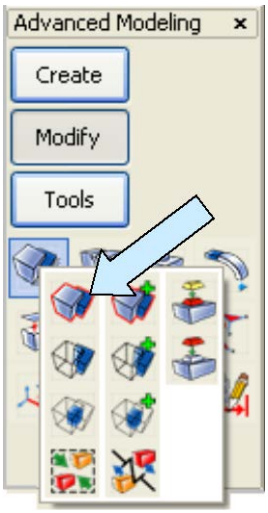
Your construction should now look like this:






Click on the EXTRUDE FACE Icon.

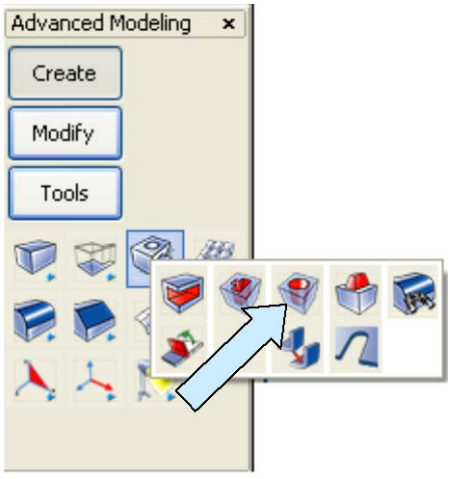
Type 0.25 for the length.
Click on the front and back circular face of each cylinder.




Now, click on the BOOLEAN UNION Icon.

Click on the main part and then on the two cylinders and drafted boss. Hit the ENTER Key.

You now have one solid on the screen.




Next, click on the DRILL Icon.
A Dialog Box appears. Select the Through Hole Option. Type 1.75 for the Diameter and hit the ENTER Key.

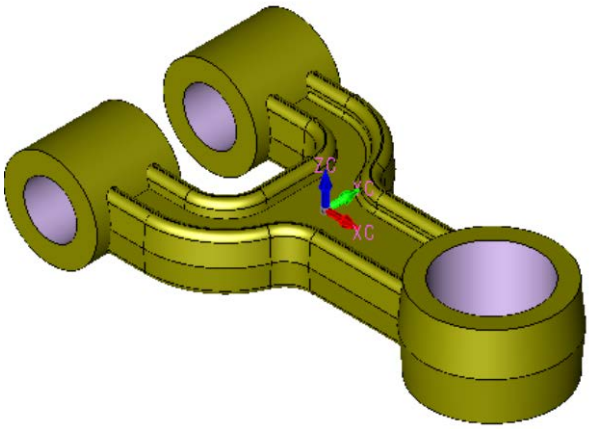
Click on the top face of the boss on the right end of the part.

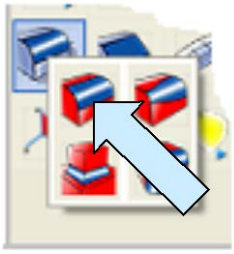
Then, using the CtrMid Option, click on the top, circular edge of the boss.

Hit the BACKUP Button twice. The Dialog Box reappears.
Type 1 for the Diameter and hit the ENTER Key.

Now, click on the front face of the front cylindrical boss on the left end of the part.
Then, using the CtrMid Option, click on the front, circular edge of that boss.

Your part should now look like this.



You can use the CONSTANT RADIUS BLEND Function to add cosmetic blends to the part.