
Introduction to assemblies

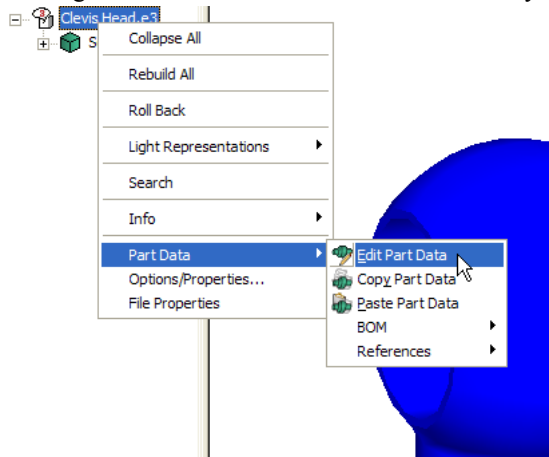
ThinkDesign's assembly module supports both Top-Down and Bottom-Up forms of assembly. The purview of this task is to bring out the concepts involved in working with both these approaches. It also aims to make you comfortable with the assembly environment so that you can learn and work on in house assembly design. Here we start.

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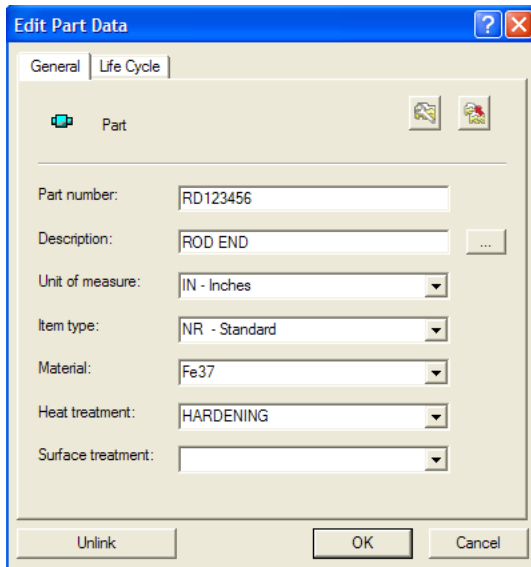
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1. Step1 : Part Data and X Refs

- **Open** file Clevis Head.e3 from the task installation folder if it is not already open. This file will reside in the files folder inside the corresponding task folder wherever it is downloaded.
- **Right Click** on the Model name in the History tree and select **Edit Part Data**



Check out the Data filled in each parameter of this Part. You can change them if you want.



Note that the file does not contain a Component. If the file is going to be inserted as an x-ref in an assembly, do not turn the solid into a component in the part file. Doing so would result in an X-ref subassembly being inserted into the assembly instead of an X-ref component as desired.

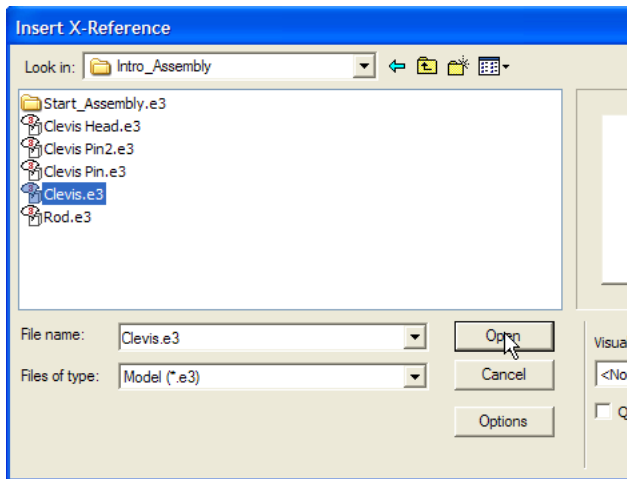
- **Save** the file and **Close** it.

We will insert the parts as X-ref components into an assembly file.

- **Open** file Start_Assembly.e3

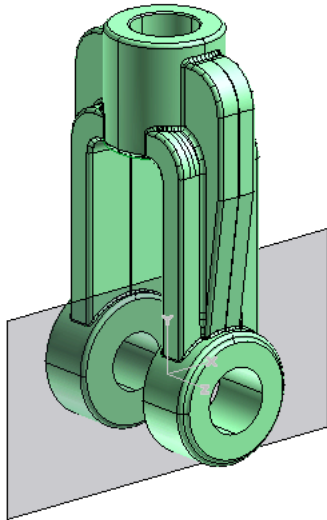
The file is empty.

- Click **X-Reference Component** command
- Browse to select file Clevis.e3



- Click on **Open**.

The component shows up in the ThinkDesign window.



The placement of the work plane on the part when it was saved will help you get it at the same location when imported into the assembly.

The concept to be understood here is that when you bring in a Model as an X ref component.

1. It Creates a link between the part file and the assembly.
- 2.Changes can be made either in the part or the assembly and both files are updated.

2. Step2 : Solid Mating

In this Step, we will work with mating different components using bottom up approach.We will bring in the Parts as External references and mate them using Solid Mating constraints.Before this check out the Mating table..

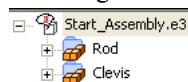
- Open ThinkDesign Help and Search for Mating Table. Show below is a part of the table.

	Coincident	Parallel	Tangent	Coaxial	On Face	Concentric	Perpendicular	On Curve	On Point
Planar Face	Planar Face/ Datum Plane	Planar Face/ Datum Plane	Cylindrical Face				Linear Edge		Point/ Datum Point
Cylindrical Face			Cylindrical Face/ Datum Plane	Cylindrical Face/ Revolution Face/ Linear Edge					Point/ Datum Point
Revolution Face			Planar face/ Datum Plane/ Cylindrical Face	Cylindrical Face/ Revolution Face/ Linear Edge					Point/ Datum Point
Linear Edge		Linear Edge		Linear Edge			Planar Face/ Datum Plane		Point/ Datum Point
Point	Point/ Datum Point				Planar Face/ Datum Plane			Curved Edge	
Circular Edge	Point					Circular Edge			Point/ Datum Point
Curved Edge									Point/ Datum Point

Understanding that the type of feature selected will determine the type of Mating constraints that can be applied.


OK, Let's get on with Some action.

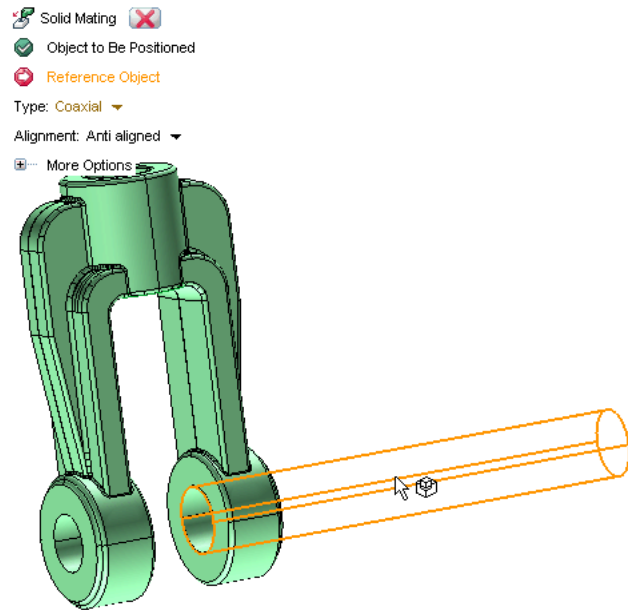
- Bring in Rod.e3 as an X-Reference Component




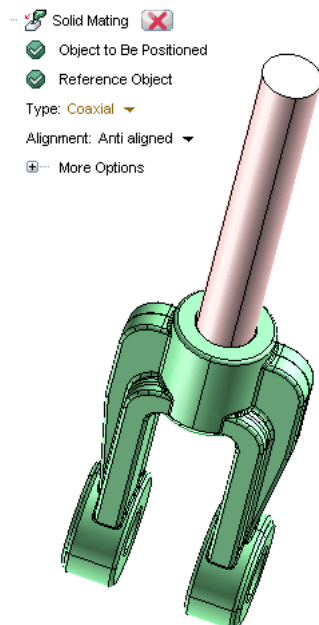
- Check the History to see that the two components show up.

The Orange icon depicts an X Ref component. We will now go about the Mating operation.

- Start the **Solid Mating** command.
- Select the Cylindrical surface of Rod for  Object to be positioned
- Set Type Coaxial.




- Pick the Inner Surface of the Clevis top for  Reference Object as shown.

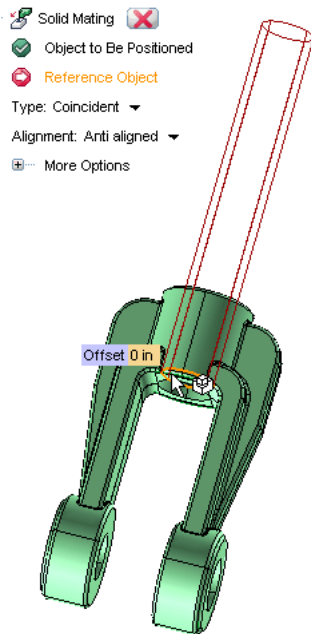


 Cancel

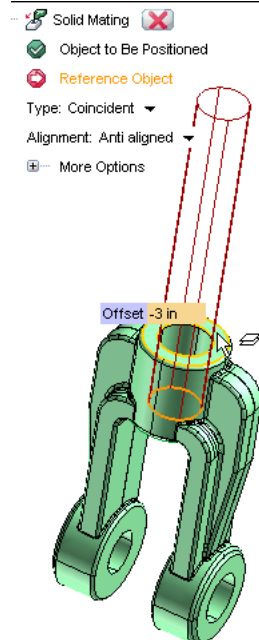
Fine, you are through with your first constraint.


Let's add another mating constraint.

- Start the **Solid Mating** command.
- Select the Face of Rod as the  Object to be positioned.
- Set Type Coincident, Alignment Anti aligned, Mode Simple.



- Pick the top face of the Clevis top as the  Reference Object and give an offset of Offset-3.




Click on the  Cancel button to confirm the constraint and exit the command.

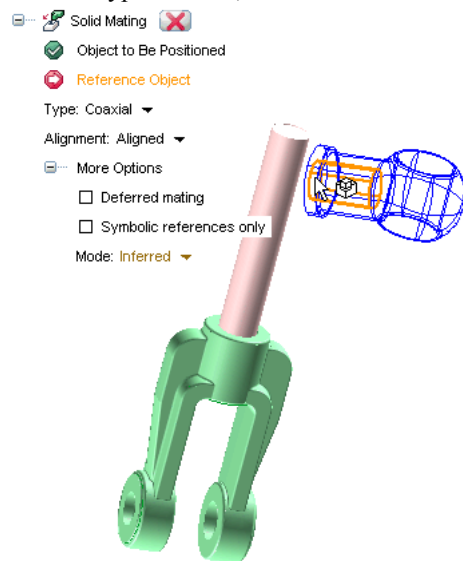
Points to be kept in mind are:.


- 1.Mating is a history based event. Mating constraints can be re-defined or deleted out of history.

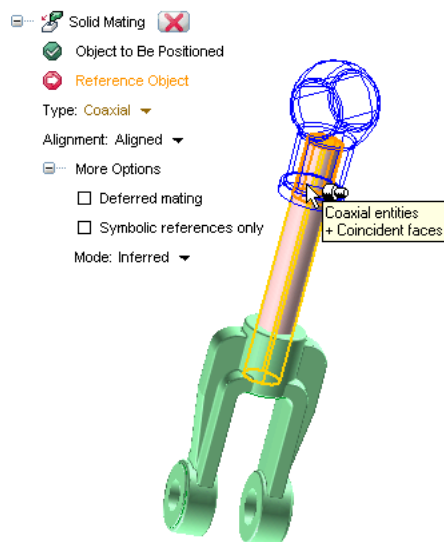
2. A component does not have to be fully constrained. In this case the Rod still has one degree of freedom, which is OK.

We will continue and bring in two more components into this assembly.

- Bring in Clevis Head.e3 as an **X-Reference Component**
- Start the **Solid Mating** command.
- Select the Inner Surface of Clevis Head as the  Object to be positioned.
- Set Type Coaxial, Mode Inferred.



- Pick Rod Surface as the  Reference Object.




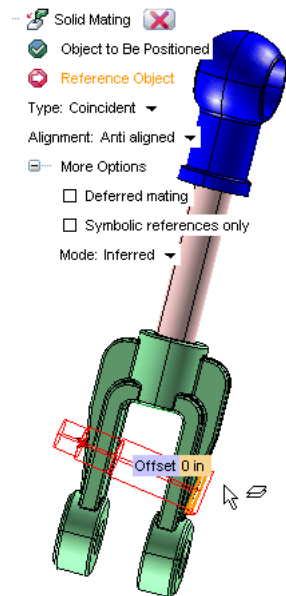
As you do this, a pop up showing that Coaxial and Coincident mating is being inferred comes up.

 Cancel

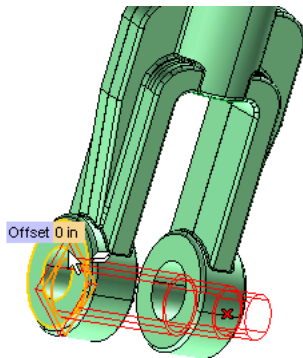
Wow inferred mating is Intelligent. Isn't it!!!

Let's bring in the Clevis Pin.

- Bring in Clevis Pin.e3 as an **X-Reference Component**.
- Start the **Solid Mating** command.
- Select the Inner Surface of Clevis Pin as  Object to be positioned.
- Set Type Coincident, Alignment Anti aligned, Mode Inferred.




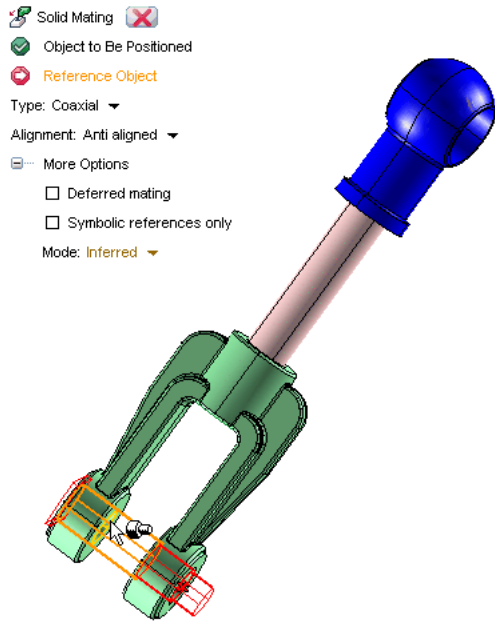
- Pick the Outer face of Clevis as the  Reference Object.



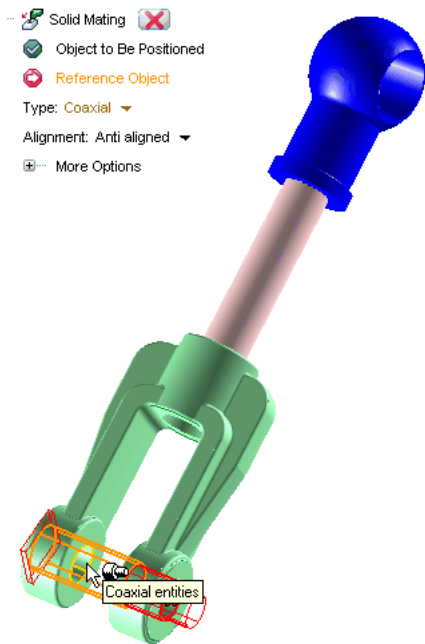
 Cancel

Insert another mating to secure this assembly.

- Start the **Solid Mating** command.
- Select the Surface of Clevis Pin as the  Object to be positioned.
- Set Type Coaxial, Mode Inferred.



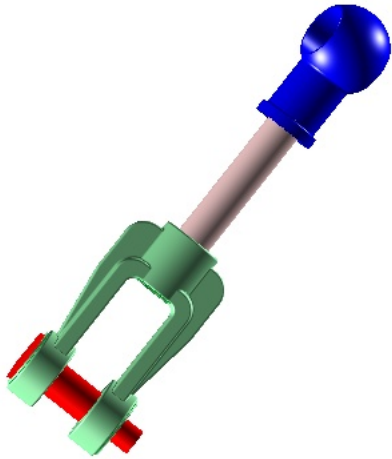
- Pick inner surface of Clevis hole as the Reference Object.



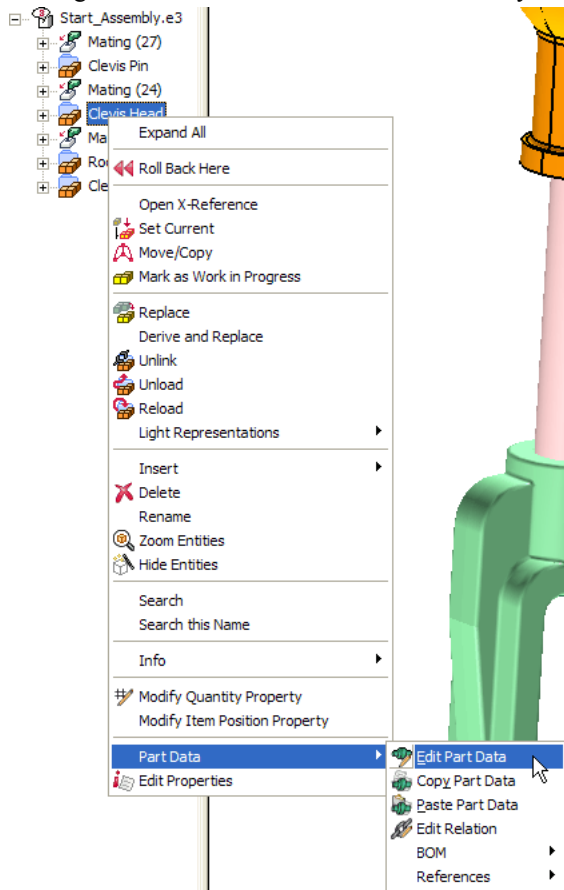
You can see the Coaxial faces mating indicator message showing up.

- Cancel to exit the command placing the constraint.

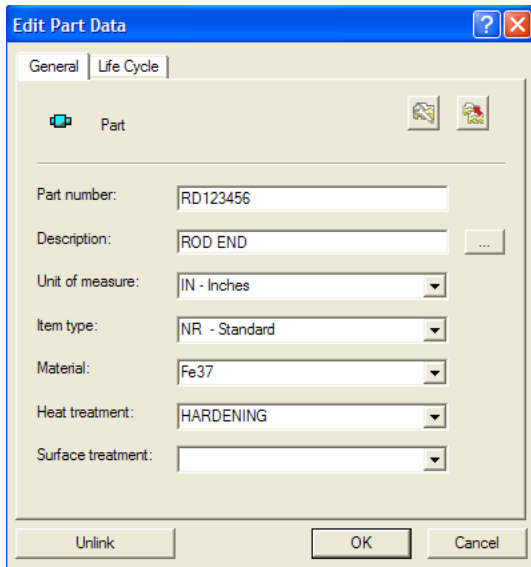
The assembly is done and looks like this.



- Right click on the Clevis Head in the History tree and say **Edit Part Data**.



You see that the Part Data entered in the part file is brought into the Assembly file.

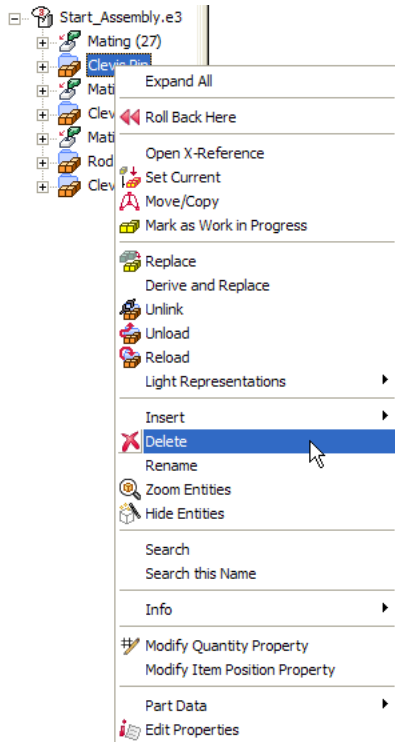


You are now familiar with the mating commands. Let's look at Top Down Approach in the next step.

3. Step3 :Top Down Assembly

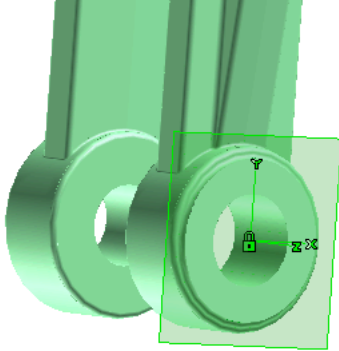
You have just seen how components can be added into an assembly file and then mating constraints inserted between them. This is the Bottom Up assembly approach. We will now see how we can directly construct a solid model in an assembly and mate it to existing component, thus showing you the ease and flexibility that ThinkDesign provides, when you want to work in Top Down assembly approach. Let's delete the component Clevis Pin from the assembly that we just created and create it directly in the assembly file and mate it as we did before.

- Right click on Clevis Pin in the History tree and say **Delete**



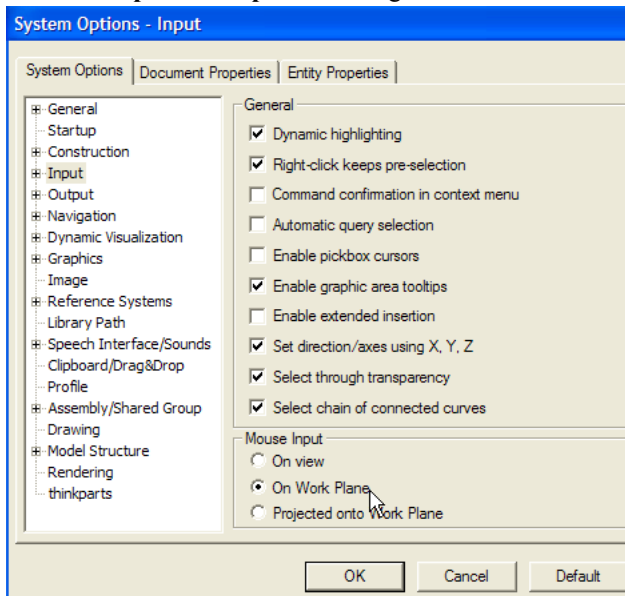
- **Hide Entities** Rod and Clevis Head as we do not need them now.

Now Check how the Work Plane is placed. Right click on the Work plane and choose the Set on Face option.

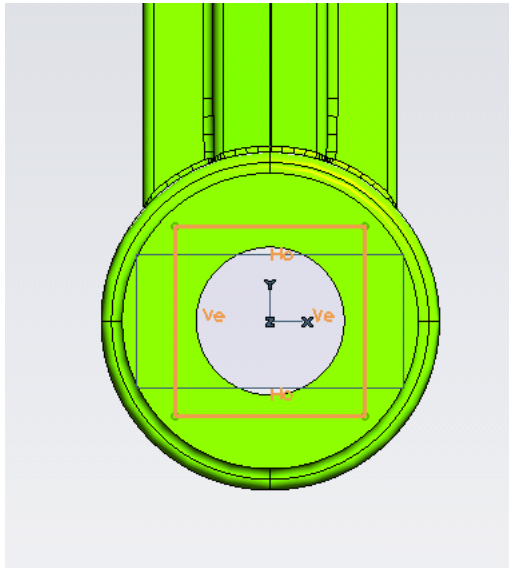



Do not Double Click on this face to set the Plane as it will set the Component Current. We want to Work using Top Down approach at the top of the assembly level.

- Go to **Options/Properties** dialog box and ensure that Mouse Input is set to On Work Plane as shown..




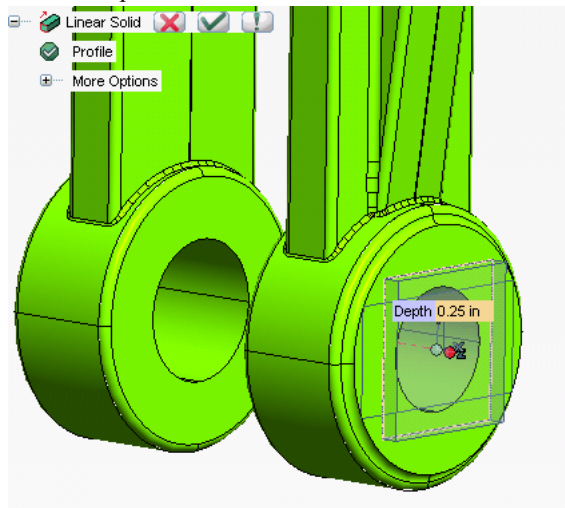
- Hit OK to switch the View to Work Plane view.
- Start the **Rectangle** command in the Profile mode.
- Set Mode Cen+Sizes and Snap to the Work Plane Center for Origin.
- Set X and Y sizes to 2.25 x 2.25




Click on  Cancel to place the Square Profile.

Note :Reference 1 -The Work plane is linked to the component Clevis,therefore the square profile is also linked to that component creating a reference between the Clevis and Clevis Pin that we are constructing.

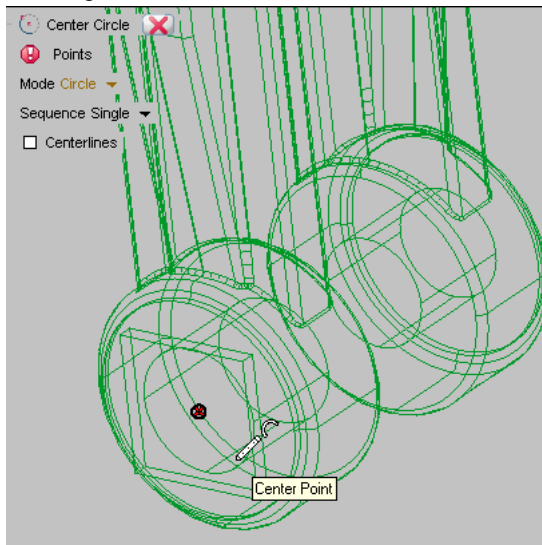
- Start the **Linear Solid** Command
- Pick the  Profile
- Set Depth 0.25 inch



 OK

- Switch the display to **Wireframe View**.
- Switch to  Profile mode.
- Use the **Center Circle** command to create the next profile.

- Snap the center of the circle to the center of the hole. Type in Diameter1.0.

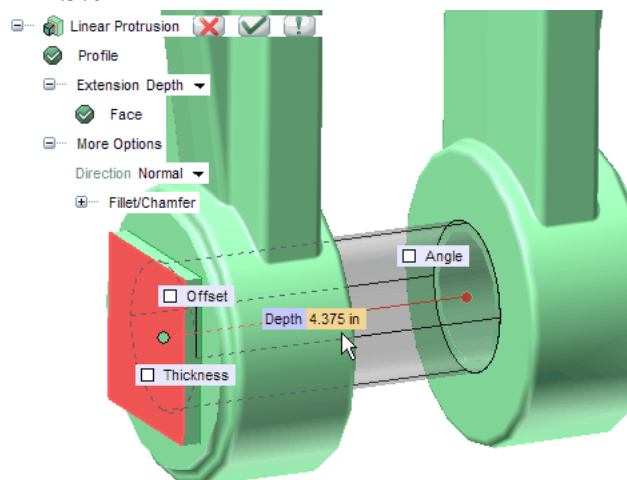


Cancel

- Add a dimensional constraint (radial dimension) to the circular profile.

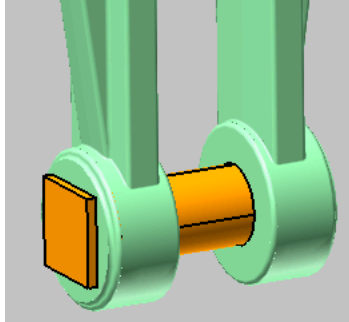
Note : Reference 2 ? The work plane was linked to the face of Clevis, therefore the circle profile is also linked to that component. Also, there is a coincident constraint between the center of the circle and the center of the hole (automatically placed on the profile when center snap was used to create circle)

- Start the **Linear Protrusion** command.
- Pick the Circular Profile as Profile
- Pick the Face of the Square Protrusion which is on Work Plane as Face
- Drag the Depth handle and Snap it to the Center of the hole on opposite Clevis arm or key in a length of 4.375





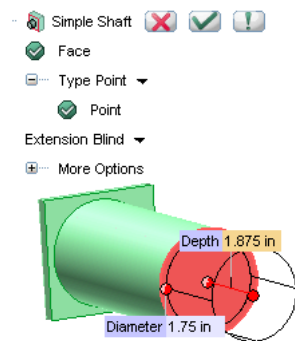
OK

At this stage, the Clevis Pin is as shown.





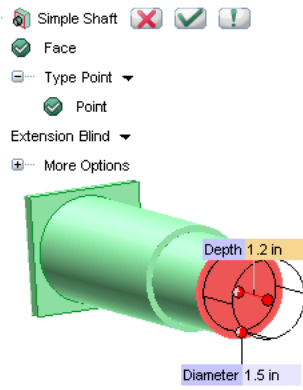
We will now add two shafts to the Clevis Pin to complete its design.

- Use **Hide Entities** to hide component Clevis
- Start the **Simple Shaft** Command
- Select the end face of Clevis Pin as  Face.
- Snap to the Center of face for  Point
- Set Diameter 1.75 and Depth 1.875.



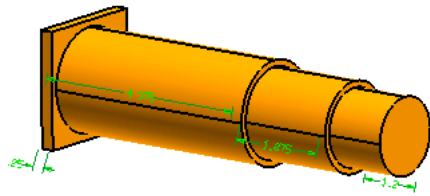
OK

- Again Start the **Simple Shaft** Command
- Select the end face of Clevis Pin as  Face.
- Snap to the Center of face for  Point.
- Set Diameter 1.5 and Depth 1.2.



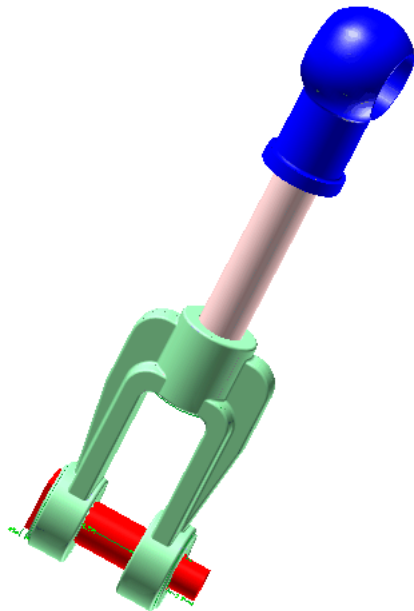
OK

Fine, you are done with the design of the Clevis Pin exactly as it was before but this time you worked directly in the assembly mode, using Top Down approach and used references from existing components. This is the final design

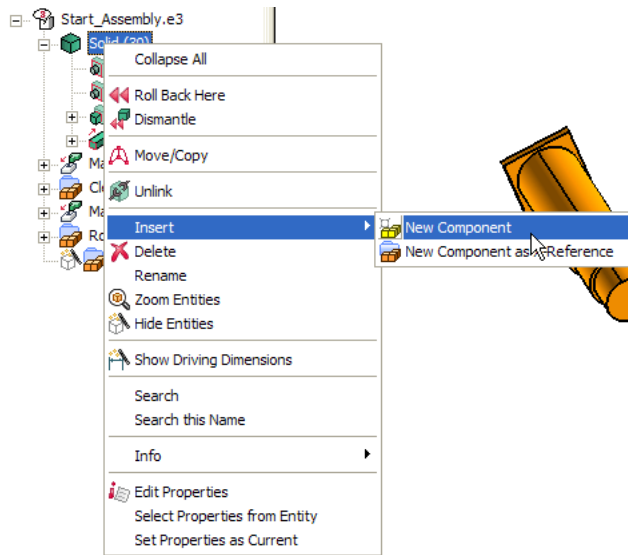


Let's now make this newly designed Clevis Pin as an **X-Reference Component**.

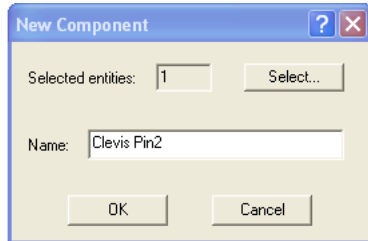
- First **Unhide Entities** all the other components in the assembly.



- Right Click on Solid and Say **New Component**

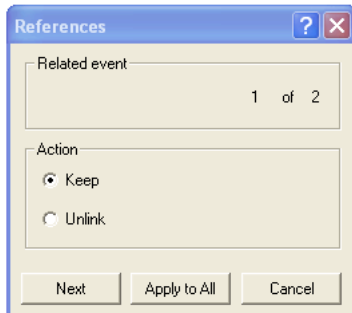


- Give the name of the Component as Clevis Pin2.



- Hit OK.
- In the References dialog box, the number of references, which is 2 in this case, is listed in the Total box.

At this point, you can choose to either Keep all the references.... OR you can scroll through each reference and choose to Keep or Unlink each one.



- Unlink both references.

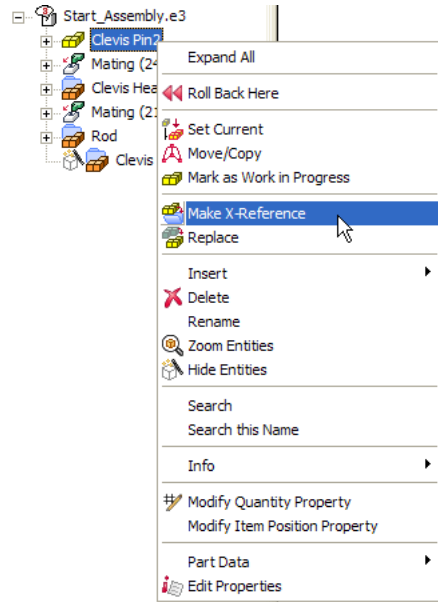
Note: ThinkDesign will highlight each reference in the reverse order that they were created. The circular profile will be highlighted in orange first. After hitting Next in the dialog box, the square profile will be highlighted.

Another important aspect here is that by Unlinking the references, you essentially create a component that is independent of the other components, but in the process you may lose some constraints. On the flip side, if you keep the references, you retain the constraints but that component may not be used in other assemblies.

You need to be aware and user discretion is recommended while performing this operation depending on the design requirements.

Now the Clevis Pin is a Local Component.

Right click on the Clevis Pin in the History Tree. Select the **Make Component X-Reference**.



- Select a directory in which you can save the X-ref file.

Great!!!. We have learnt how easy it is to work with ThinkDesign assembly command, be it a Bottom Up or Top Down approach.

A.

- Introduction to Solid Modeling.
- Mating of Components.