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Introduction

CATIA Version 6 Assembly Design

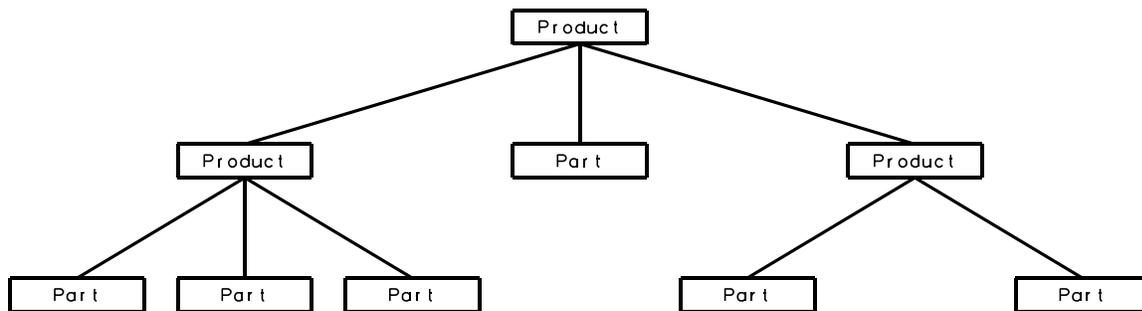
Upon completion of this course the student should have a full understanding of the following topics:

- Inserting models into an assembly
- Manipulating models in an assembly
- Constraining models in an assembly
- Using engineering connections to define kinematic joints
- Modeling within the context of the assembly
- Creating and using publications
- Analyzing assemblies for clashes and gaps
- Modifying assembly components and updating assemblies

Assembly Design

Very few finished designs are a single part. Usually a finished design consists of several to millions of individual parts to define them. This is where CATIA V6 Assembly Design is utilized. Assembly Design allows parts and small assemblies of parts to be inserted to make larger, more complete products. In CATIA V6 Part Design and Sketcher, you learned how to generate parts. The primary objective of this class is to utilize those parts to create a complex assembly of those parts that can be later used in stress analysis, kinematics, fitting simulations, and other areas.

It is important to understand some of the terminology that CATIA uses when working with assemblies. There are basically three types of documents that are used in Assembly Design: the overall assembly, sub-assemblies, and individual part models. CATIA uses the word 'product' to refer to an assembly, and 'part' to refer to an individual model. You can use parts to create products and, in turn, use those products to produce other products. The diagram shown below represents the concept of the overall structure.



The first product at the top is generally regarded as the assembly, whereas the two products that are underneath are generally regarded as sub-assemblies of this assembly. This assembly could in turn be used to create an even bigger assembly at some other time, or the sub-assemblies could be used as sub-assemblies of a different assembly. With this concept in mind be aware that an assembly could be a very complex document due to its ability to have multiple levels of sub-assemblies and parts. Because of this complexity it is important that you have a plan of attack when building assemblies. There are basically two approaches that a user or company can take when building assemblies. One is to pre-determine what sub-assemblies a particular assembly is going to need. The other is to produce all of the parts and then determine what sub-assemblies are going to be created.

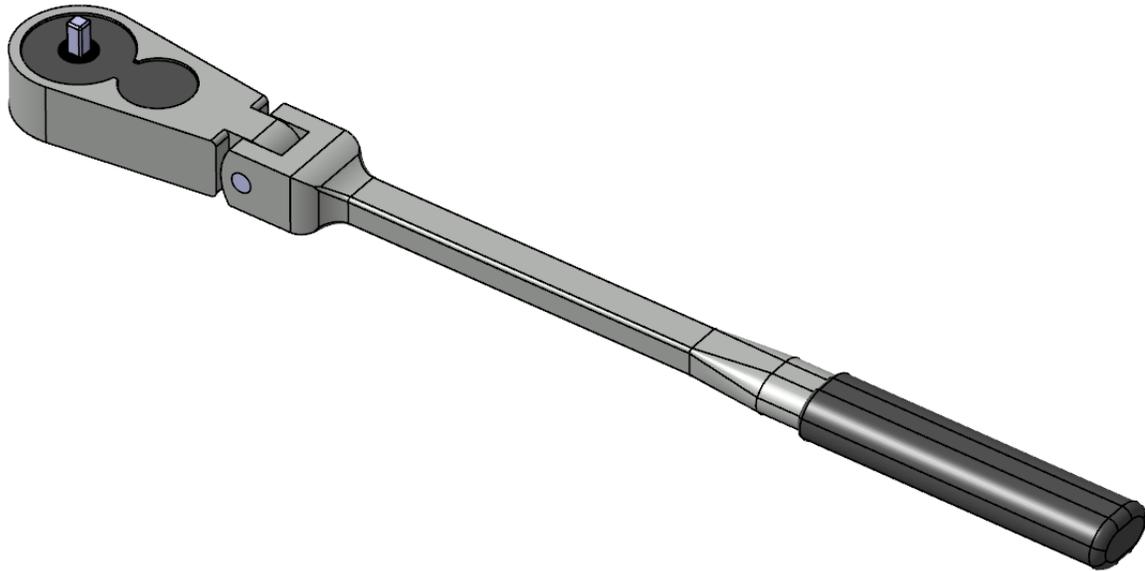
Assembly Design

The first section of this manual will involve inserting, creating, and replacing documents and other components in the assembly design. Those documents can be a variety of things including parts and other assemblies.

*Note: All icons utilized on a less-frequent basis are denoted by a * throughout this manual.*

Inserting Documents

All of the assemblies created in this first section will not need to have constraints added to properly position them. The first assembly that will be built is a basic ratchet wrench.



Select the **Add icon** and select *Physical Product*.  The *Physical Product* window appears.

Change the *Title* to **ASSY010 - Ratchet Wrench**. Select *OK* when done. This will give the assembly an unique name.

One of the most important ideas to keep in mind with assembly design is that all parts must have an unique ID. The assembly should also have an unique ID, especially if it is going to be used as a sub-assembly.

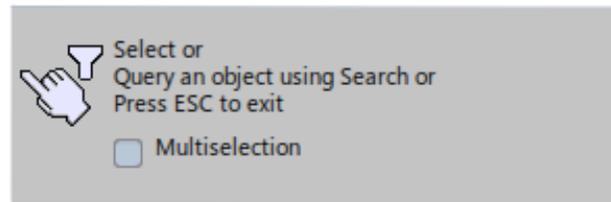
Be sure you are in the Assembly Design workbench. It should be listed along the top of the window.

3DEXPERIENCE | CATIA Assembly Design

If you were not in the Assembly Design workbench, you could switch to it by using the West quadrant of the compass in the upper left corner of the window.

Select the Insert Existing 3D Part icon.  It is located in the *Product Edition* section in the sub-toolbar of the Insert Existing Product icon. The command is waiting for the user to select a product to insert the component into.

Select ASSY010 - Ratchet Wrench in the tree. This will define what product the component will be placed into. A prompt appears.



Key in ASSY010 - Handle in the Search field as shown.



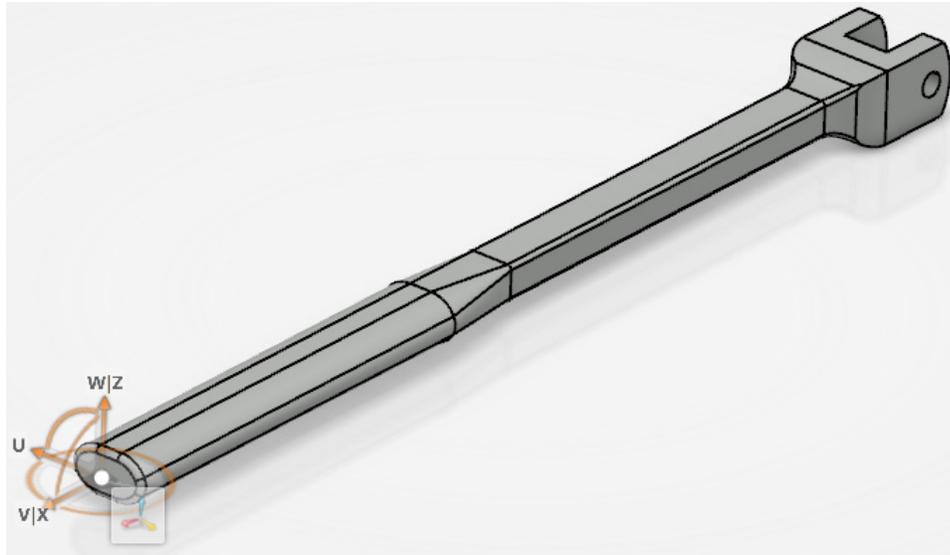
Select Enter to execute the search. The results should appear. In this case, the results are being shown in Datagrid View.  You may switch the view display using the 3 icons in the upper right corner of the window.

2 Results | 

	<input type="checkbox"/>	Actions	Title	Type
1	<input type="checkbox"/>	▼	 ASSY010 - Handle Grip	Physical Product
2	<input type="checkbox"/>	▼	 ASSY010 - Handle	Physical Product

Select ASSY010 - Handle. This will specify the document to be inserted.

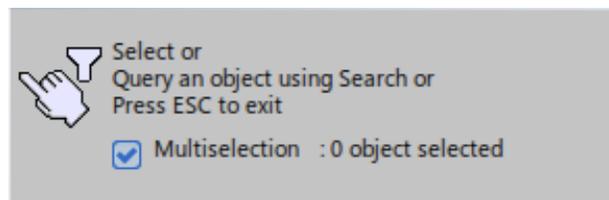
The handle is inserted into the product. Notice the robot is attached to the part, allowing you to move it within the assembly. The Move To Origin icon is also available.  It is helpful if you move the part using the compass and then decide to move it back to the origin.



Select in the display to set the part. The robot returns to the lower right hand corner of the window.

Press the third mouse button on *ASSY010 - Ratchet Wrench* and select *Insert, Existing 3D Part*.

Turn on the *Multiselection* option in the query box as shown. This will allow you to insert multiple parts at once.

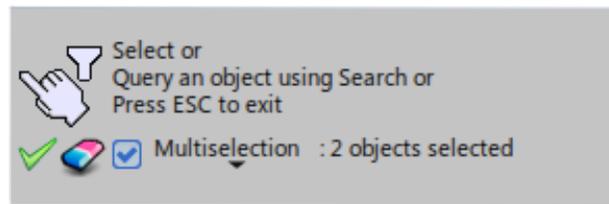


Search for ASSY010. This will locate all parts with the ASSY010 prefix. There should be 10 results.

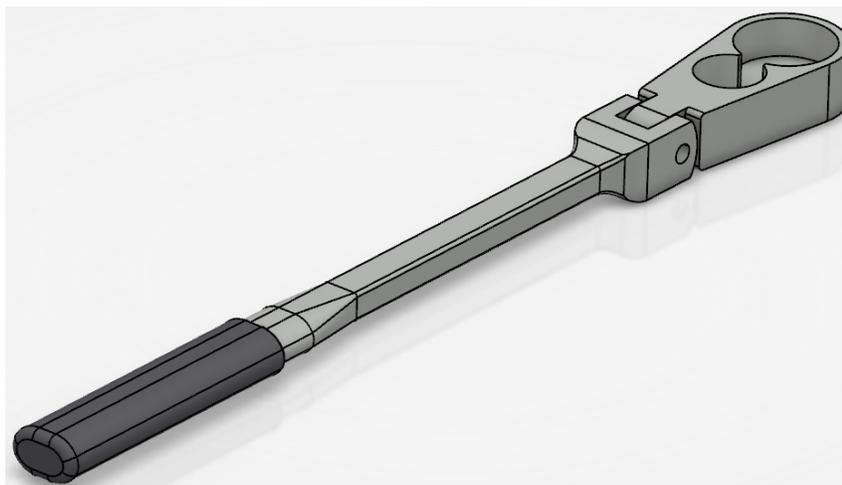
10 Results | 🕒

	<input type="checkbox"/>	Actions	Title	Type
1	<input type="checkbox"/>	▼	ASSY010 - Cover Plate	Physical Product
2	<input type="checkbox"/>	▼	ASSY010 - Flex Head	Physical Product
3	<input type="checkbox"/>	▼	ASSY010 - Handle Grip	Physical Product
4	<input type="checkbox"/>	▼	ASSY010 - Handle	Physical Product
5	<input type="checkbox"/>	▼	ASSY010 - Hex Socket Large	Physical Product
6	<input type="checkbox"/>	▼	ASSY010 - Large Sprocket	Physical Product
7	<input type="checkbox"/>	▼	ASSY010 - Socket Stub	Physical Product
8	<input type="checkbox"/>	▼	ASSY010 - Small Sprocket	Physical Product
9	<input type="checkbox"/>	▼	ASSY010 - Torx Bit Socket A	Physical Product
10	<input type="checkbox"/>	▼	ASSY010 - Torx Bit Socket B	Physical Product

Select ASSY010 - Flex Head and ASSY010 - Handle Grip. The query window should populate as shown.



Select the Accept All icon to insert them into the assembly. The parts are inserted and pre-positioned as shown.



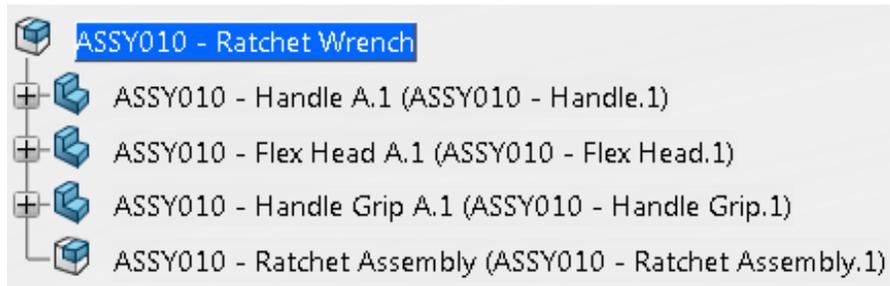
Sometimes components can be made into a sub-assembly. This can either be done beforehand, as with the Ratcheting Mechanism that you are going to insert later, or the sub-assemblies can be generated on-the-fly as you will do next.

Creating a New Product

When you create a new product within an existing product, also known as a sub-product, it is important to understand that this product is a document that will eventually need to be saved. A sub-product or sub-assembly is no different than a product other than the fact it is used within a higher level product.

Right select on ASSY010 - Ratchet Wrench and select *Insert, Product*. The Physical Product window appears.

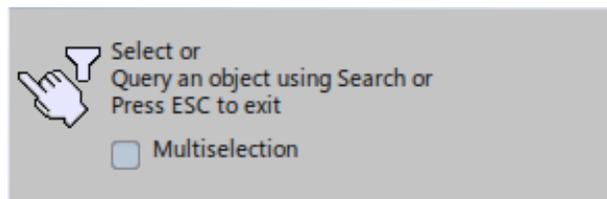
Key in ASSY010 - Ratchet Assembly for the *Title* and select *OK*. This will insert a new product into the assembly. It should appear as shown.



Now that you have a nested product assembly, a product within a product, you need to make sure that you insert new and existing components into the proper product.

Right select on *ASSY010 - Ratchet Assembly.1* and select *Insert, Existing Product*.

Turn off the *Multiselection* option in the query box.

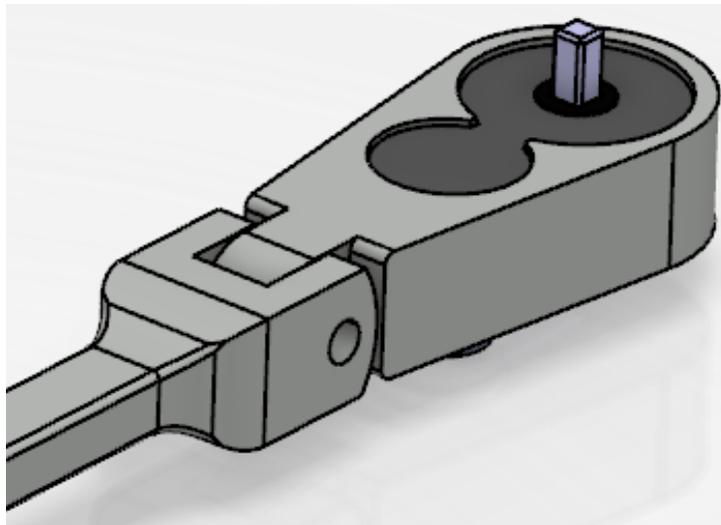


Key in ASSY010 in the *Search* field and select *Enter*. Select *ASSY010 - Ratcheting Mechanism* from the results and select in the display to set the location. This will insert the subassembly into the product.

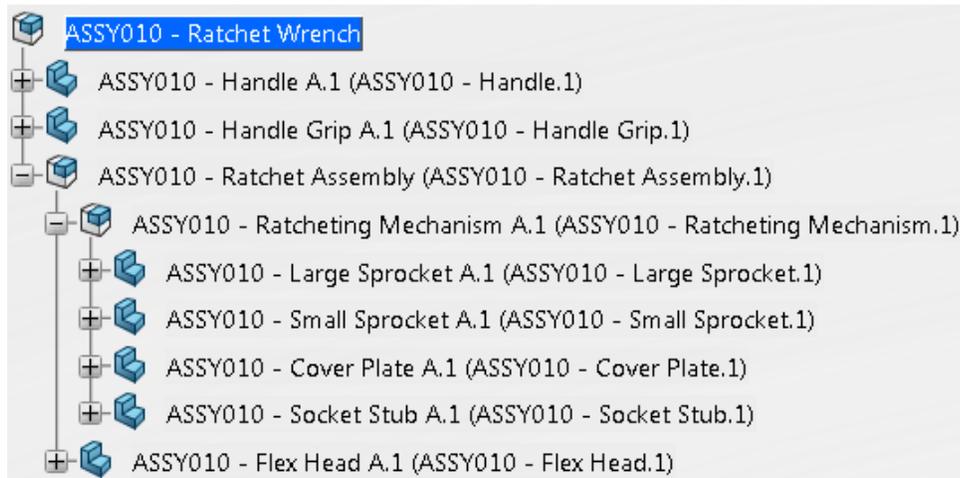
Expand the specification tree should appear as shown. You may want to rename the instance of the Ratcheting Mechanism.



The Flex Head part should be part of the Ratchet Assembly. It will be added to the new subassembly.



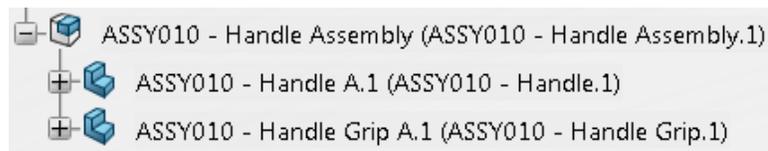
Drag the *ASSY010 - Flex Head* instance onto the *ASSY010 - Ratchet Assembly.1* product in the specification tree and drop it. It should appear as shown.



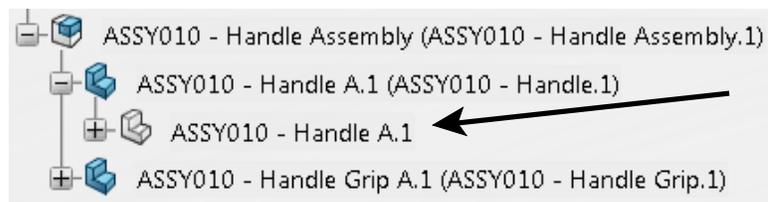
This is just another way of inserting parts or products into a subassembly. Dragging and dropping a component is the same as cutting the component and then pasting the component.

Insert a new product into the *ASSY010 - Ratchet Wrench* and set the *Title* to be ASSY010 - Handle Assembly. The component is created.

Drag both the *ASSY010 - Handle* and the *ASSY010 - Handle Grip* into the new product. It should appear as shown.

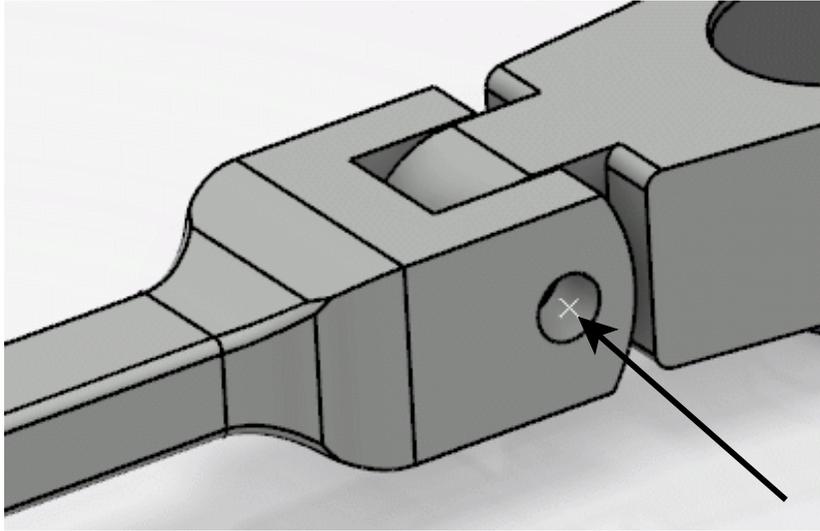


Double select on the *ASSY010 - Handle* shape representation as shown. This will activate the part and switch you into Part Design. The 3d shape representation should have a blue box appear around it in the specification tree.



The blue box signifies the active component within the product structure. Since the Handle is now the active component and it is a part document, you were put into a workbench for parts.

Show Geometrical Set.1 in the specification tree. You should see a point appear on the handle as shown.



The point will be used to locate a new part.

Double select *ASSY010 - Ratchet Wrench* to make it the active product. A blue box does not always appear around the high level product when it is the active component. A blue box will always appear around an active component at a lower level.

Creating a New Part

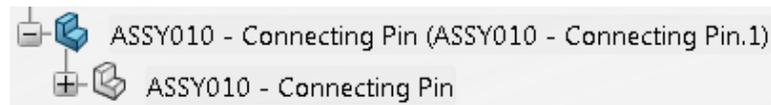
When you create a new part within the product, it is really no different than when you normally create a new part. It will be a separate document, and can be called up independent of the main product. When you create a new part you will have the option of defining the absolute origin of the part by specifying a location or it will establish its absolute origin to coincide with the main product's absolute origin.

Right select on *ASSY010 - Ratchet Wrench* and select *Insert, 3D Part*.

Change the *Title* to be ASSY010 - Connecting Pin. This will provide a unique name for the part to be saved into the database.

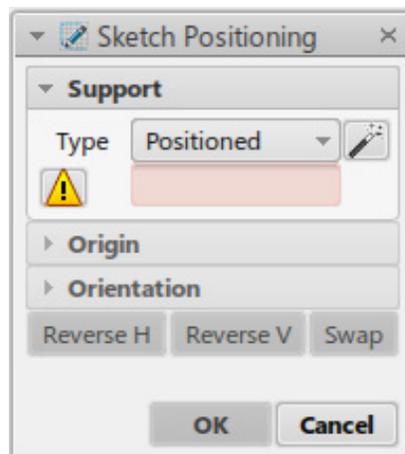
Select the *3D Shape* tab and change the *Title* to be ASSY010 - Connecting Pin. This will ensure the 3d shape name matches the 3d part name.

Select *OK*. It should appear as shown in the tree.



Double select on the *ASSY010 - Connecting Pin* shape representation in the specification tree. The part is activated and you are switched to Part Design.

Select the *Positioned Sketch* icon.  The *Sketch Positioning* window appears.

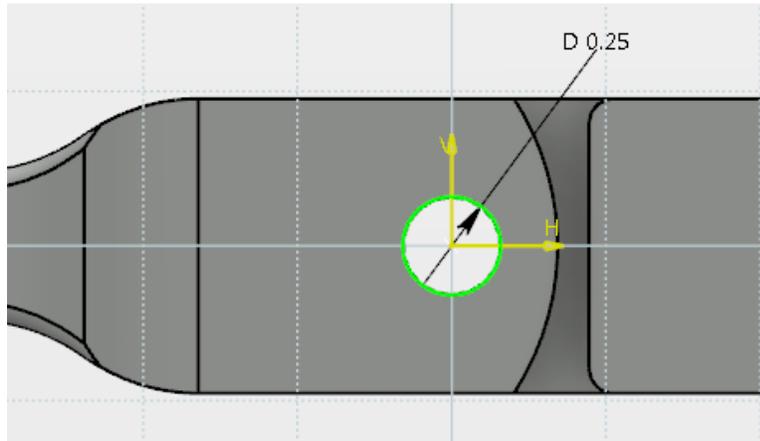


Select the *zx* plane of the part to define the sketch plane. It may be easiest to select the plane from the tree.

In the *Origin* area, change the drop-down menu to *Projection Point* and select the point that was shown earlier. You may need to turn off the Smart Positioning of the Support icon in order to use the drop-down menu. 

Select *OK*. You are taken into sketcher.

Create a 0.25 diameter circle at the origin of the sketch. It should appear as shown.



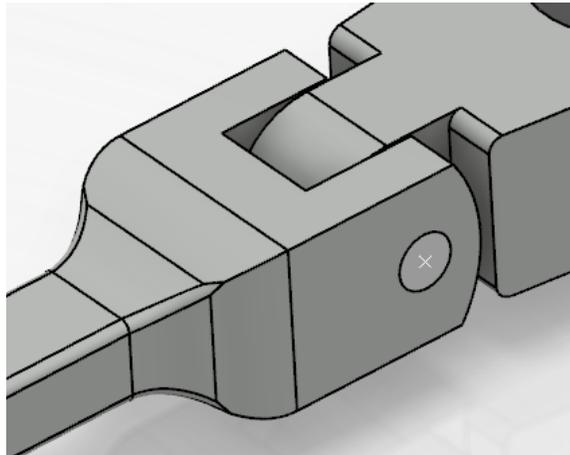
Select the *Exit Workbench* icon.  You are switched back to Part Design.

Be sure the sketch is selected and select the *Pad* icon.  The *Pad Definition* window appears.

Key in 0.5 for the *Length* of the pad, turn on the *Mirrored extent* option and select *OK*. The pin is created.

Hide the reference planes of the part. This will clean up the model a little bit.

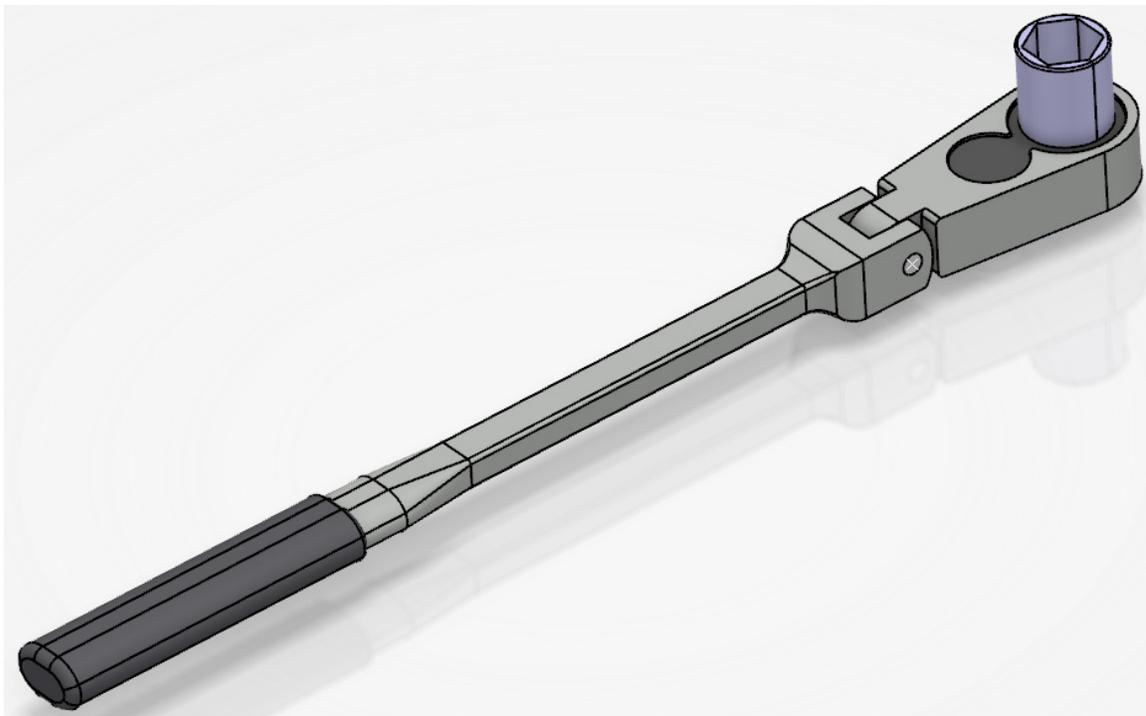
Double-select on *ASSY010 - Ratchet Wrench* to activate the product. The model should appear as shown. The pin should be flush on both sides with the handle.



Press the third mouse button on *ASSY010 - Ratchet Wrench* in the specification tree and select *Insert, Existing 3D Part*.

Search for the *ASSY010 - Hex Socket Large*. You will need to key in ASSY010 in the search field.

Select the *ASSY010 - Hex Socket Large* from the results list and then select in the display to set the location. The socket is inserted into the assembly.



Creating a New Part from an Existing Part

You will now create a new part from an existing part. You have to be careful when trying to create a new part from an existing part. If you have instanced the part into the product multiple times and then you try to change just one of them, they will all change. This happens since all of the instances refer to the same 3d shape representation. If you do create a new part from an existing part you will have to take special care in renaming the part and instance to use the correct name and then make sure you save it with a different name. Otherwise you will overwrite the original part.

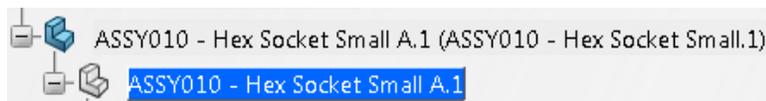
Double select on the *ASSY010 - Hex Socket Large* shape representation to activate the part.

Double select on *Sketch.1* in the specification tree and change the diameter constraint to be 0.75in. Exit the sketch. It is located in the *PartBody* under *Pad.1*.

Right select on *ASSY010 - Hex Socket Large* instance in the specification tree and select *Properties*. The *Properties* window appears.

Change the *Instance Title* to be ASSY010 - Hex Socket Small.1 under the *Instance* tab and the *Title* to be ASSY010 - Hex Socket Small under the *Reference* tab. Select *OK*.

Change the shape representation *Title* to be ASSY010 - Hex Socket Small as well. The tree should appear as shown. Make sure you remember to change not only the part number but also the instance name to avoid confusion. Now all you have to do is remember to save it with a different name so that you do not overwrite the original document.

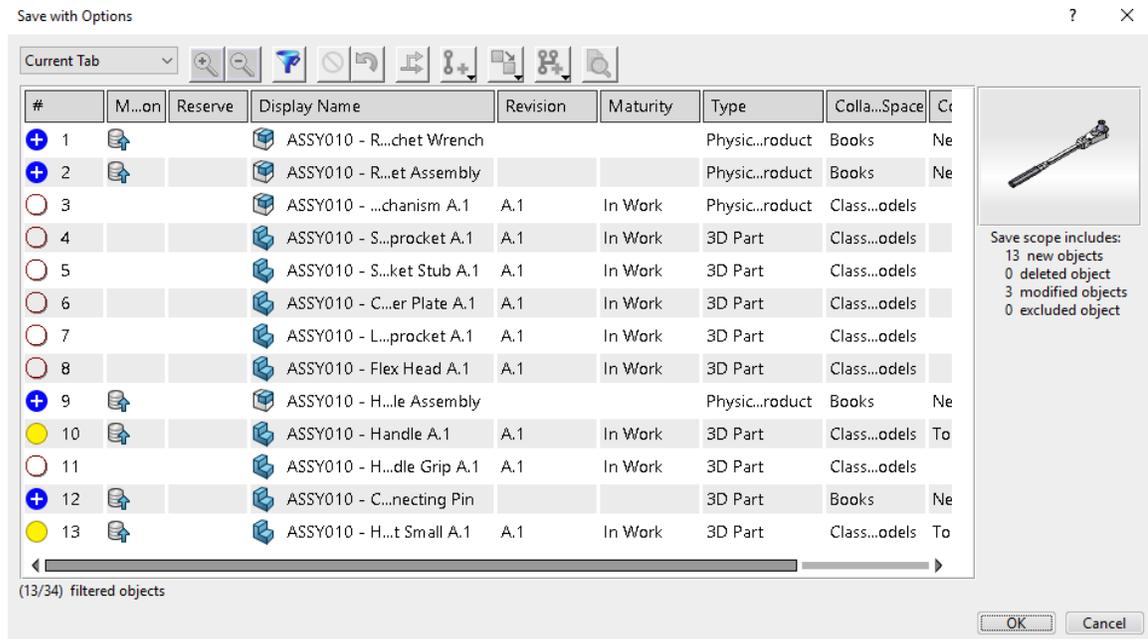


Double select on *ASSY010 - Ratchet Wrench* to activate the product.

Save

Save is used to save you work back to the database. This window gives you a graphical representation of what will be saved and how it will be saved before actually performing the save operation. The options are covered in detail in Appendix B.

Select the **Share** icon and select **Save with Options**.  You will have to select the expansion arrow next to the **Save** option to get to the additional save options. The **Save with Options** window appears.



There are various symbols in the left-hand column. The blue circle with the + means a new object to be saved to the database. A yellow circle represents a modified object to be saved. An empty circle represents no change to be saved to the database.

Notice the *ASSY010 - Connecting Pin* shows it is a new part to be saved to the database. This is because you created it from scratch.



Notice the *ASSY010 - Hex Socket Small* is shown as a modified part. This is because you modified the *Hex Socket Large* to create it. You will need to specify you want to save it as a new part and not save over the *Hex Socket Large* part.



Select the *ASSY010 - Hex Socket Small* part from the window and select the **Save As New icon**.  The *Save as New* window appears. This part needs to be saved as *ASSY010 - Hex Socket Small*. This will make it a unique new part in the database, so the original *Hex Socket Large* is not overwritten.



Since you already renamed this part when you modified it, change the *Duplication string* to *XXX* and select **OK**. The XXX should be your initials. You are returned to the *Save with Options* window. Notice the part is now designated as a new part.

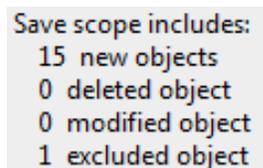


Select the *ASSY010 - Handle* from the list. Notice it says it was modified since you unhid the geometrical set within it. This isn't your part, so you do not want to save over it.

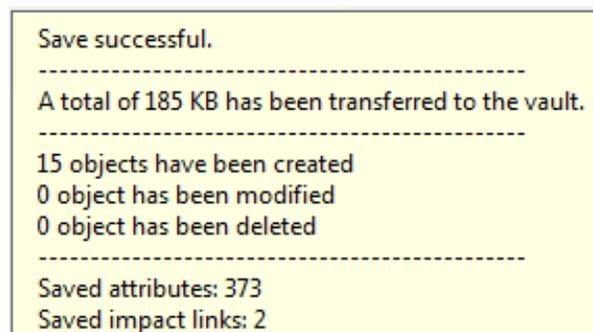


Select the **Exclude icon**.  It will now be excluded from the save.

Notice the summary on the right side of the window shows what is going to be saved.



Select **OK**. All the models are saved and a *Save successful* message appears.

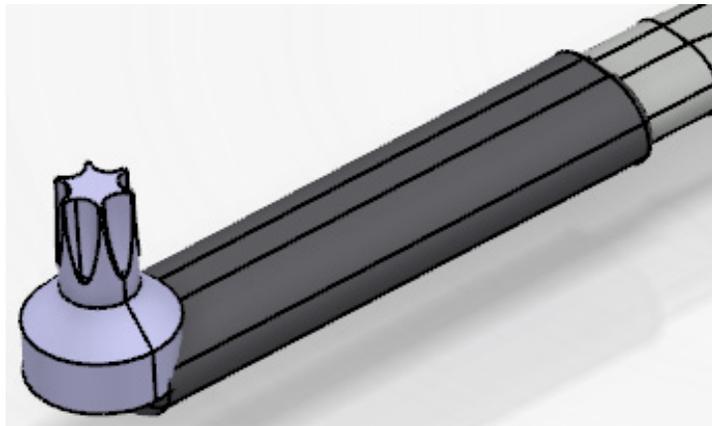


Replacing Parts

In many instances it is necessary to change a part of an assembly due to alternative designs. This exercise will cover how to replace components with other components without having to recreate the entire assembly.

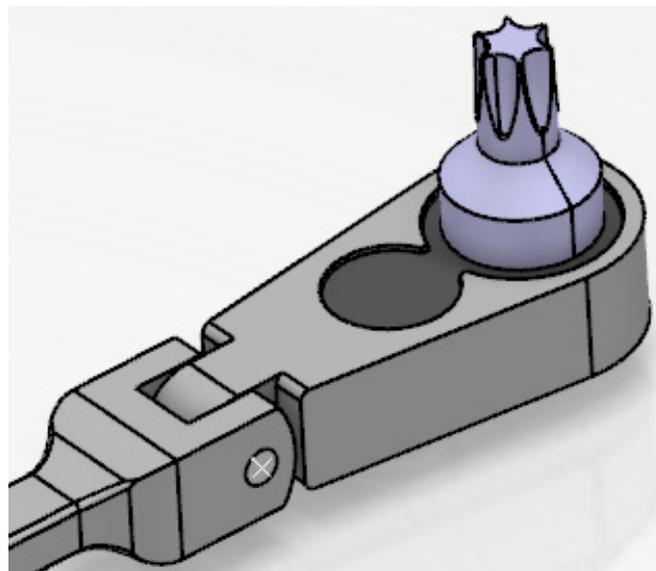
Press the third mouse button on *ASSY010 - Hex Socket Small.1* and select *Replace, Replace By Existing*. *ASSY010 - Hex Socket Small* will be replaced with *ASSY010 - Torx Bit Socket B*.

Search for *ASSY010 - Torx Bit Socket B* and select it and select in the display to set the location. The torx bit replaces the hex socket. Notice the torx bit does not come into the assembly at the correct location. This is because the part was built at the local axis system instead of in assembly coordinates.



Notice the *ASSY010 - Torx Bit Socket B* still has an instance name of *ASSY010 - Hex Socket Small.1*.

Using the same method, replace *ASSY010 - Torx Bit Socket B* with *ASSY010 - Torx Bit Socket A*. The new torx bit is located correctly because it was built in assembly coordinates.

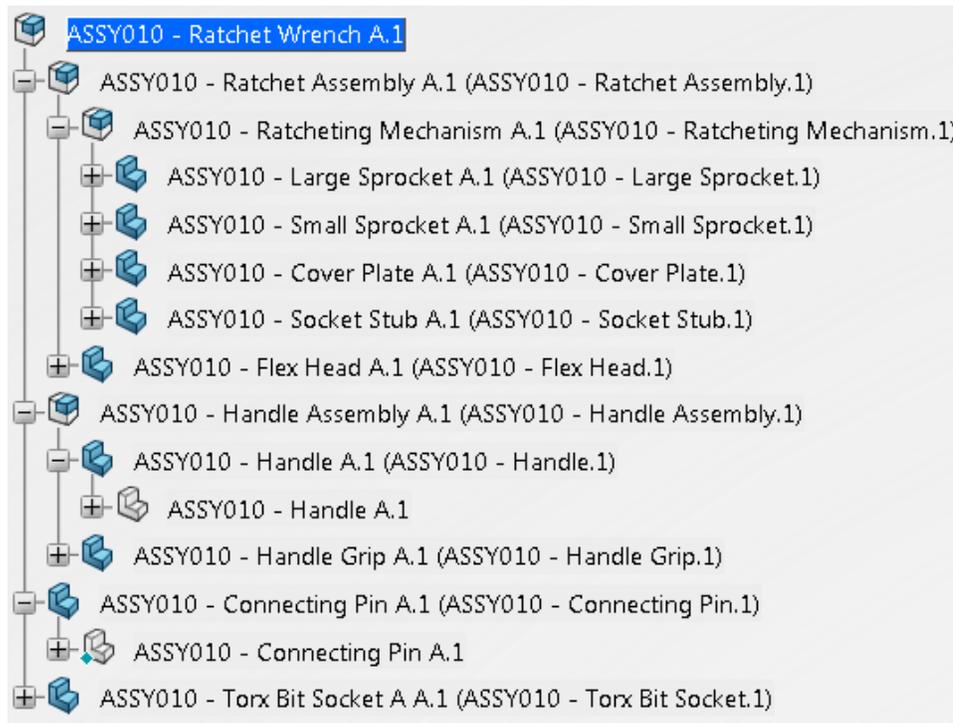


Either one of the torx bit sockets would have worked for this assembly, but you would need to know how to use assembly constraints to move the first torx bit socket into the correct location.

There is still one problem. Notice the instance name of *ASSY010 - Torx Bit Socket A* is still *ASSY010 - Hex Socket Small.1*. This should be changed.

Right select on *ASSY010 - Hex Socket Small.1* in the specification tree and select *Properties*. The *Properties* window appears.

Change the *Instance Title* to be *ASSY010 - Torx Bit Socket.1*. The specification tree should now appear as shown.



You are going to take a moment to investigate a few things about the assembly. Looking at the tree you have two different icons next to the various components in the tree. As you work with assemblies, you will notice there are many different icons that appear next to the components, each one tells you something about the component. Sometimes the icons only differ by the color so you need to pay close attention to the specification tree. The two icons you currently have represent a part and a product.



Part



Product

As you go through this course, you will be studying the specification tree from time to time. The tree holds a lot of information about your assembly so you need to be very familiar with it.

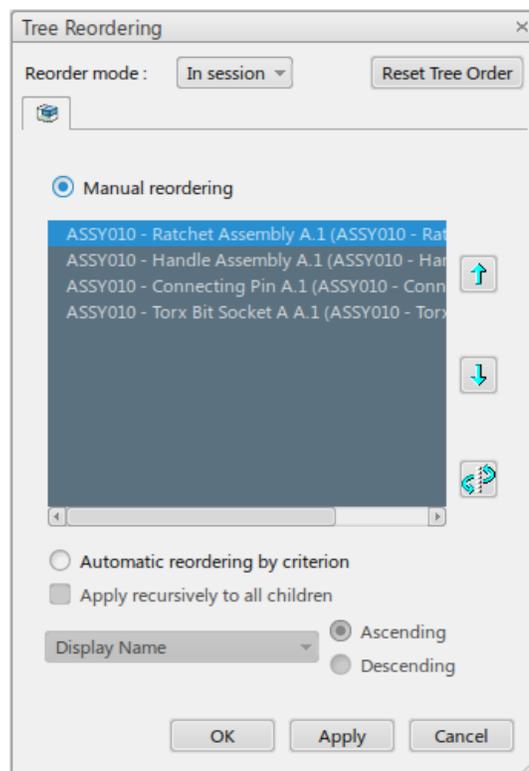
The other important aspect of an assembly you need to understand is the links that exist between the assembly and its components. When an assembly is stored it only contains pointers to the particular files and does not contain the data itself. This changes a little bit when other options are introduced.

Reordering the Specification Tree

With the assembly complete, you will look at two other options for assembly management.

Select the Tree Reordering icon.  It is located in the *Product Edition* section. This will allow you to reorder the tree. Since the components are not related to each other in terms of order of creation, as in part design, they can be reordered at will.

Select ASSY010 - Ratchet Wrench. The *Tree Reordering* window should appear. Notice only the parts and sub-assemblies appear that are under the *ASSY010 - Ratchet Wrench* assembly and not the parts that make up the sub-assemblies. If you want to reorder the components of the sub-assemblies, you can either reorder them while having the particular sub-assembly opened or you can select that sub-assembly instead of selecting *ASSY010 - Ratchet Wrench*.



The up and down arrows allow the highlighted part to be moved higher or lower in the tree respectively. The third icon moves the selected part to another selected location. This could be useful to help organize a large assembly by reordering components to be listed together in the specification tree. It is nice to know you do not have to insert all of your components in a specific order, you can always reorder them later.

Move the ASSY010 - Handle Assembly to the top of the tree and select Apply when done. This can be done by selecting the *ASSY010 - Handle Assembly* and then selecting the up arrow until it is on the top of the list. The tree will be reordered when *Apply* is selected. As you can see, it is fairly simple to reorder components of the assembly. If you have a large number of components, it may be better to use the third icon to position the component at a particular location without having to select the up or down arrows multiple times.

Select the *ASSY010 - Torx Bit Socket* and select the **Selection** icon in the window.



Select the *ASSY010 - Handle Assembly*. Notice the *ASSY010 - Torx Bit Socket* appears before *ASSY010 - Handle Assembly* now. This is a quicker way to move elements further up or down in the tree.



Move the *ASSY010 - Torx Bit Socket* back to the bottom of the tree.

Turn on the *Automatic reordering by criterion* option and select *Display Name* from the drop down menu. This will order the tree by name.

Select the *Ascending* option and select *Apply*. The tree instances are reordered to be alphabetical as shown.



Select **OK**.

Select the **Tree Reordering by Drag and Drop** icon.  The icon is active.

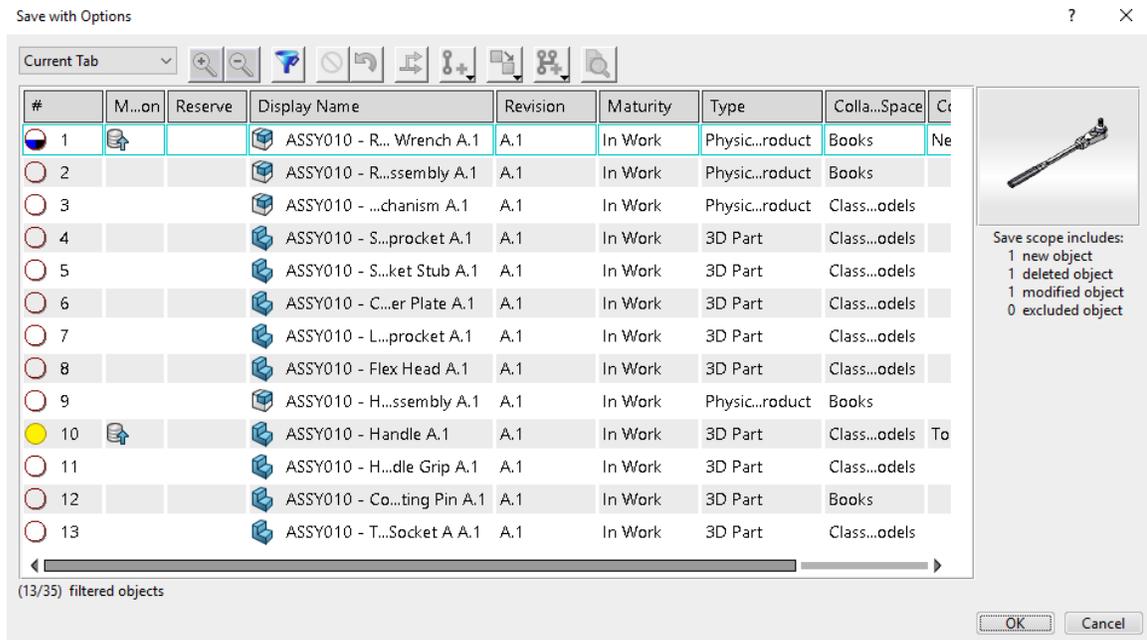
Select the **ASSY010 - Connecting Pin**, drag it below the **ASSY010 - Handle Assembly** in the tree and release it. The tree is reordered as shown.



Select the **Tree Reordering by Drag and Drop** icon again.  It is deactivated.

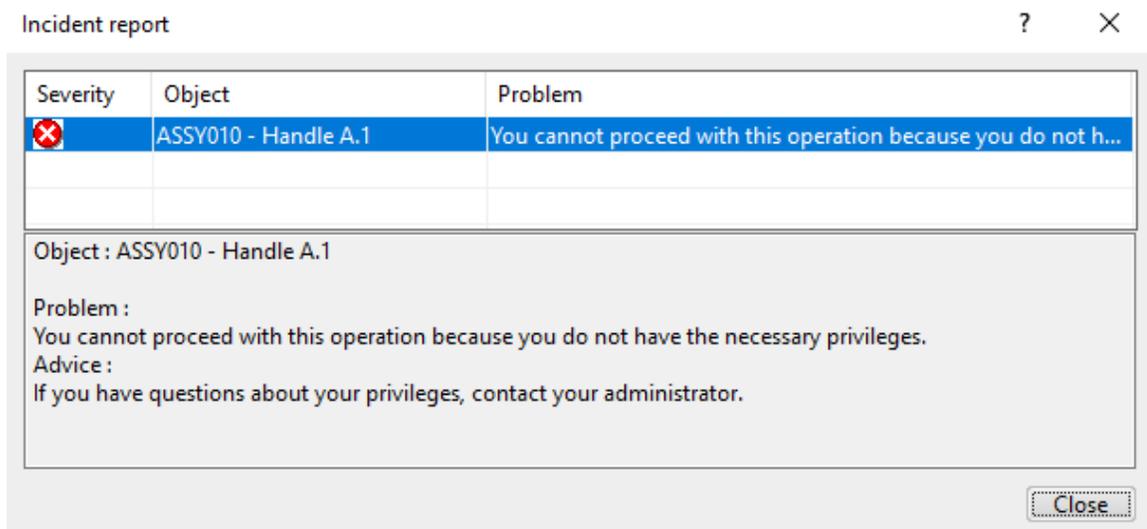
This provides an easy way to reorder straight from the specification tree.

Select the **Share** icon and select **Save with Options**.  You will have to select the expansion arrow next to the **Save** option to get to the additional save options. The **Save with Options** window appears.



Notice the **Handle** again is flagged as being modified since you had unhid the geometrical set. You will receive an error if you try to save this part.

Select **OK**. An **Incident Report** window appears. This is because you don't have access to save over the **ASSY010 - Handle**.



Select **Close**. You are returned to the **Propagate** window.

Select the **ASSY010 - Handle** in the list and select the **Exclude** icon.  It will now be excluded from the save.

Select OK. The models are saved.

Close the model. A *Close* window appears.



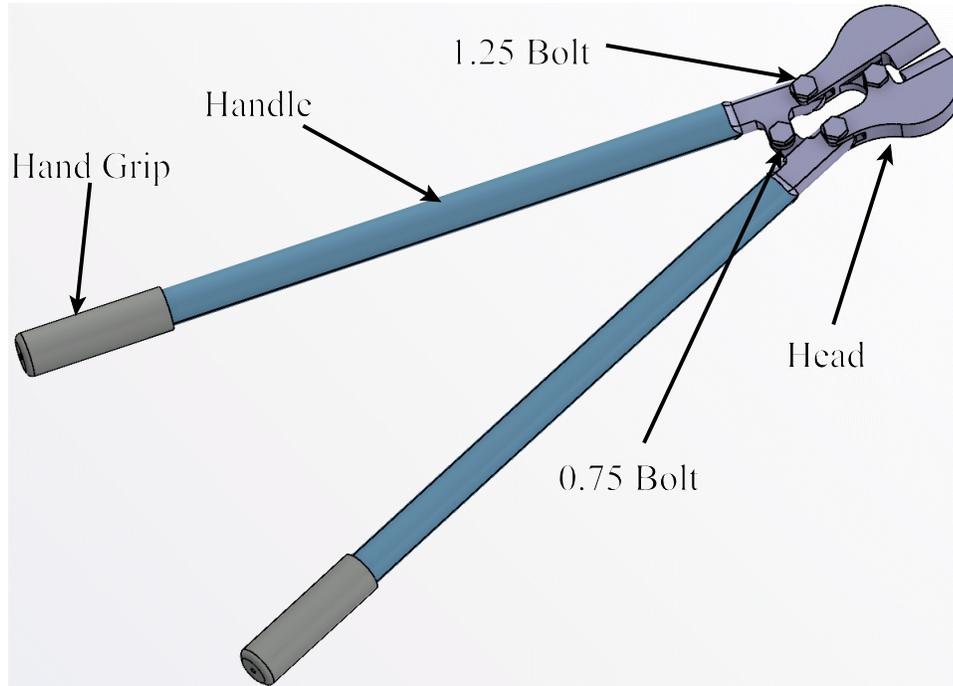
This is only because the Handle is modified, but cannot be saved.

Select No. The model is closed. If you were to reopen it, you would find the geometrical set in the *ASSY010 - Handle* model will be hidden again, since you could not save over it.

Constraining and Manipulating Parts Review

This exercise will review the use of the various assembly constraints while introducing some additional options to improve efficiency.

The first assem

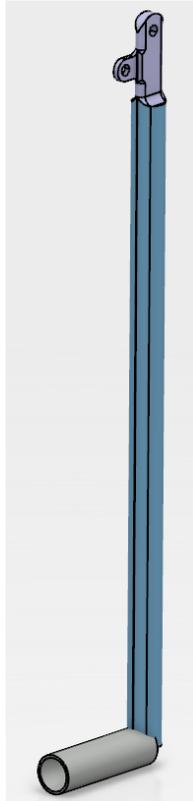


For any parts that are connected together and will not be moving independently, such as the hand grip and the handle, it is usually a good idea to put them into a sub-assembly. The two jaws of the head will move independently of each other, so it is not advisable to put them into a sub assembly.

Start a new product called ASSY040 - Handle Assembly. You should close all other documents and then start a new product. Make sure you are in the assembly workbench before continuing.

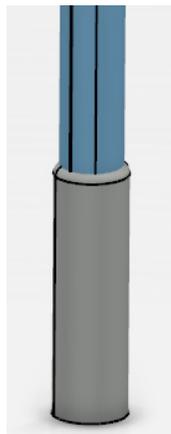
Insert the ASSY040 - Handle document into the assembly and select in the display to set the position of the part. This can be done the same way you did it in the previous section.

Insert the ASSY040 - Hand Grip into the assembly. Notice the hand grip and the handle do not insert into the correct location. This is common unless you build your parts where they would appear in the assembly.



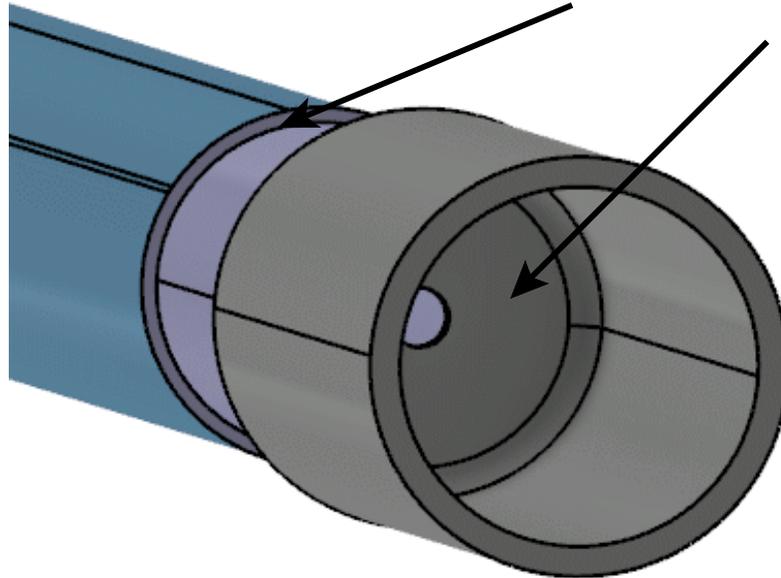
Create a fix in space constraint on the Handle. The part will not be allowed to move.

Create a coincidence constraint between the center lines of the Handle and the Grip. Set the orientation *Preference* to be *Undefined*. You may have to select the inside cylindrical surface of the handle in order to select the center line. The center lines are snapped together. The grip may be in the wrong orientation.

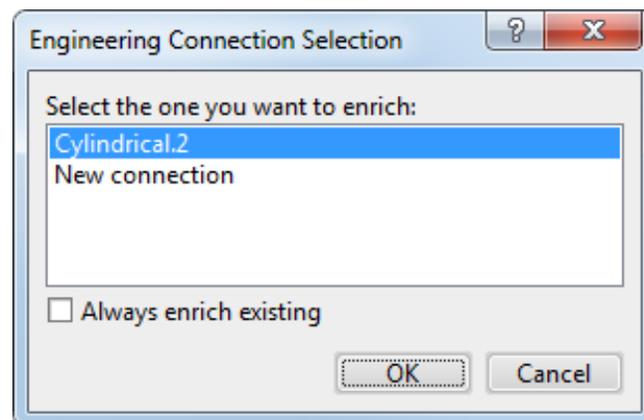


Select the **Engineering Connection** icon and select the two faces as shown. 

You may want to manipulate the models around in order to select the faces.



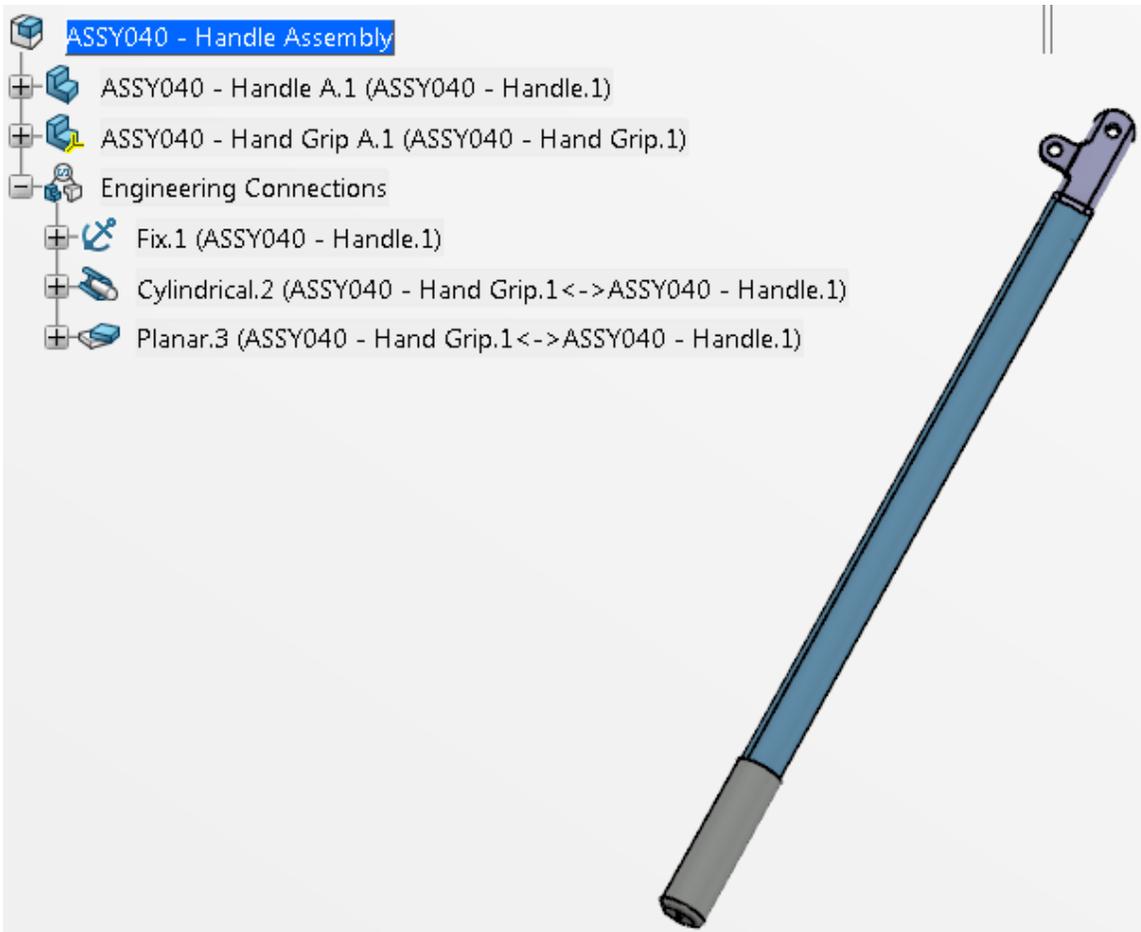
The *Engineering Connection Selection* window appears.



Select *New connection* and select *OK*. The *Engineering Connection Definition* window reappears.

Be sure the *Preference* is set to be *Opposite* and select *OK*.

The parts are snapped together as shown.



This completes the assembly.

Save and close your assembly. You will reuse your ASSY040 - Handle Assembly to complete the ASSY040 - Bolt Cutters assembly.

Start a new product called ASSY040 - Bolt Cutters. Remember, it is always a good idea to give the new part or product a unique, descriptive name right away. Don't forget to save your assembly now and throughout the exercise so if something happens you will not have to start all over.

Insert your ASSY040 - Handle Assembly twice. Both handle assemblies will be inserted in the same place, so there will only appear to be one inserted. However, if you look in the specification tree you will see there are two. *Note: You may need to wait a couple minutes for the server to update before your Handle Assembly will appear in the search.*

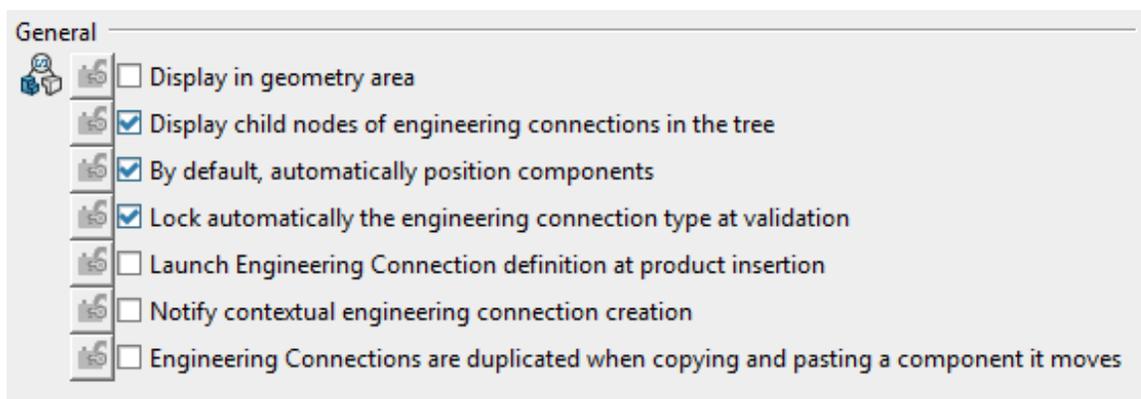
Move the second instance of the handle assembly so both are visible. This can be done by either selecting it in the tree, and dragging one of the box lines of the manipulation bounding box or by using the Manipulation icon.

Select the Engineering Connection icon and select the first instance of the handle assembly in the tree.  The *Engineering Connection Definition* window appears.

Change the Type to be Fix in space and select OK. This will fix the assembly in space.

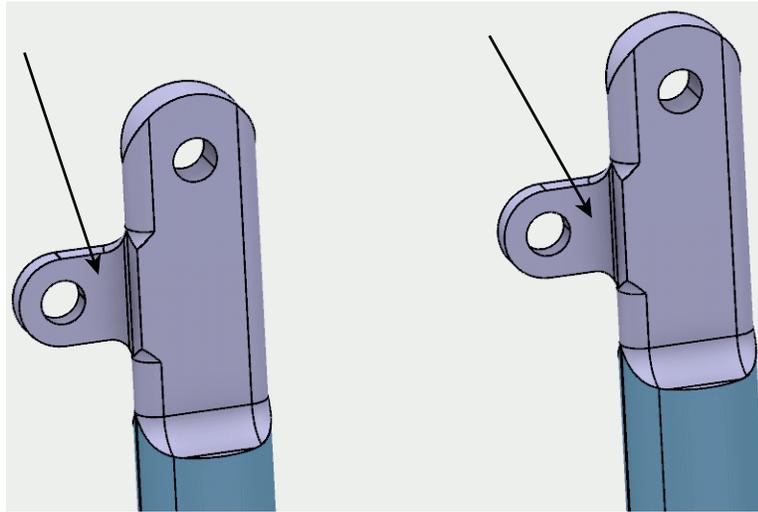
Before continuing, a setting will be changed in order to facilitate better updating of the connections.

Go to **Me** , **Preferences, Legacy Preferences, Mechanical, Assembly Design and the Engineering Connection tab.** Turn on the **By default, automatically position components** option as shown.



Select OK. When creating connections, this will allow CATIA to decide which part will be moved into position. When multiple connections are created, this allows more flexibility in updating the connections to satisfy all of the constraints.

At the top of the handle, there is a pivot point that is split in half as shown below. These two faces are going to be constrained together.



Select the **Engineering Connection icon**.  The *Engineering Connection Definition* window appears.

Select the two faces as shown above. The two handle assemblies move to get the two faces in position where they can be in coincidence with one another. You should note the two handle assemblies did not align themselves with the centers of the hole. Next, the orientation needs changed so the two faces will be facing each other.

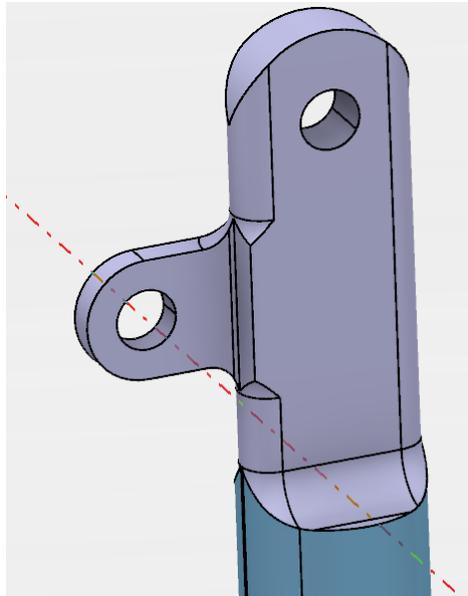
Change the Type to be *Coincidence* and the orientation Preference to *Opposite*. This will make the two faces face each other. You can also change the orientation by selecting one of the green arrows on the faces.

Select *OK* when finished. This may disorientate the two handles dramatically. Not to worry.

You may be wondering why a contact constraint was not used. It would make sense, especially when the two faces are contacting each other. The difference is, a contact constraint means that the two faces are locked together in the physical world, such as glued, welded, or bolted. That is not the case in this scenario. The two handles need to be free to rotate. It does not mean that a contact constraint will not work. Other applications within CATIA V6 will use the constraints that are applied to perform other analysis and simulations, and a contact constraint may be confusing. By defining the constraints correctly here, the downstream users will not have to re-define them later.

Select the **Engineering Connection** icon.  The *Engineering Connection Definition* window appears.

Select the **centerline at the inside of the pivot hole as shown below**. Moving the mouse cursor to the inside of the pivot hole will bring up the centerline. Either pivot hole can be selected. It can be difficult to select the centerlines at times, but if there is a curved surface available to select that is associated with the centerline, you can select on the curved surface and it will select the centerline. In this case you may find it easy to select on the curved surface of the hole and you will see the centerline appear as you select.



Select the **other centerline to align the two pivot holes**. The *Engineering Connection Selection* window appears.

Select **New connection** and select **OK**. The two handles will snap together. It is very likely that they are rotated in a way which is invalid for the actual operation of the bolt cutters. There are no constraints prohibiting the two handle assemblies from rotating around the centerline.

Change the orientation *Preference* to be **Undefined** and select **OK**.

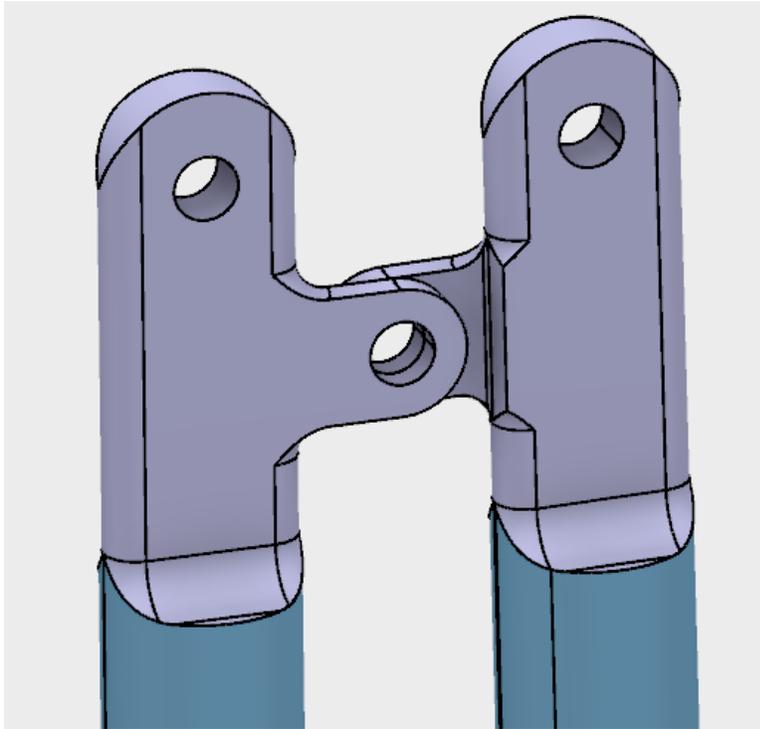
Update the engineering connections. Now it is time to move the two handles with respect to their constraints.

Select the **Manipulation** icon.  This time the two handles are going to be rotated about an arbitrary axis.

Select the **Drag Around any Axis** icon.  Now a centerline needs to be selected.

Select the **centerline of the two pivots**. This will be the same centerline you selected when creating the coincidence constraint.

Rotate one of the handles around to appear as the picture below. Select *OK* when done. This will keep things oriented in a manner that will make it easier to finish building the bolt cutters. You will only be able to rotate *ASSY040 - Handle Assembly.2* since *ASSY040 - Handle Assembly.1* is fixed.

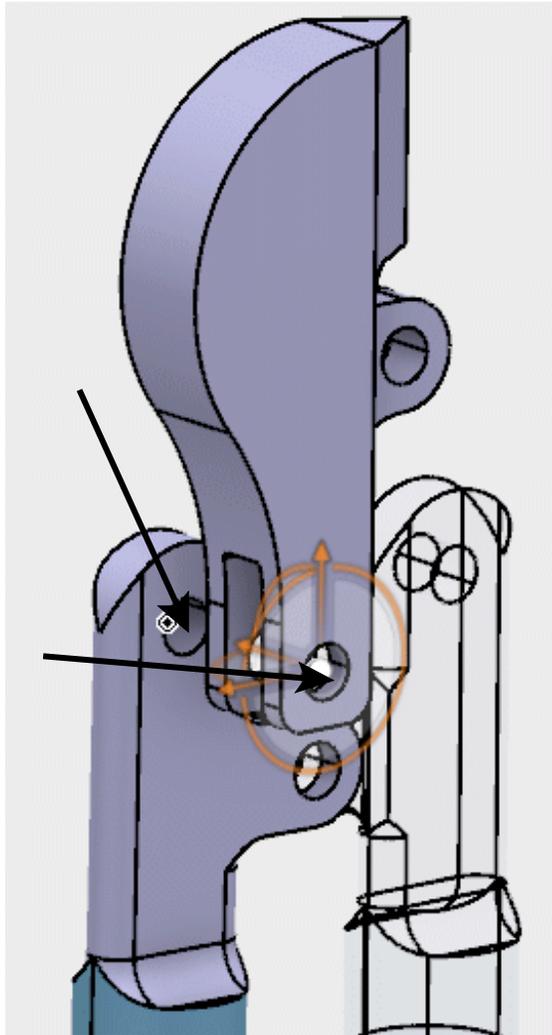


This would be a good point to save your document before continuing.

Insert the *ASSY040 - Head* document into the assembly. Go ahead and move the head to a more suitable location at the top of the handles. You may have to zoom out in order to see the head.

Select the **Engineering Connection** icon.  The *Engineering Connection Definition* window appears.

Apply a coincidence constraint between the center lines of the hole in the Head and the hole in the Handle as shown. This will make sure that the two center lines will always remain aligned with each another.

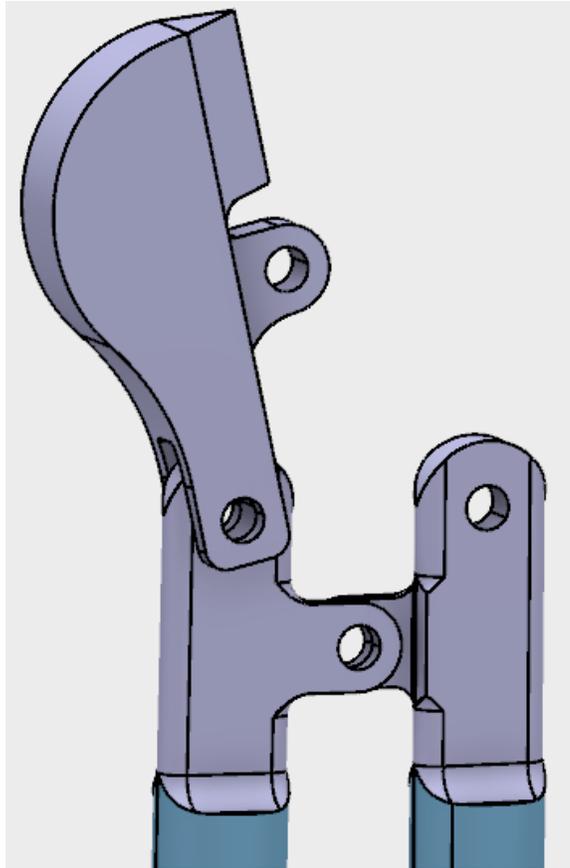


Set the orientation *Preference to Undefined* and select *OK*. It is rare that you will want centerline constraints to have an orientation.

Apply a coincidence constraint between the two faces that will move the head into the proper location. You will want to create a new connection and make sure the orientation *Preference* is defined correctly. The end product should appear similar to the one shown below.

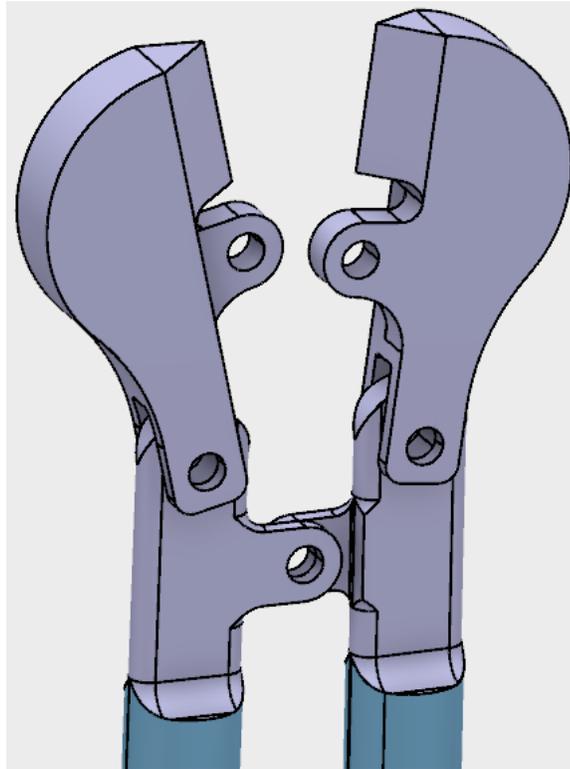
Update the connections.

Manipulate the head using the manipulation icon to look similar to the diagram shown below. This will be the same process as investigated earlier. Make sure you use the rotate about any axis, and then select the pivot hole for the head.



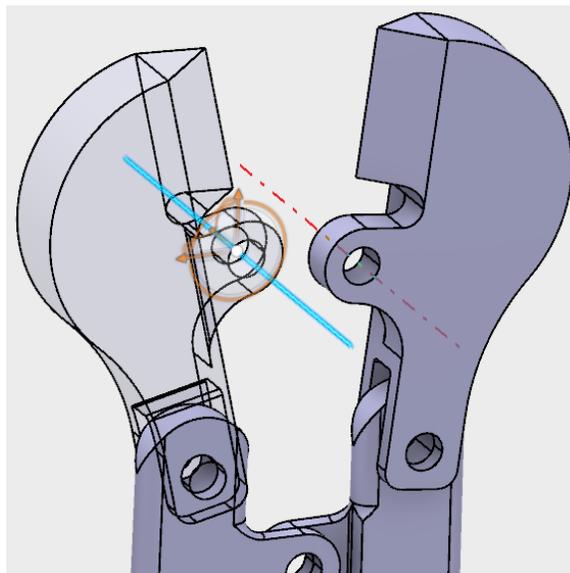
Insert another ASSY040 - Head into the model.

Add coincidence constraints to the new head and manipulate it so it looks similar to the diagram shown below. Remember, it is imperative the head be in the right direction. You will want the constraint between the faces to control the orientation with the centerline constraint having its orientation undefined.



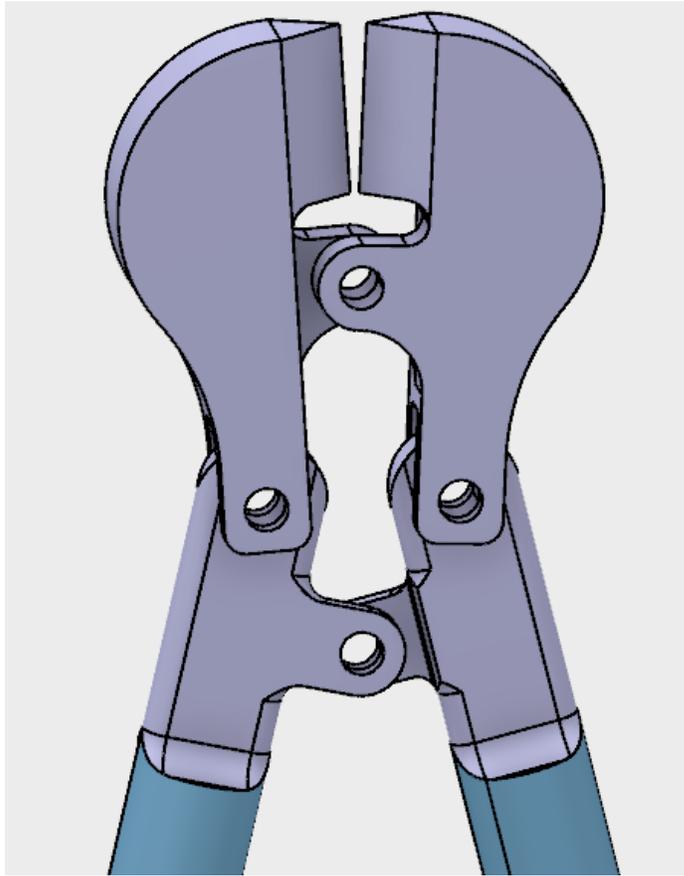
Select the Engineering Connection icon.  The *Engineering Connection Definition* window appears.

Create a coincidence constraint between the two centerlines shown below.



Set the orientation *Preference to be Undefined* and select *OK*.

Update the connections. The assembly should appear as shown.



Select the **Manipulation** icon, then select the **Drag Around Any Axis** option in the *Manipulation Parameters* window and ensure the *With respect to constraints* option is selected.  This will allow the parts to rotate about any given axis that is selected with respect to any constraints placed on the assembly.

Select the **inside of the pivot between the two handles** and then select and drag one of the handles. You will have to select the handle that is not fixed. Notice the whole assembly moves. Note, this is not kinematics because the assembly can be moved beyond the physical limits but this is a good way to test the model to insure the entire assembly is constrained correctly.

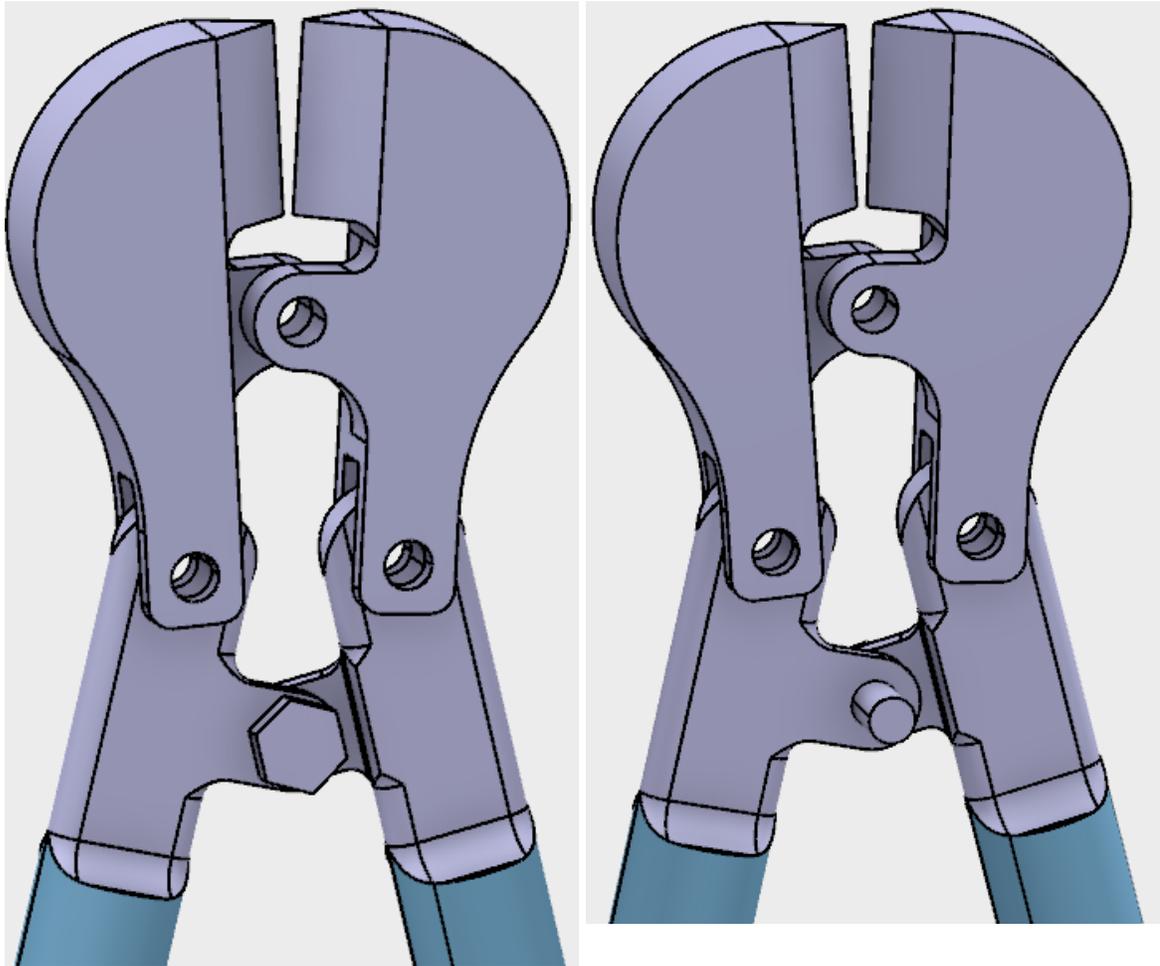
Select **OK** when finished. This exits the manipulation window.

Next, bolts will be added to complete the assembly.

Insert the **ASSY040 - 1.25 Bolt** document into the assembly and move it close to the **pivot location**. This bolt will go in the pivot between the handles.

Add a coincidence constraint between the centerline of the bolt and the centerline of the pivot between the handles without any orientation. Remember, this will align the bolt with the hole. Be sure to select *OK* when you are done.

Add a contact constraint, as a new connection, to put the bolt into the pivot hole. This can be accomplished by adding a contact constraint between the outside of the handle and the bottom of the bolt head. Your assembly should appear similar to the diagram shown below. This time a contact constraint was wanted due to the bolt being tight against one of the handles.

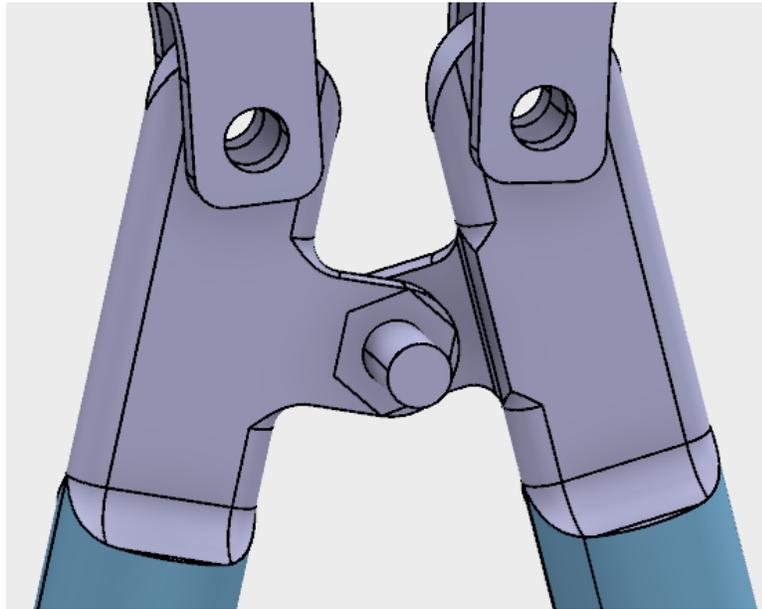


The bolt sticks out too far. You need to modify the bolt to be 0.75 inches instead of 1.25 inches long, then save it as a separate part. You will do that next.

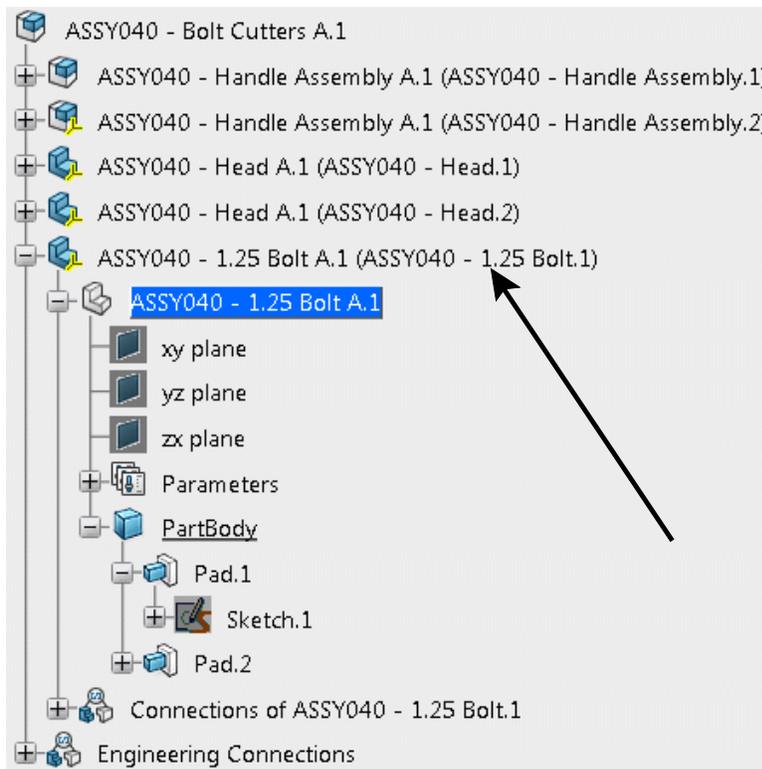
Expand the *1.25 Bolt* branch until you see the *PartBody*. This will allow you to enter the Part Design workbench through the assembly.

Double select *PartBody* with the first mouse button. This takes you to the Part Design workbench allowing you to perform part design tasks. This is similar to what you did when you created a new part in the assembly. This time you are going to modify a part.

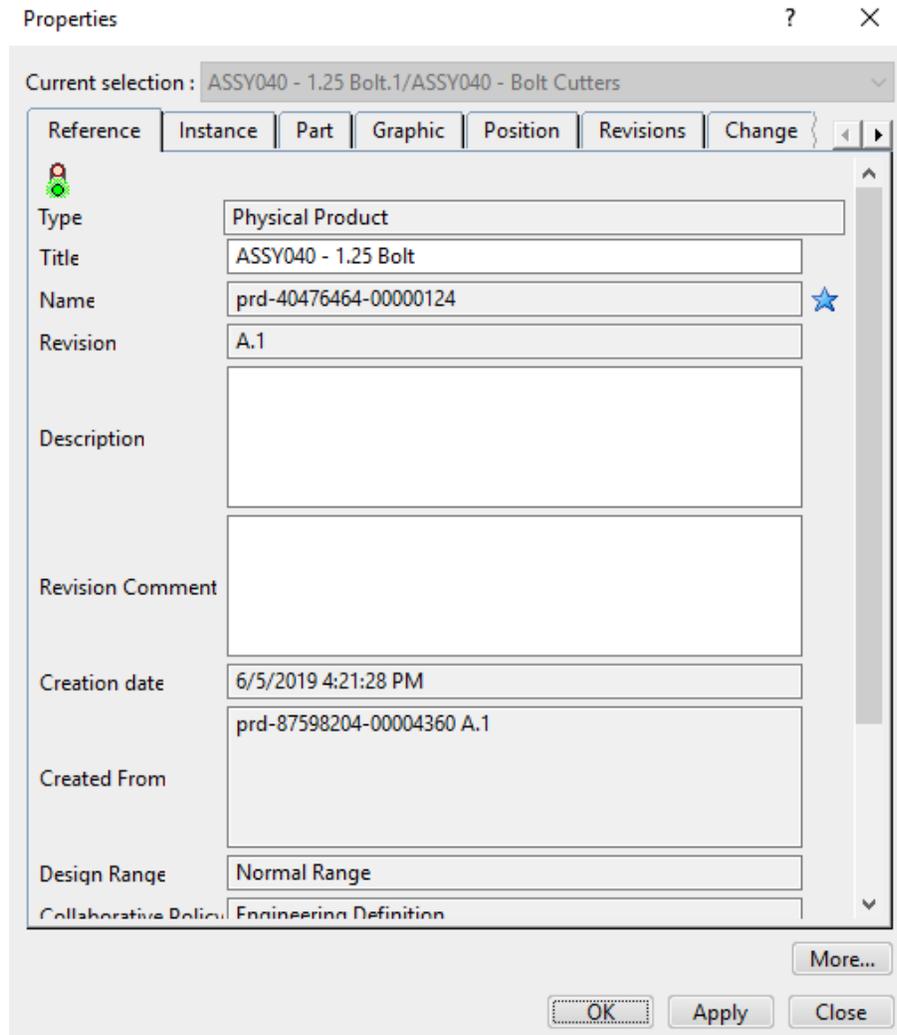
Change Pad.1 to have a Length of 0.75 inches. Notice the bolt becomes shorter but the head of the bolt moves instead of the end of the bolt, this is determined by the original design of the bolt.



Right click on the 1.25 Bolt 3d part branch in the tree as shown below. Since you do not want this new bolt to be referred to as a 1.25 Bolt, you are going to change its name before you save the modified part.



Select the **Properties** option. A *Properties* window appears similar to the one shown below.



Change the *Title* to be ASSY040 - 0.75 Bolt. On the *Instance* tab, change the *Instance Title* to be ASSY040 - 0.75 Bolt.1. Select *OK* when finished.

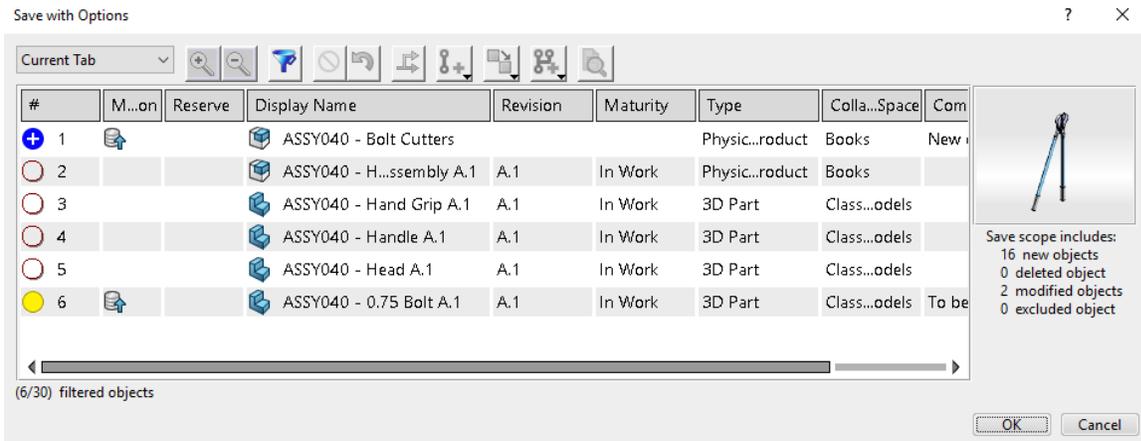
Change the name of the 3d shape to be ASSY040 - 0.75 Bolt as well. It should appear as shown.



Double select on ASSY040 - Bolt Cutters. This will return you to the main assembly.

Update the connections. The constraints are updated positioning the bolt in the correct location. Remember, you used a contact constraint between the bottom face of the bolt head and the outside face of the handles. You can collapse the *ASSY040 - 0.75 Bolt* branch if you want.

Select the **Share** icon and select **Save with Options**.  The **Save with Options** window appears.



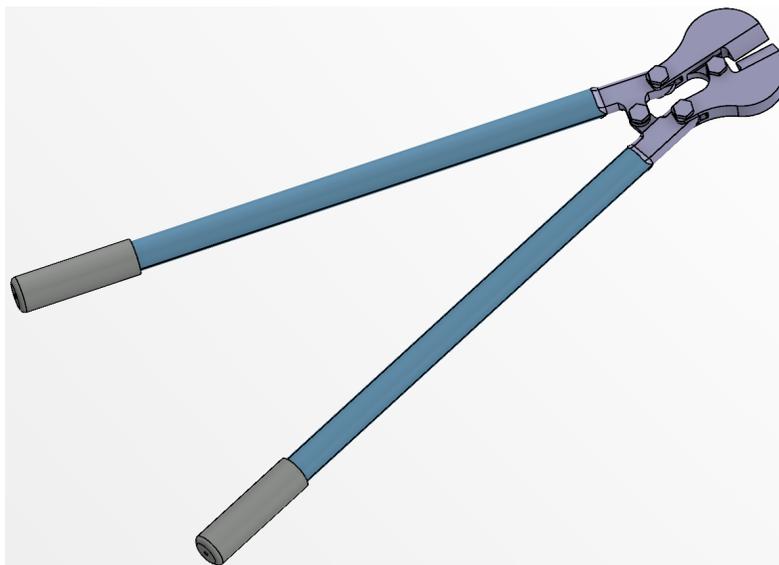
Notice the ASSY040 - 0.75 Bolt one of the Class Models which is read only. This is because you created it from the 1.25 Bolt. If you wish to save the modified part, you will have to save it as a new part.

Select the **ASSY040 - 0.75 Bolt** and select the **Save as New** icon.  The symbol next to the part changes to show it will be saved as a new part.

Select **OK**. The assembly is saved.

Insert the 1.25 Bolt and constrain it to the pivot between the two heads. This is done the same way as you constrained the previous bolt.

Add two more bolts and constrain them into the proper locations. This will finish the assembly.

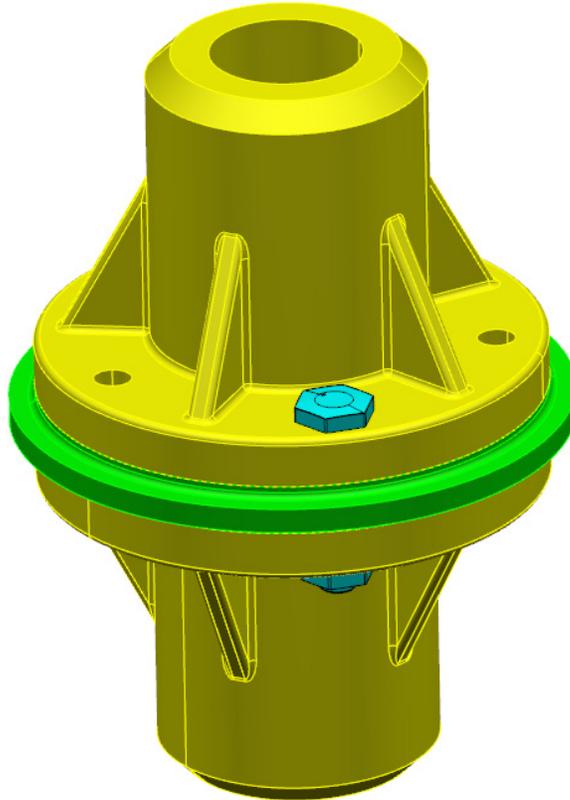


Save and close the document.

Assembly Pattern

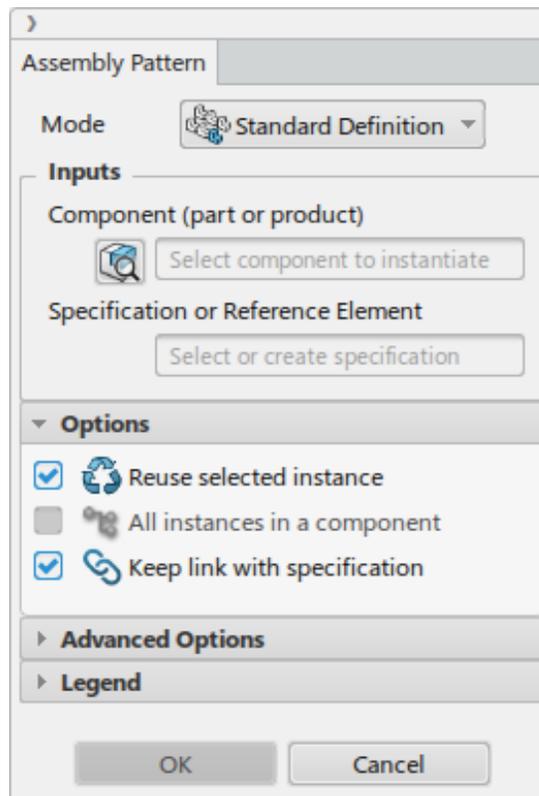
The assembly pattern option allows you to use an existing pattern within a part to help position and constrain components. This is very useful for constraining fasteners to patterns.

Open the ASSY090 - Assembly Pattern product. You should see a couple of casings attached with a bolt and nut. You will place the rest of the fasteners in the assembly.



Expand the ASSY090 - Casing.1 branch to the PartBody level in order to see the patterns available. The *CircPattern.2* is the one you are going to reuse.

Select the Assembly Pattern icon.  The Assembly Pattern pane appears on the right side of the display. You may have to expand it. This icon will allow the user to reuse any pattern used in a part design.

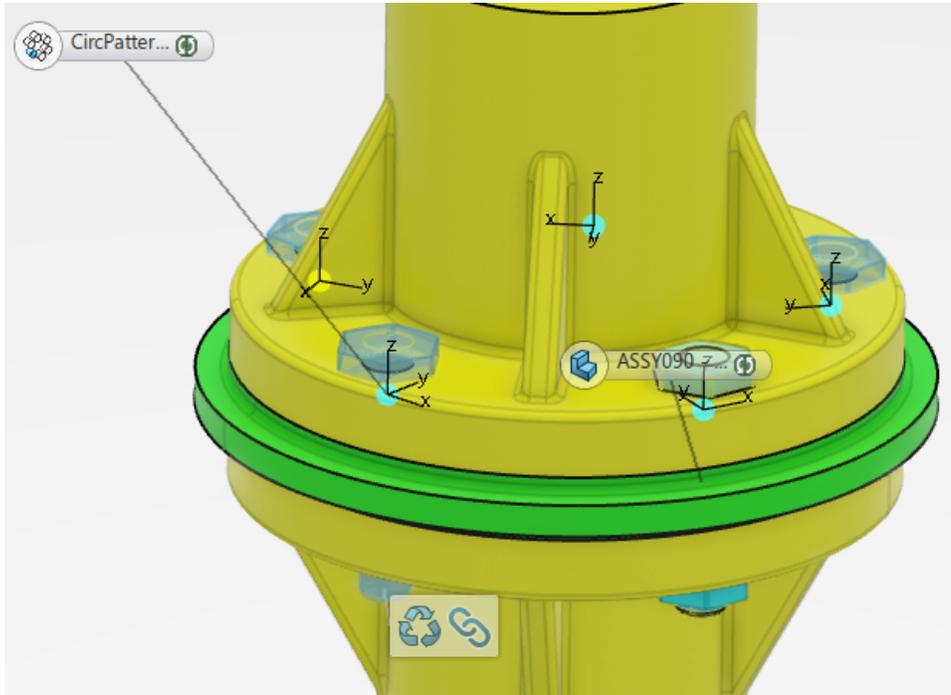


<i>Mode</i>	Specifies whether the pattern will be linked or if existing engineering connections will be copied to each instance
<i>Component</i>	Specifies what will be patterned
<i>Specification or Ref...</i>	Defines the pattern to be used
<i>Reuse selected instance</i>	The original component is used and others are added
<i>Keep link with spec...</i>	Updates the number of instances as the pattern is modified and creates a link back to the pattern

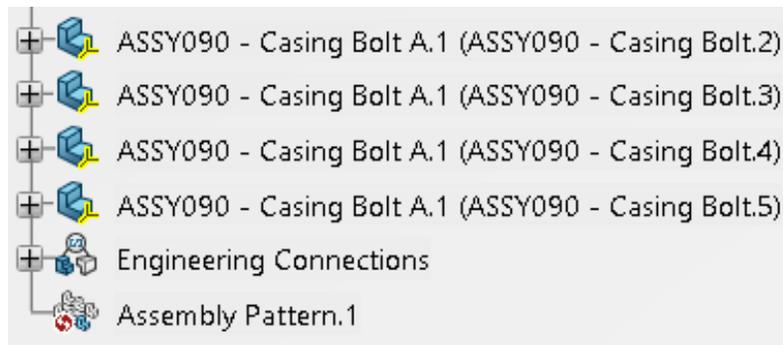
Select the ASSY090 - Casing Bolt.1 in the tree to define the Component.

Select CircPattern.2 from the specification tree to define the Specification.

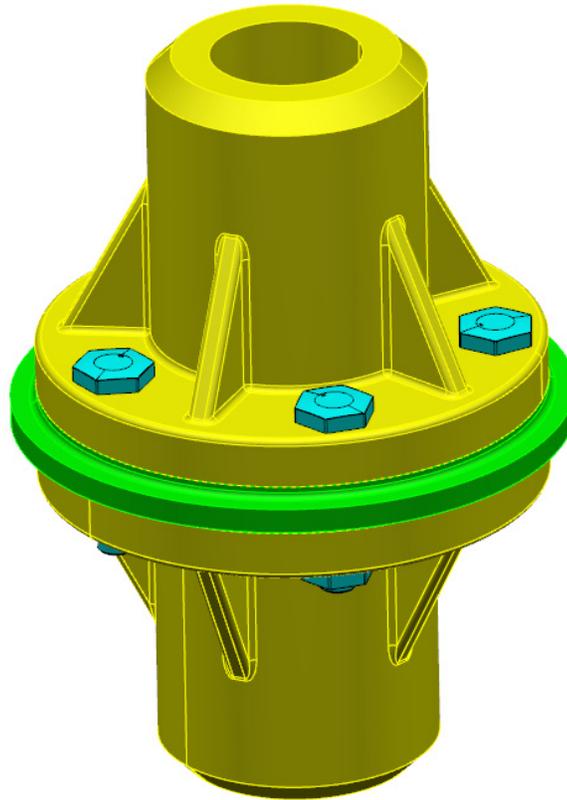
The selections appear in the display along with a couple of icons.



Make sure the *Reuse selected instance* and *Keep link with specification* options are on and select in the display to finalize the pattern. Four more instance of the bolt appear in the tree along with *Assembly Pattern.1*.



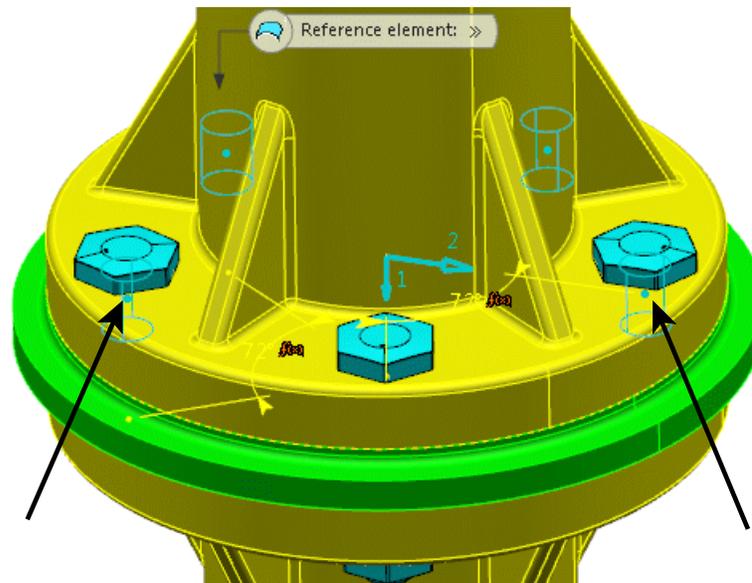
The bolts should appear in the display as shown.



You should note that the original bolt was used and then four more instances were created.

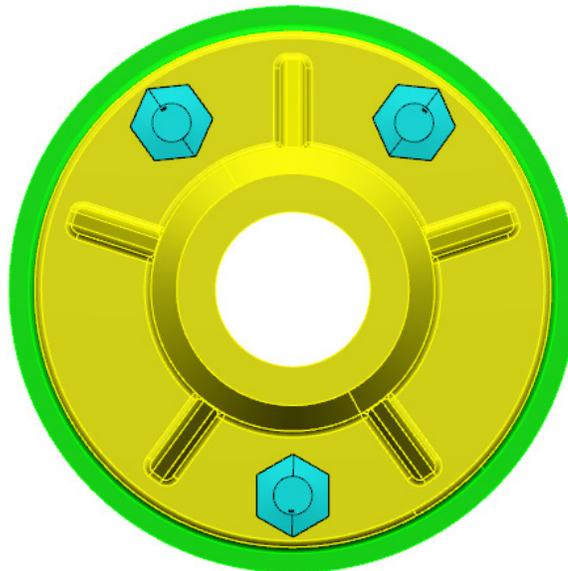
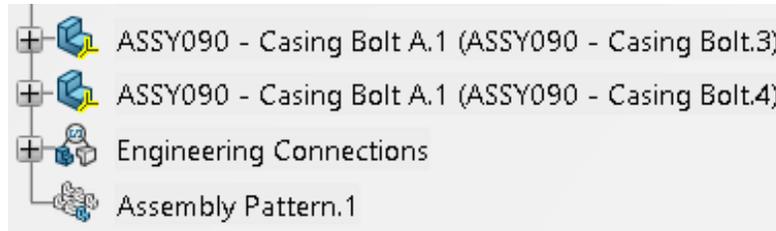
Double select on the *PartBody* of *Casing.1*. This makes that part the active component.

Double select on *CircPattern.2* and remove the two instances shown below and select *OK*. Just select on the two dots as shown to deactivate the two locations. This will modify the pattern to have only three holes. You want to select the two locations on either side of the original hole.



Double select on the ASSY090 - Assembly Pattern product. This makes the top level assembly the active component.

Update the assembly. The number of instances of the bolt changes. This is due to the link to the pattern. The diagram shown below has the product rotated.

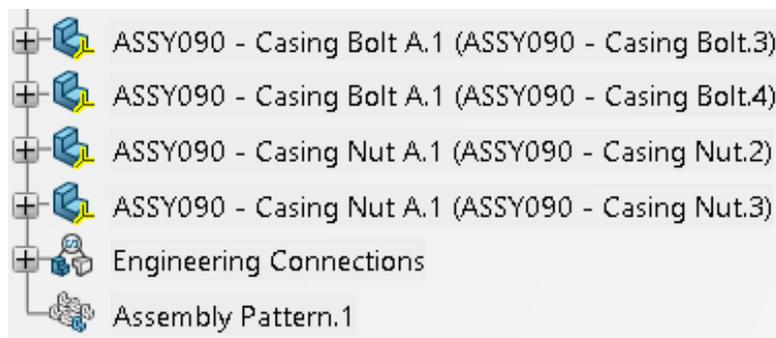


Select the Assembly Pattern icon.  The *Assembly Pattern* window appears.

Select the ASSY090 - Casing Nut.1 in the tree to define the Component.

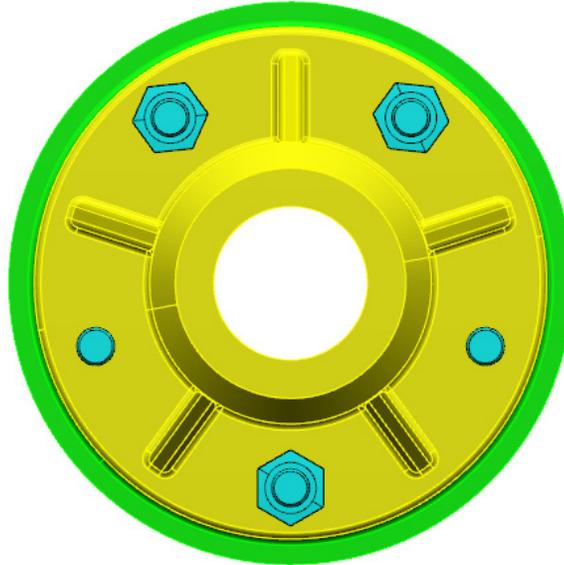
Select CircPattern.2 from the specification tree to define the Specification.

Turn off the Keep link with specification option and select OK. Two more instances of the nut appear in the tree, but no assembly pattern operation is created.



Modify the *CircPattern.2* to turn on the two holes you had deactivated earlier.

Activate the top level product and update the assembly. Notice the assembly pattern for the bolts updated to add in two additional bolts. However, since the link was not kept for the nuts, they remain unchanged.



Delete the two nut instances that were added previously and delete the assembly pattern in the specification tree. Notice the patterned bolts are automatically removed as well.

Now the pattern will be created with copied engineering connections.

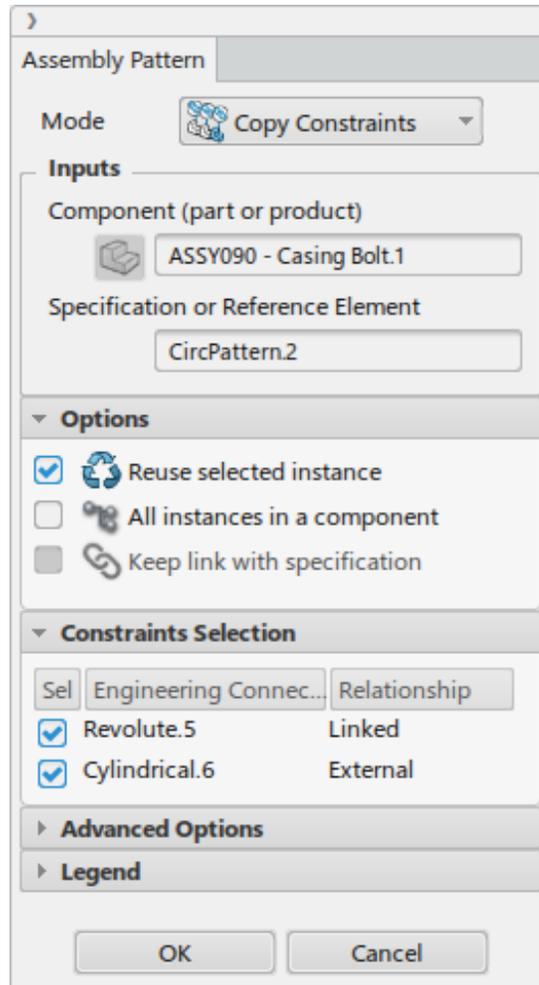
Select the **Assembly Pattern** icon.  The *Assembly Pattern* window appears.

Change the *Mode* to *Copy Constraints*.

Select the *ASSY090 - Casing Bolt.1* in the tree to define the *Component*.

Select *CircPattern.2* from the specification tree to define the *Specification*.

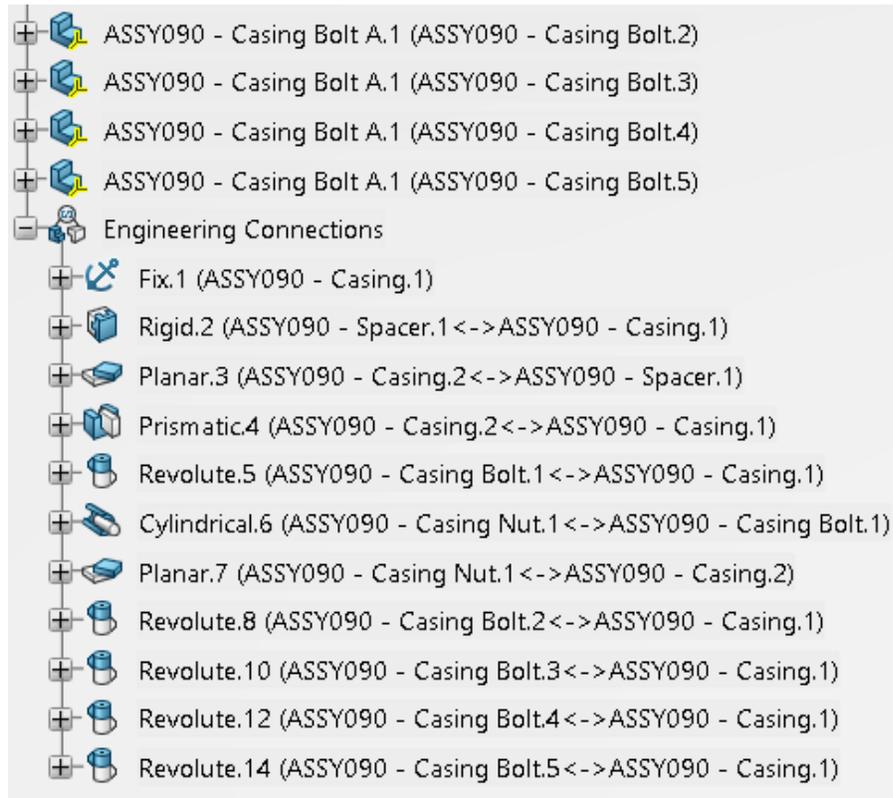
The window should expand as shown.



Notice the *Keep link with specification* option is automatically greyed out and the engineering connections for the bolt appear. In this case, only the *Revolute.5* is between the Casing and the Casing Bolt.

Turn off the check box next to the cylindrical engineering connection and select *OK*.
The bolts appear along with new engineering connections in the tree.

Update the assembly. Notice there is no assembly pattern in the tree. The bolts are positioned by the engineering connections.



Save and close the document.