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Introduction

CATIA Version 5 Assembly Design

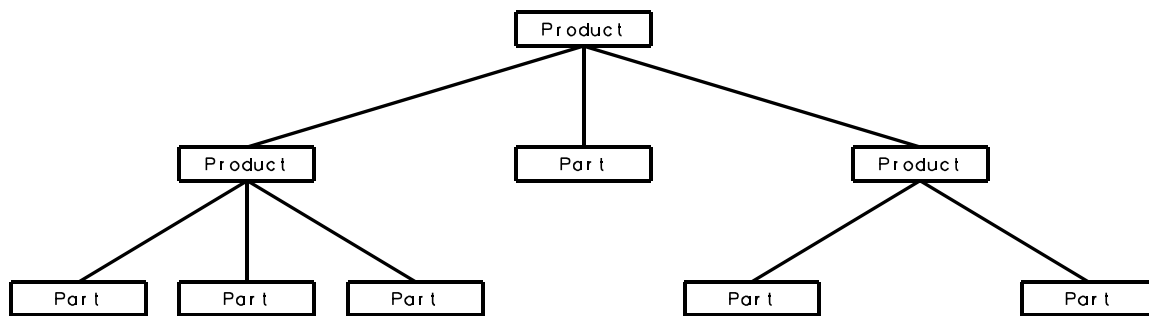
Upon completion of this course the student should have a full understanding of the following topics:

- Inserting models into an assembly
- Manipulating models in an assembly
- Constraining models in an assembly
- Using advanced methods to insert and constrain assemblies
- Analyzing assemblies for clashes and gaps
- Modifying assembly components and updating assemblies

Assembly Design

Very few completed designs consist of a single part. Usually, finished designs contain several to millions of individual parts to define them. This is where CATIA V5 Assembly Design is utilized. It allows individual parts and smaller assemblies of parts to be combined in order to make larger, more complete products. In CATIA V5 Part Design and Sketcher, you learned how to create parts. The primary objective of this class is to utilize those parts to build a complex assembly that can later be used in stress analyses, 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 in Assembly Design: 1) the overall assembly, 2) sub-assemblies, and 3) individual models. CATIA uses the word 'products' to refer to assemblies, and 'parts' to refer to individual models. You can use parts to create products. In turn, those products can be used to build other products. The diagram below represents the concept of the overall structure.



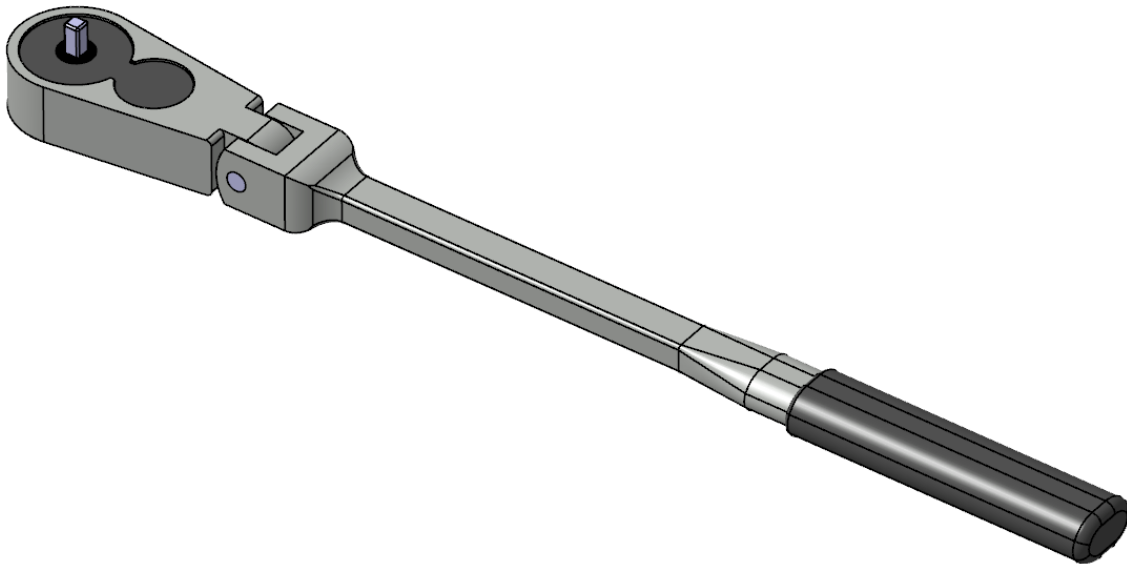
The product at the top is generally regarded as the assembly, whereas the two products that are underneath are typically regarded as sub-assemblies of this assembly. The main assembly could in turn be used to create an even bigger assembly at some other time, or the sub-assemblies could be used as sub-assemblies of a different assembly. With this concept in mind, be aware that an assembly can 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 which sub-assemblies a particular assembly will need. The other is to produce all of the parts first, then determine which sub-assemblies will be created.

Assembly Design

The first section of this manual will involve inserting, creating, and replacing various documents within the assembly, such as parts and other assemblies.

Inserting Documents

All of the assemblies created in this first section do not require constraints in order to properly position them. The first assembly you will build is a basic ratchet wrench.




Start a new product document. This can be accomplished by selecting the **New** icon and choosing *Product*. Ensure you are in the Assembly Design workbench before continuing. To switch to Assembly Design, select the *Start* pull-down menu and choose *Mechanical Design*, then choose *Assembly Design*.

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

Press the third mouse button on *Product1* in the specification tree and choose *Properties* from the contextual menu. The *Properties* window appears.

Ensure the *Product* tab is selected. This is where various information about the assembly, as well as individual parts, can be stored. Adding all of the information is not necessary, but it can be quite useful for down-stream users. In this exercise, only the *Part Number* field will be changed.

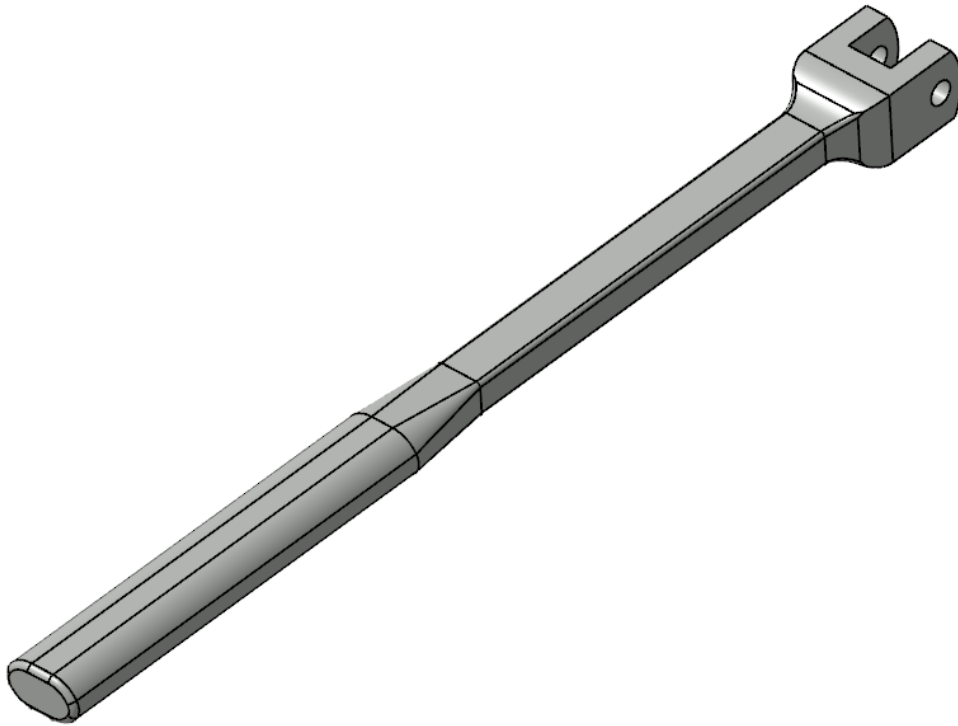
Change the *Part Number* field to Ratchet Wrench, then select *OK*. This gives the assembly a unique ID, which will prevent naming conflicts if it is added to a larger assembly.


Select the **Existing Component icon.**  It turns orange to indicate that it is active. Next, you must choose a product to insert the component into.

Select **Ratchet Wrench** from the tree. The *File Selection* window appears.

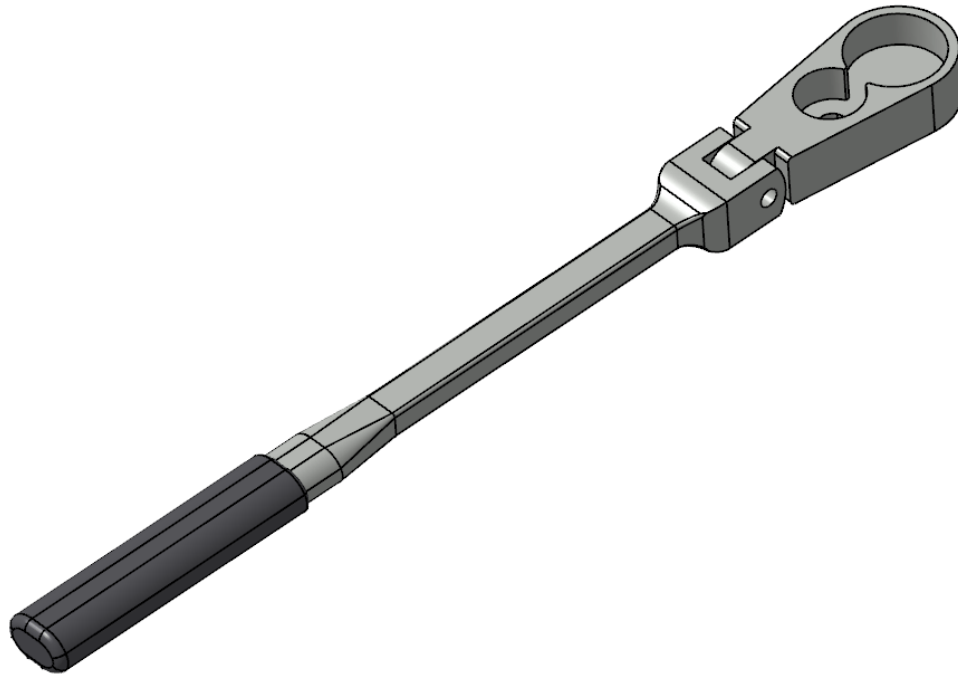
Browse to the *Assembly Basics* directory and select the **Handle** document. This will specify the document to be inserted. You can insert multiple documents at once, if desired.

Select **Open**. The handle is inserted into the product. The other components will be inserted to complete the assembly.



Select the **Existing Component** icon, then select **Ratchet Wrench**.  The *File Selection* window will automatically open to the *Assembly Basics* directory.


Select **Flex Head** and **Handle Grip** while holding the **Ctrl** key, then select **Open**. The two parts are inserted into the assembly, pre-positioned. The **Ctrl** key allows you to manually select which files you want, whereas the **Shift** key will select a range of files. Ensure all parts are shown before continuing.

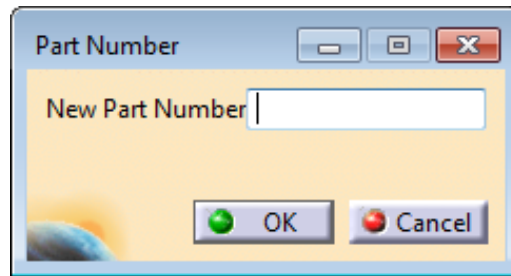


Sometimes components can be made into a sub-assembly. This can be done either before hand, as with the Ratcheting Mechanism that you are going to insert later, or on-the-fly, as you will do next.

Creating a New Product


When you create a new product within an existing 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 from a product, other than the fact that it is used within a higher-level product.

Select the **Product icon and pick *Ratchet Wrench*.**  This will insert a new product into the assembly. The *Part Number* window appears.



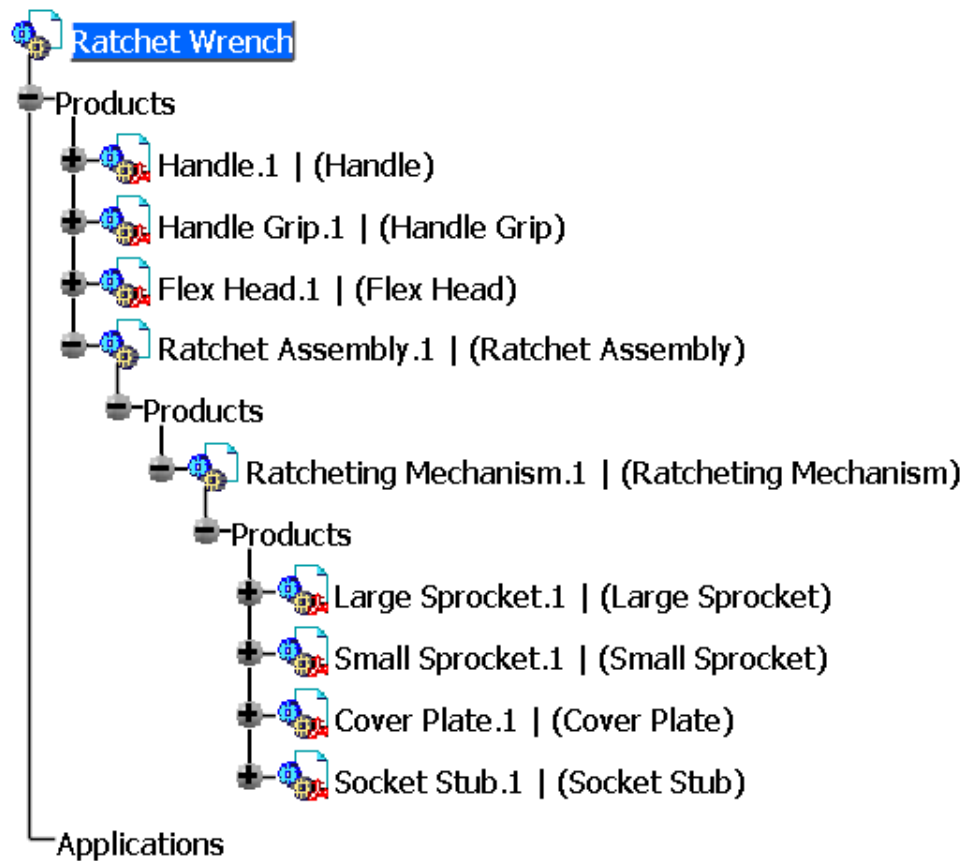
A part number must be assigned to the new product. Part numbers can either be generated by the computer, or generated on-the-fly. The option to change how new product part numbers are generated is under *Tools, Options, Infrastructure, Product Structure, Product Structure tab*, and *Part Number Manual Input*. Your system is set up for manual input of the part number. To ensure your ability to distinguish between the different products, you will name this one Ratchet Assembly.

Enter Ratchet Assembly in the window and select *OK*. This will change the instance name, allowing for better model management. Now that you have a nested product assembly, a product within a product, you need to make certain that you insert new and existing components into the proper products.

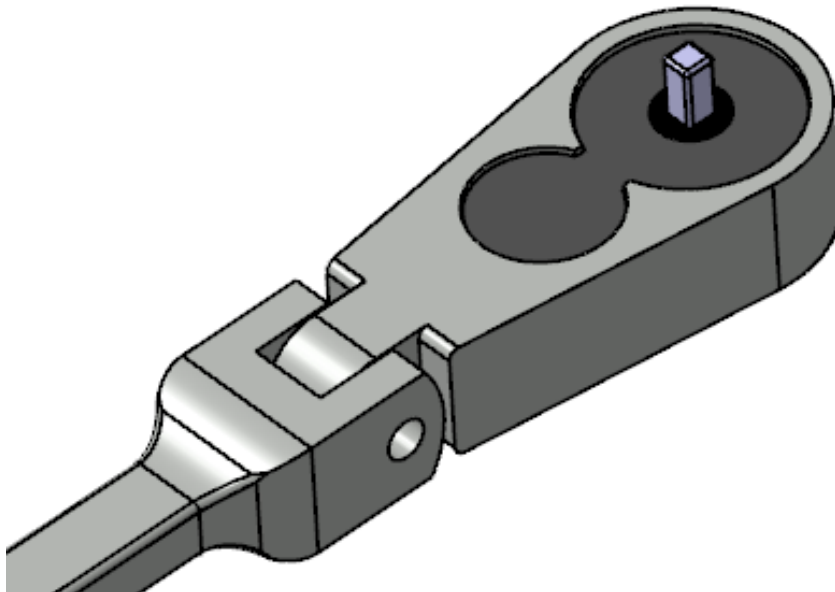
Select the **Existing Component icon and pick *Ratchet Assembly.1*.**  The *File Selection* window appears. This time, you will insert a subassembly.

Select **Ratcheting Mechanism, then select *Open*.** This will insert the subassembly into the product.

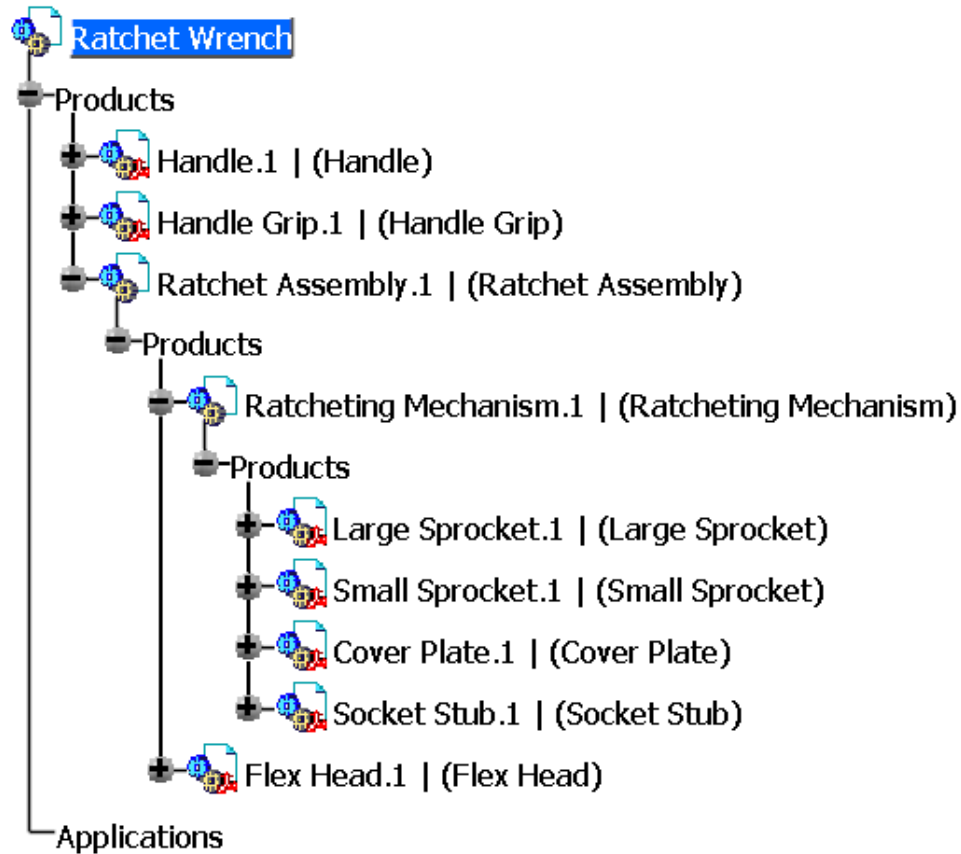
Expand the specification as shown.



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



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



This is another way of inserting parts or products into a sub-assembly. Dragging and dropping a component is the same as cutting and pasting it.

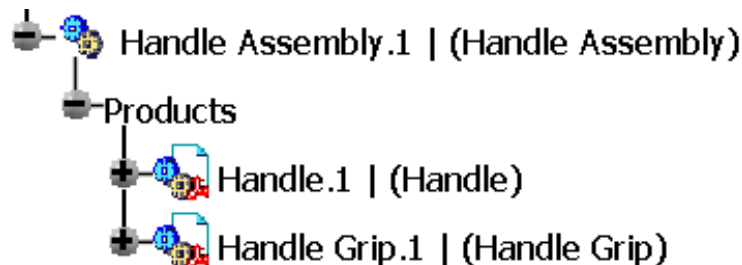
Creating a New Component

The meaning of the term ‘component’ is often confused. In normal operation, every entity that is in a product is a component of the product. That term is used whether the entity is a part, a product, or an actual component. The entity that is called a component acts just like a sub-product, except that it only exists within the main product. When you create a new product within the main product, it is actually a document that can be saved and opened independently from the main product. When you create a new component, it is not a separate document, and can only be accessed through the main product.

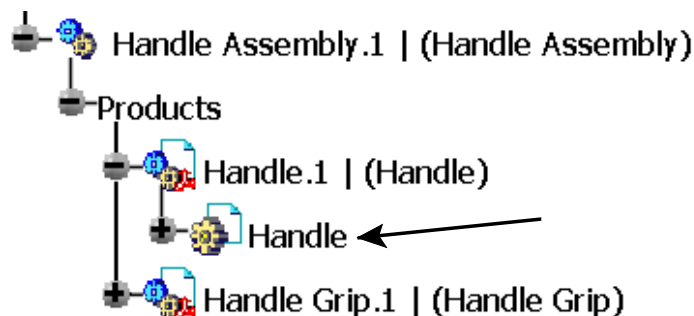
Select the **Component** icon and pick *Ratchet Wrench* from the tree.  The *Part Number* window appears.

Enter **Handle Assembly** for the *New Part Number* and select **OK**. The component is created.

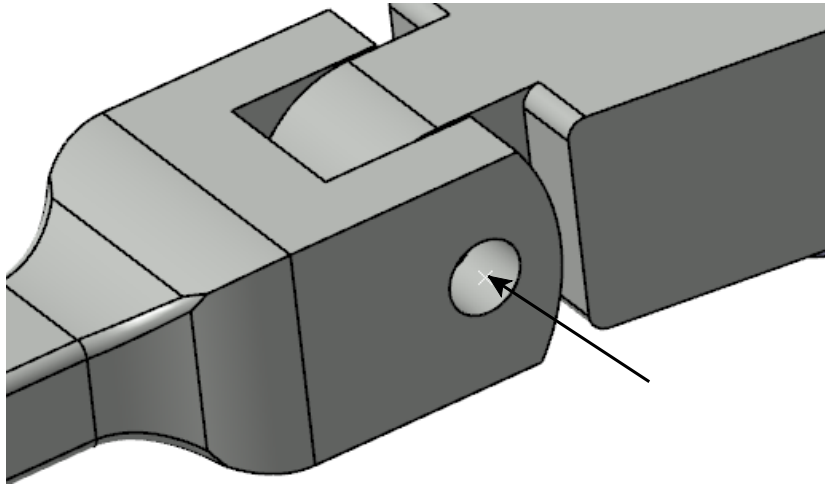
Drag both *Handle.1* and *Handle Grip.1* into the new component. It should appear as shown. The symbol in the tree for a component does not contain a sheet of paper like the parts and products. This signifies that it is not a separate document, but just a “grouping” within a product.



Double-select on the *Handle* document as shown below. This will activate the part and switch you to Part Design, since the *Handle* is a part document. It should have a blue box around it in the tree to signify that it is the active component within the product structure.



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.

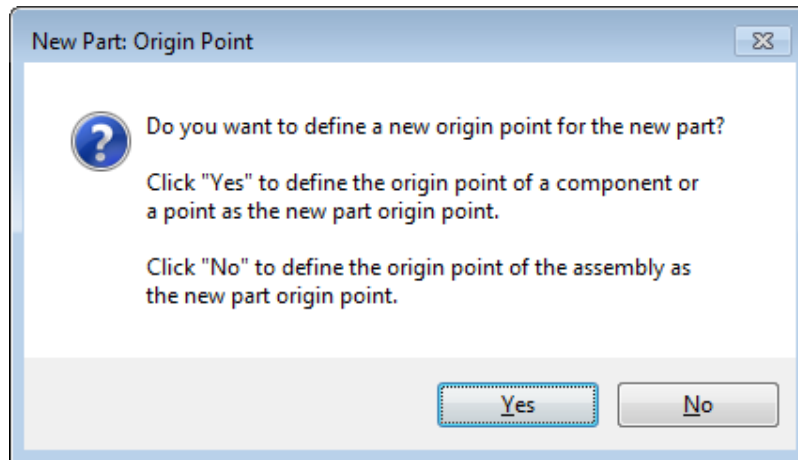
Double-select *Ratchet Wrench* to make it the active product. A blue box does not always appear around the top-level product when it is the active component. However, a blue box will always appear around an active component at a lower level.

Creating a New Part

A new part that is created within a product is no different than a new part created by itself. It will be a separate document that can be opened independently from the main product. However, when creating the part inside a product, you have the option of defining the absolute origin of the part at a specific location, or at the absolute origin of the main assembly.

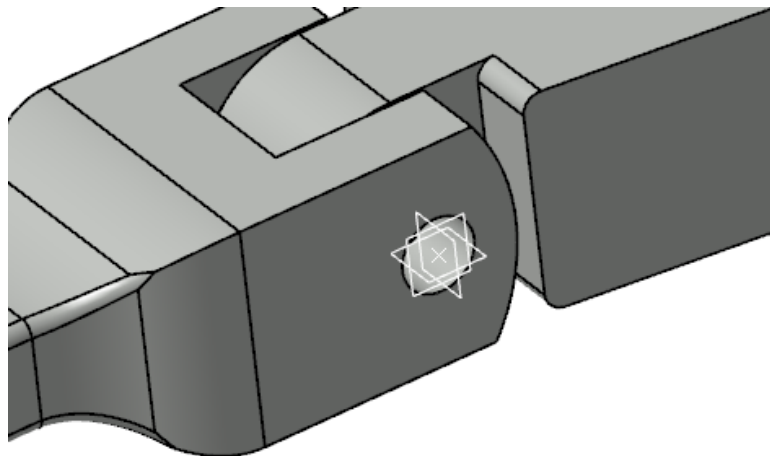
Select the **Part** icon and pick *Ratchet Wrench* from the tree.  The *Part Number* window appears.

Enter **Connecting Pin** and select **OK**. The *New Part: Origin Point* window appears.




Selecting *Yes* will allow you to define a new origin point for the part. Selecting *No* will use the origin of the assembly as the origin of the new part.

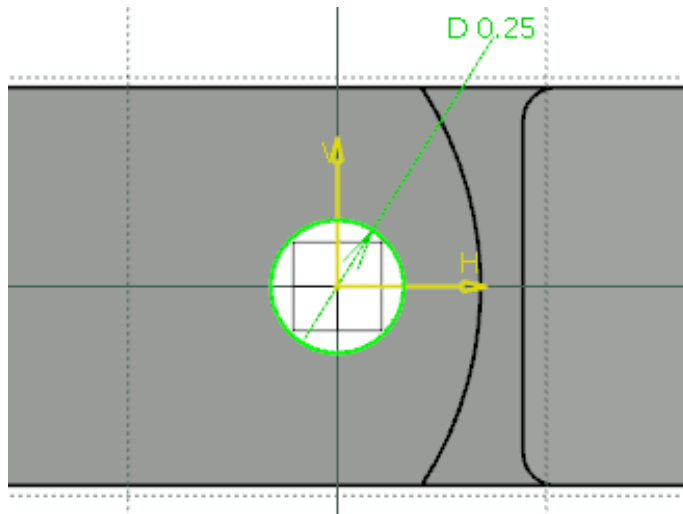
Select **Yes** and pick the point on the handle that you just showed. The new part is created, and its local axis system snapped to the specified point.




Double-select on the *Connecting Pin* document in the specification tree. The part is activated and you are switched to Part Design. You will need to expand the tree to see the document.

Select the **Sketch** icon, then select the **zx** plane of the part. 

Create a **0.25 inch diameter circle** and constraint it to the origin of the sketch. It should appear as shown.



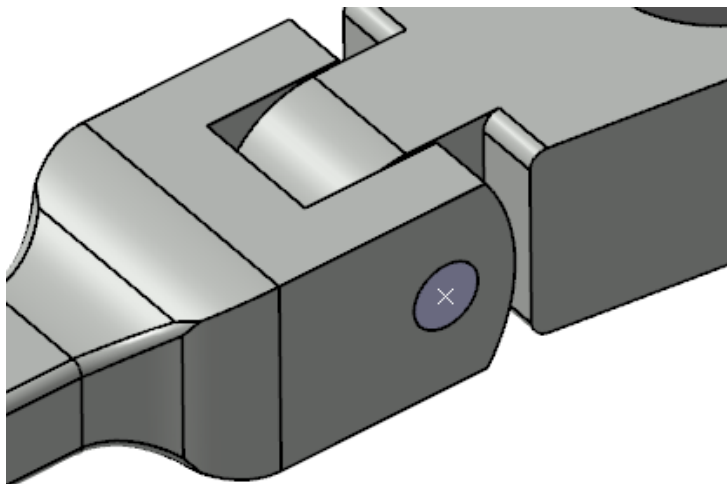
Exit the workbench.  You are switched back to Part Design.

Ensure the sketch is selected and click the **Pad** icon.  The *Pad Definition* window appears.

Enter **1.0** for the *Length* and reverse the direction if necessary so that the pin goes toward the *Handle* part. The pin is created.

Hide the reference planes of the part. This will clean up the model a bit and reduce clutter.

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

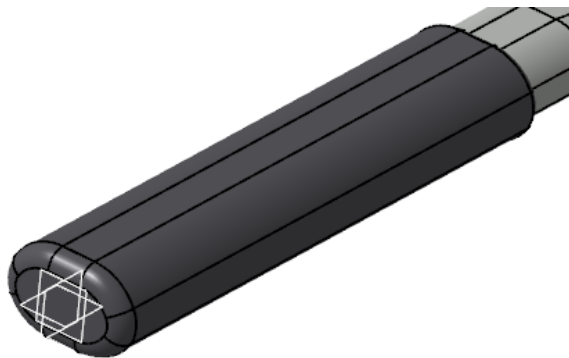



Another way to insert new or existing components is to press the third mouse button on a product and select the *Components* option. This gives you the ability to perform all of the items discussed so far without using an icon. This functionality is also available in multiple workbenches, whereas the icons are available in only a few workbenches.

Press the third mouse button on *Ratchet Wrench* in the tree and select *Components*, then select *New Part*. The *Part Number* window appears.

Select *OK* to the window without entering anything. The *New Part: Origin Point* window appears.

Select *No*. The local axis of the part is located at the global axis of the assembly. In this case, it happens to be at the base of the handle. If you need to build in aircraft coordinates, this is ideal.

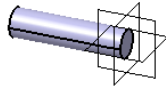


Select *Undo* until the new part disappears.  You will not need a new part. If you had created the pin using this origin, the pin would need to be located away from its origin. An example of each method is shown in the following images.

Using assembly origin:

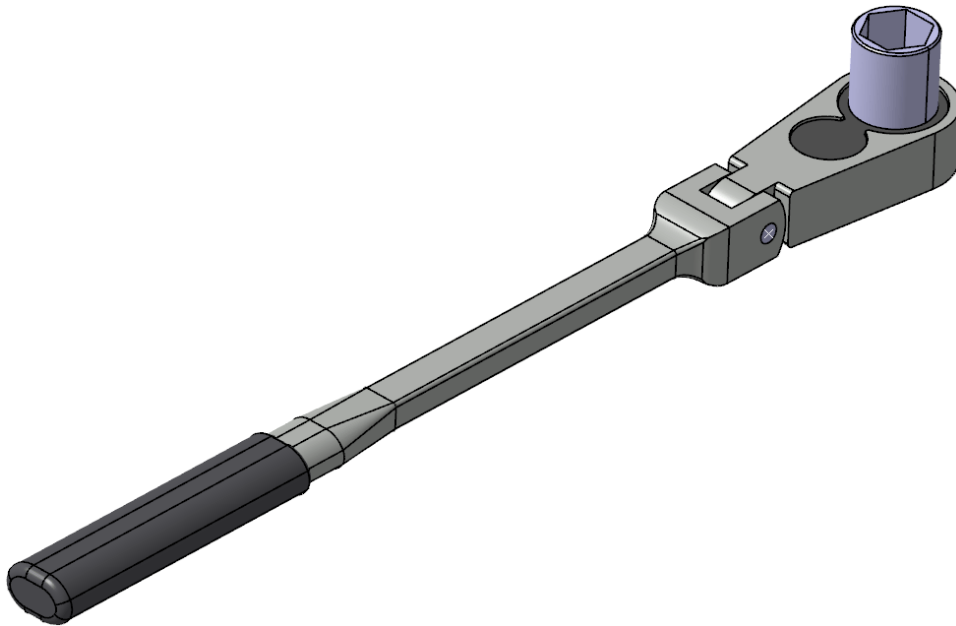


Using defined origin:



Press the third mouse button on *Ratchet Wrench* and select *Components*, then *Existing Component*. The *File Selection* window appears.

Select the **Hex Socket Large** document from the *Assembly Basics* directory and click *Open*. The socket is inserted into the assembly.



Creating a New Part from an Existing Part

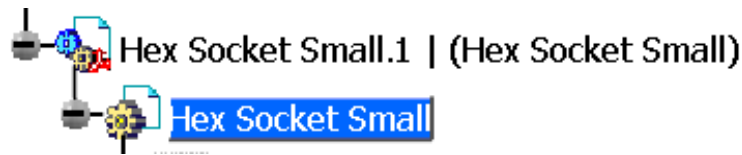
You will now create a new part from an existing part. Be careful when doing this, however, because once a part is instanced multiple times, changing any one of the instances will modify all of them. This is due to the fact that all of the instances refer to the same part document. You will need to take special care in renaming the part and instance with a different name than the original. Otherwise, you will overwrite it.

Double-select on the *Hex Socket Large* document to activate the part level.

Double-select on *Sketch.1* in the tree and change the diameter constraint to 0.75in, then exit the sketch.

Press the third mouse button on *Hex Socket Large.1* in the tree and select *Properties*. The *Properties* window appears.

Change the *Instance name* to Hex Socket Small.1 and the *Part Number* to Hex Socket Small, then select *OK*. The tree should appear as shown. Remember to change both the part number and the instance name to avoid confusion. Now, all you must do is save the file with a different name so that you do not overwrite the original document.

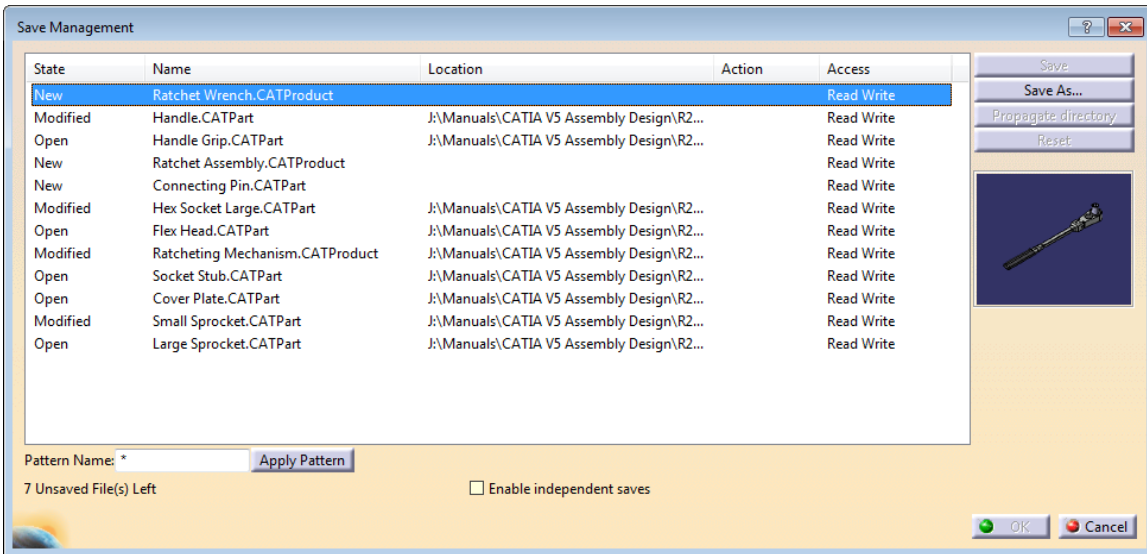


Double-select on *Ratchet Wrench* to activate the product.

Save Management

Save Management is the most commonly used tool to save products when working in a file-based environment. It provides a graphical representation of what will be saved before the operation is performed. Appendix B covers the Save Management options in detail.

Select the **File** pull-down menu and choose **Save Management**. The **Save Management** window appears. The **Handle Assembly** does not appear in the list because it is a component, which means that it is not a stand-alone document. It only exists inside the **Ratchet Wrench** product and is, therefore, saved with it.



Ensure the **Ratchet Wrench** product is selected in the window and click the **Save As** button. The **Save As** window appears.

Browse to your area and create a new folder called **Ratchet Wrench**, then open the **Ratchet Wrench** folder and select **Save**. You are returned to the **Save Management** window.

Select the **Propagate directory** button. All of the parts and products used in the **Ratchet Wrench** assembly are repointed to your directory.

Select the **Hex Socket Large** part from the window, then click the **Save As** button. The **Save As** window appears. This part needs to be saved as **Hex Socket Small**.

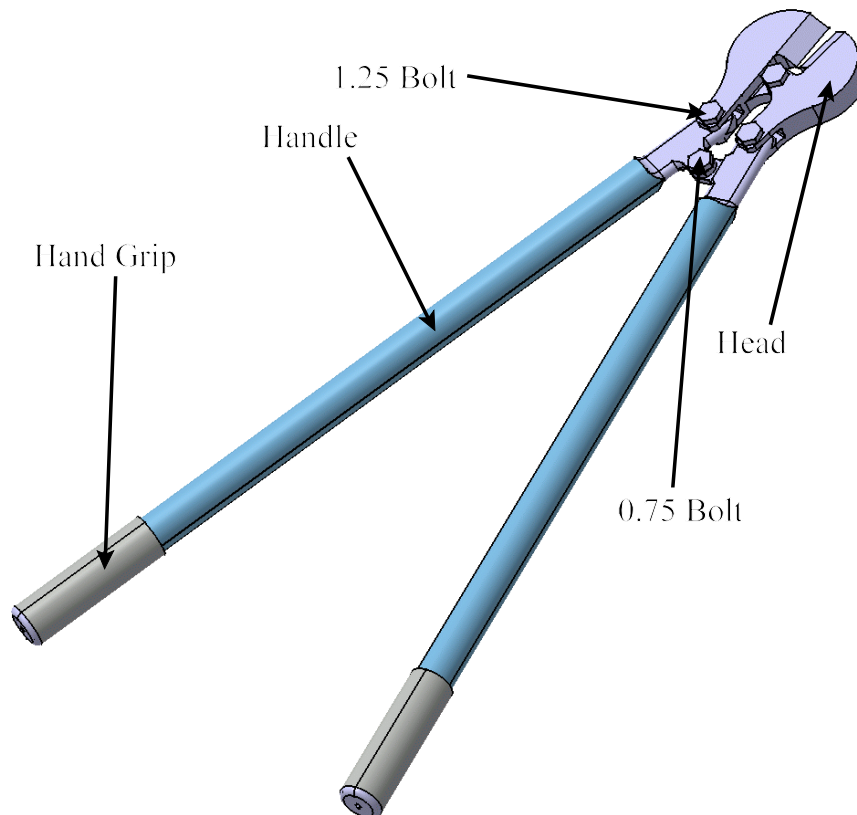
Change the name to **Hex Socket Small** and click **Save**. You are returned to the **Save Management** window.

Select **OK**. All the models are saved.

Constraining and Manipulating Parts Review

This exercise will review the various assembly constraints while introducing options to improve efficiency.

The first assembly will be a pair of bolt cutters. The completed model is shown below, with the filenames pointing to the parts. All of the parts are pre-existing with uniquely defined part numbers, except for the *0.75 Bolt*.



As a rule-of-thumb, for any parts that are connected together but will not move independent of each other (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. Close all other CATIA documents, then start a new product. Ensure you are in the Assembly Design workbench before continuing.

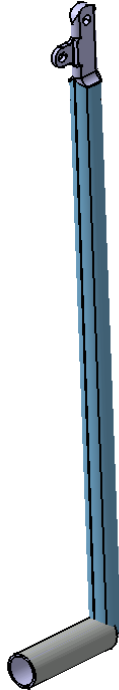
Press the third mouse button on *Product1* and select *Properties*. The *Properties* window appears.


Switch to the *Product* tab and change the *Part Number* to Handle Assembly, then select *OK*. This gives the unique ID of *Handle Assembly* to the product.

Save the document. The product name is automatically chosen as the file name.


Insert the **Handle** document from the *Constraints and Manipulation Review* directory. This can be done by selecting the Existing Component icon, then choosing the *Handle Assembly* product from the tree.

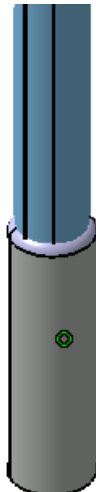
Insert the **Hand Grip** into the assembly. The hand grip and the handle do not insert in the proper place. This is common unless the parts are built at specific locations in space.



Create a **Fix Component constraint on the handle**.  The part will not be allowed to move.

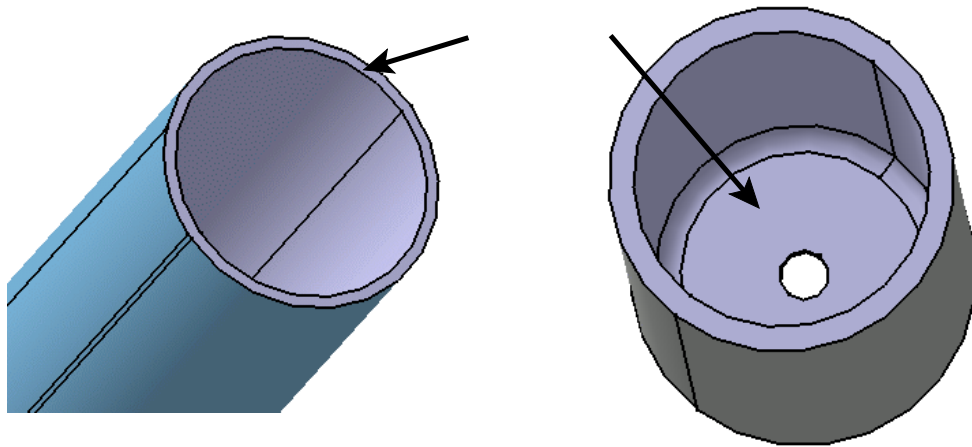
Create a **coincidence constraint between the center lines of the handle and the grip**.

 You may have to select the inside cylindrical surface of the handle in order to select the center line. Initially, the grip may be in the wrong orientation.

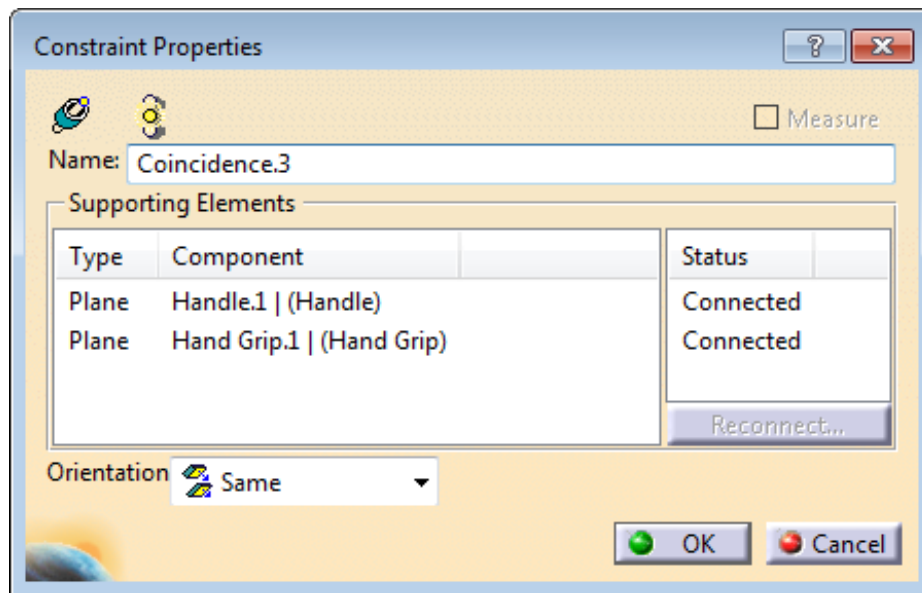


Select the **Coincident Constraint** icon, then select the two faces shown below. 

You will have to rotate the model in order to select the faces.

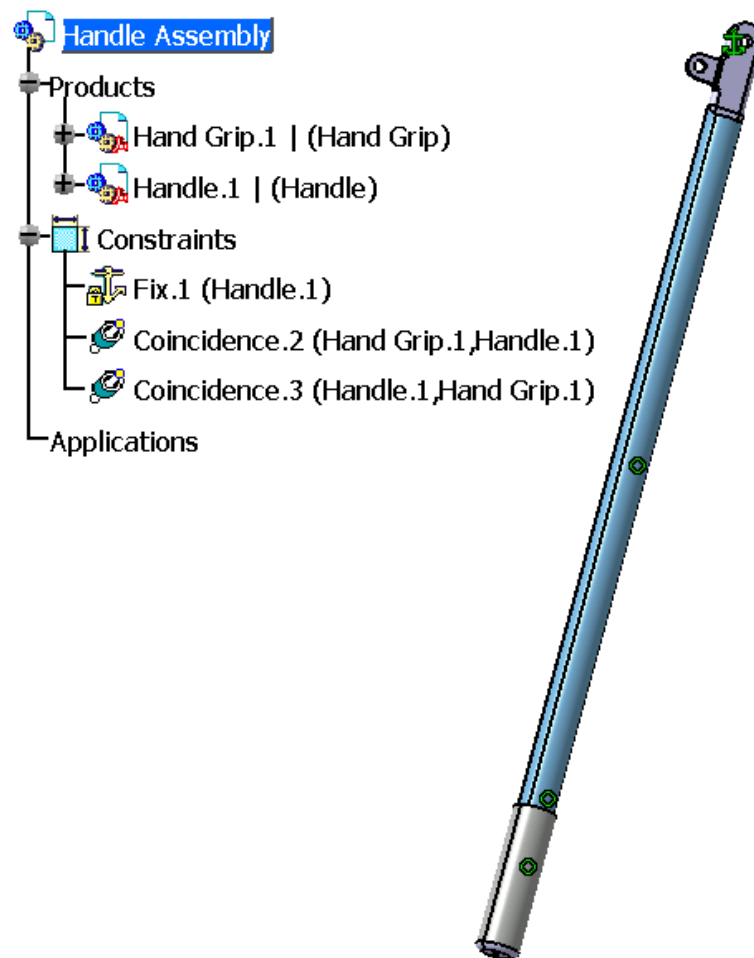


The *Constraint Properties* window appears.



Change the *Orientation* to *Opposite* and select *OK*.

The parts are snapped together as shown.



This completes the assembly. It is a good idea to hide the constraints to prevent them from cluttering the display when the assembly is used elsewhere.

Hide the constraints, then save and close the assembly. You will use *Handle Assembly* as a sub-assembly in another product.

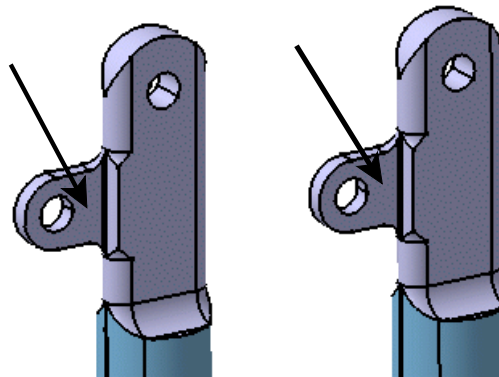
Start a new product named Bolt Cutters. Do not forget to save your assembly throughout the exercise.

Insert your Handle Assembly twice. Both assemblies will appear in the same location, so it may seem like there is only one. However, in the specification tree you will see that there are two.

Move the handle assemblies so both are visible. This can be done with the manipulation bounding box, the compass, or the Manipulation icon.

Select the Fix Component icon, then select *Handle Assembly.1* from the specification tree.  This will fix the sub-assembly in space.

At the top of the handle, there is a rotation point, which is where the handle assemblies will be connected.



Select the **Coincidence Constraint** icon, then select the two faces indicated above.



The orientation needs to be changed in order to flip one of the assemblies around.

Change the *Orientation* to *Opposite*. You can also change the orientation by selecting one of the green arrows on the faces.

Select *OK*. This may have changed the position of the handles dramatically, but you will correct it in the following steps.

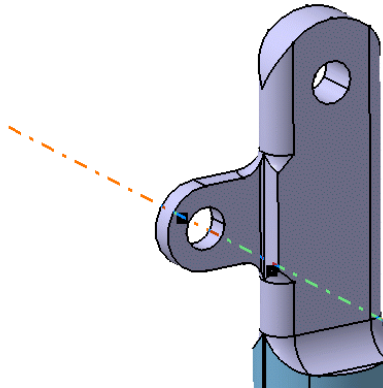
Perhaps you wondered why a contact constraint was not used. It would make sense here since the faces are contacting each other. The difference, however, is that a contact constraint means the two faces are locked together. In other words, they are glued, welded, bolted, etc., in the physical world. This is not the case for the current scenario. The handles need to rotate freely. It does not mean that a contact constraint would fail. However, other applications within CATIA V5 will use the constraints that are applied to perform analyses and simulations, and a contact constraint may be confusing. By defining the constraints correctly here, downstream users will not have to re-define them later.

Select the **Coincidence Constraint** icon.




Next, you will select the center lines of the holes in order to align them properly.


Select the centerline of the pivot hole as shown below. Move the mouse cursor inside the pivot hole to display the centerline.



Select the other centerline to align the holes. The two handles will snap together. It is very likely that they are rotated in such a way as to be invalid for actual operation of the bolt cutters. There are no constraints prohibiting the two handle assemblies from rotating around the centerline.

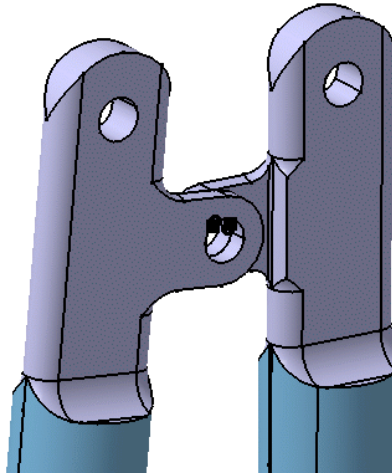
Now it is time to move the two handles with respect to their constraints.

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

Select the Drag Around any Axis option.  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 *Handle Assembly.2* so that it appears like below, then select **OK**. This will make it easier to finish building the bolt cutters. Remember, you will only be able to rotate *Handle Assembly.2* because the other handle is fixed.

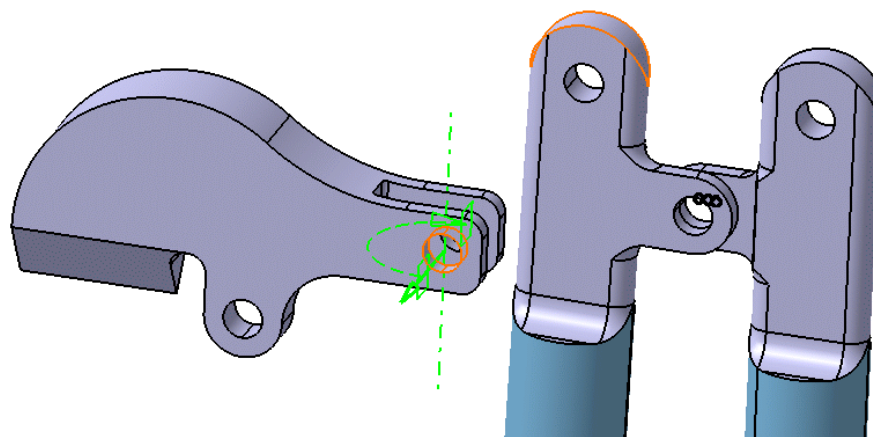


This would be a good point to save your document.

Insert the Head document into the assembly. Move the head to a more suitable location at the top of the handles. You may have to zoom out to see the head part. With the head near the proper location, the snap option will be used to move it in place.


Select the Snap icon.  This option will allow you to position or “snap” two objects together.


Select the cylindrical surface of the hole in the head and the top surface of one of the handles. When you select the top of the handle, the centerline is used.



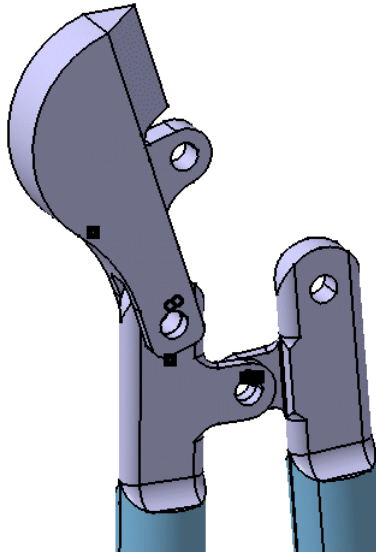
Note the series of green lines and arrows. Selecting these will reverse the direction of the snap and flip the part in various ways. Use them to orient the head properly to the handle. Selecting anywhere on the screen will release the snap motion. No constraints are applied when using the snap.

Note: The rotation of the head may be different than what is depicted in the previous image. Do not be concerned; you will rotate the head later.


Apply a coincidence constraint between the center lines of the head and the handle that was just snapped together.  This ensures that the two center lines are always aligned with each other.

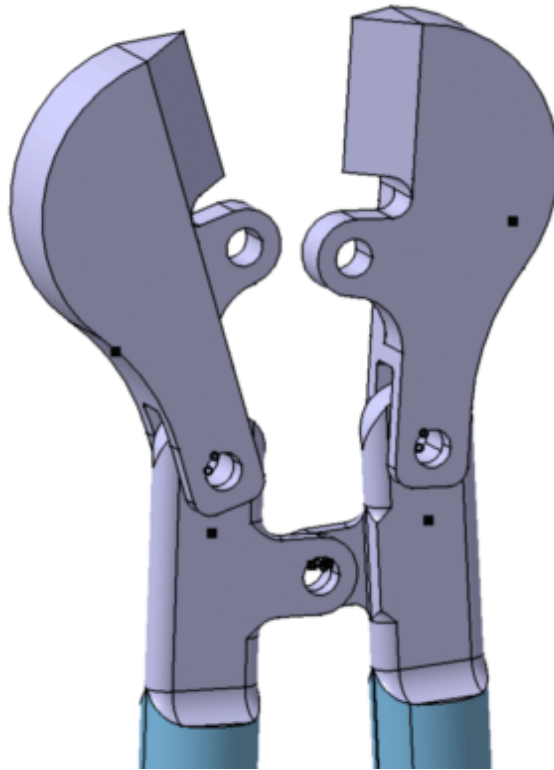
Apply a coincidence constraint between the two surfaces that will move the head into the proper location. 

Position the head so that it looks similar to the diagram shown below.

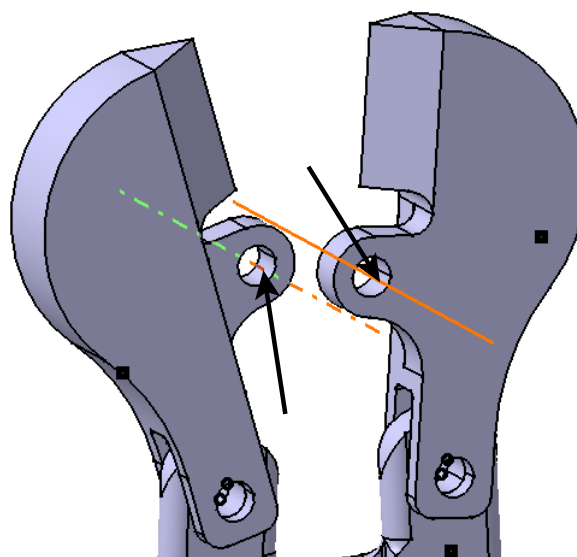


Insert another Head into the model.

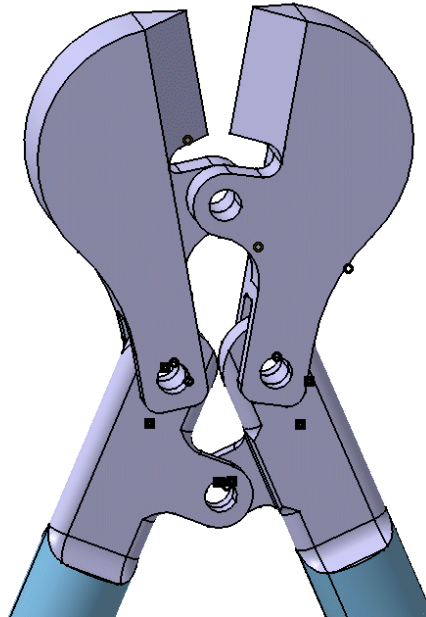
Add coincidence constraints to the new head and manipulate it so it looks similar to the diagram shown below.  Remember, it is imperative that the head be facing the correct direction.




Select the **Coincidence Constraint** icon, then select the two centerlines shown below.



The assembly should appear similar to this.




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

Select the **inside of the pivot between the two handles, then select and drag the handle that is not fixed.** The whole assembly moves. This is not kinematics because the assembly can be moved beyond the physical limits, but it is a good way to test the model to ensure that the entire assembly is constrained correctly.

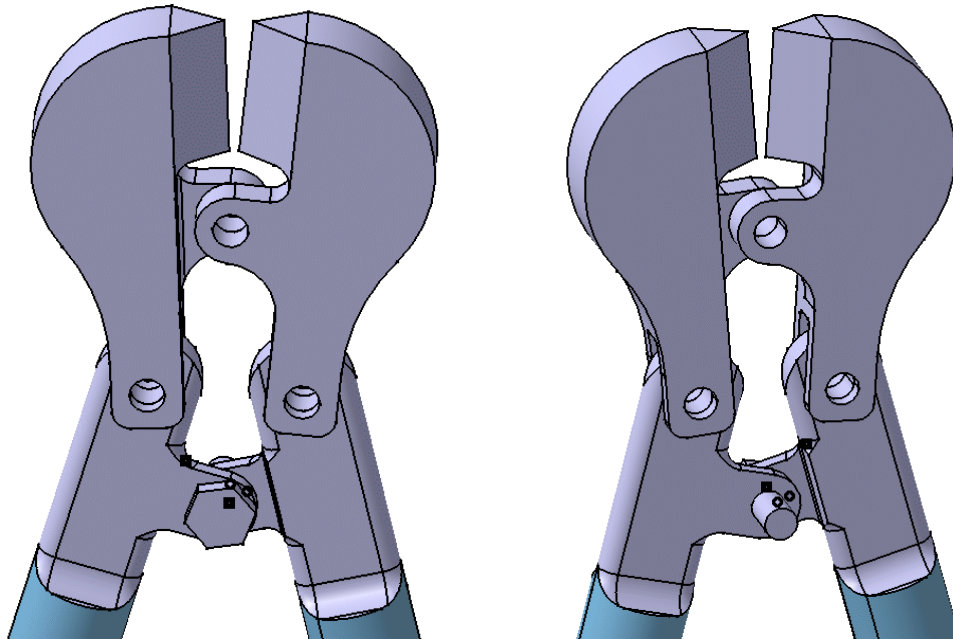
Select **OK when finished.** There are only a few things left to add and they are bolts. You will now insert them.

Hide your constraints, then insert the 1.25 Bolt document 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.  Remember, this will align the bolt with the hole. Be sure to select *OK* when you are done.

Add a contact constraint to place the bolt head against the face of one of the handles.

Your assembly should appear similar to the diagram shown below. This time a contact constraint was used since the bolt should be tight against the handle.

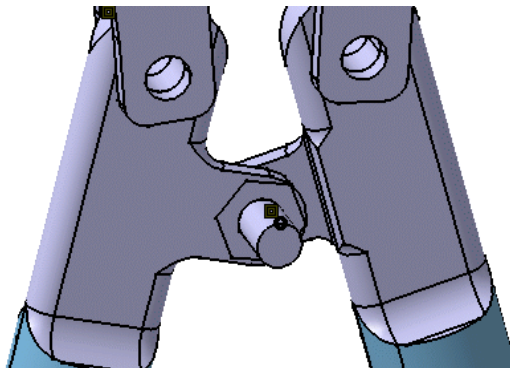


This would be a good time to save your document. Notice that the bolt sticks out too far. You need to modify the bolt to be 0.75 inches long instead of 1.25 inches, then save it as a separate part. You will do that next.

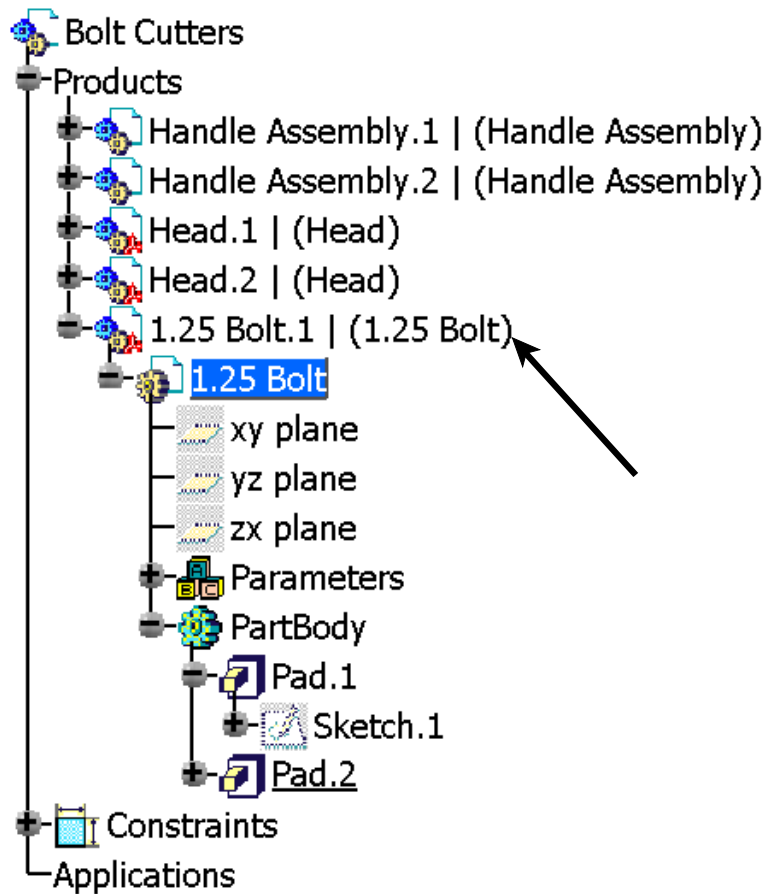
Expand the 1.25 Bolt branch until you see the *PartBody*. This will allow you to enter the Part Design workbench through the assembly.

Double-select the *PartBody*. You are taken to the Part Design workbench. 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.

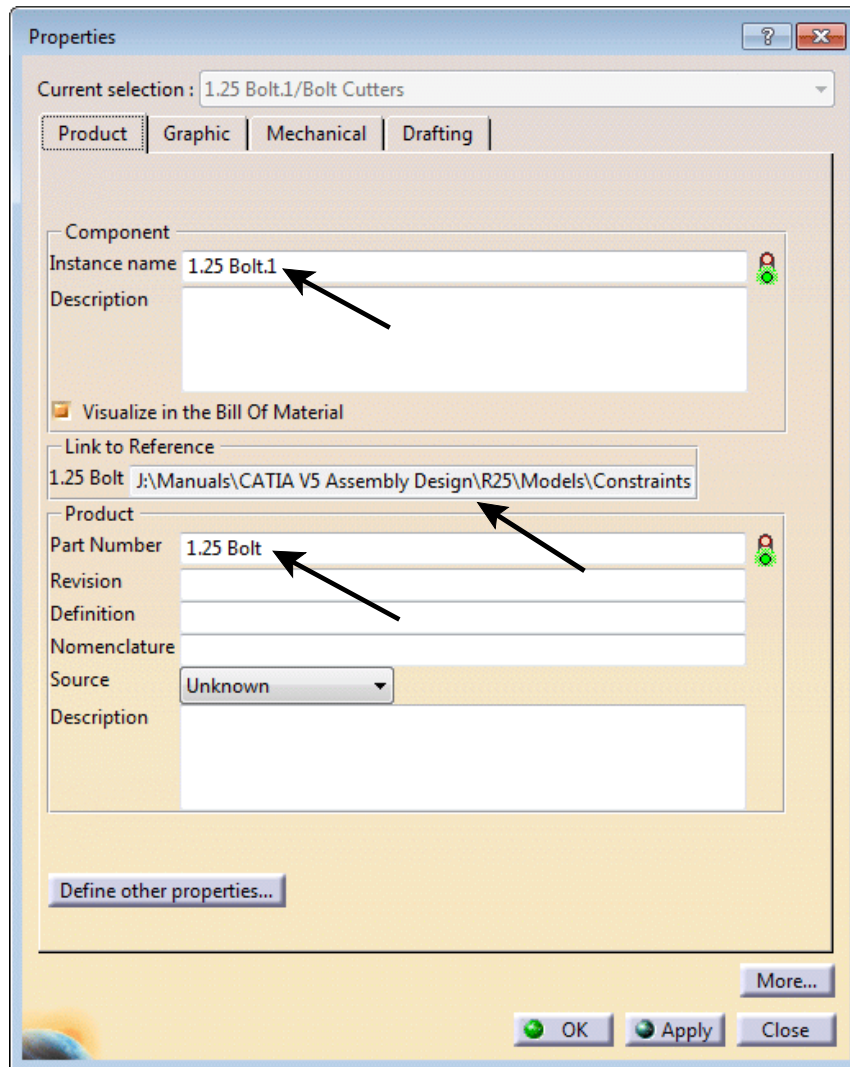
Change *Pad.1* to have a *Length* of 0.75 inches. The bolt becomes shorter, but the head of the bolt moves instead of the handle. This is determined by the original design of the bolt.



Press the third mouse button on the *1.25 Bolt* branch 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 saving the modified part.

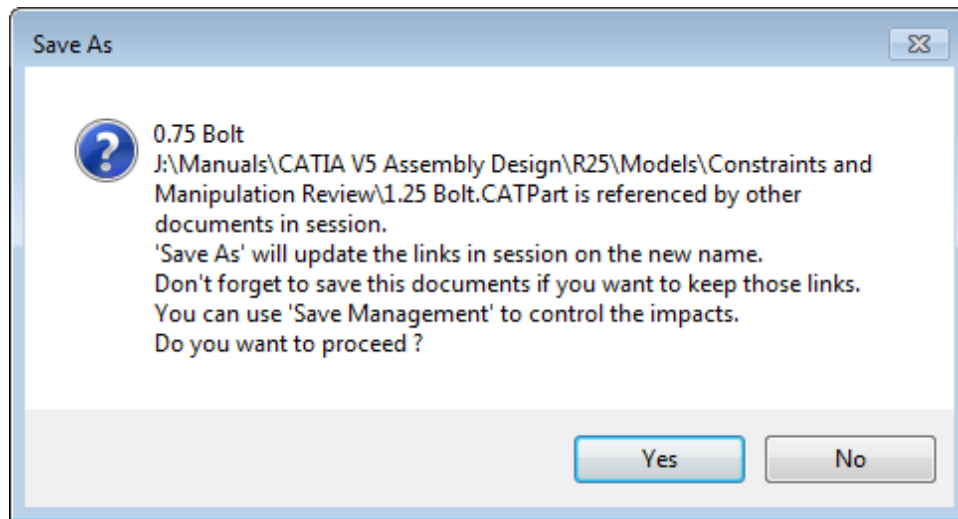


Select the **Properties** option. A *Properties* window appears similar to the one shown below.



Change the **Instance name** and the **Part Number** to **0.75 Bolt**, then select **OK**. Notice the *Link to Reference* area in the *Properties* window. This is very important. When you change the name, you want to save the part in a manner that will update this link.

Using *Save As*, save the modified part as **0.75 Bolt**. It is important that you use *Save As* instead of *Save* or you will save over the 1.25 Bolt. Be sure to browse to your area. This will bring up a window similar to the one shown below.



Select Yes. This will update your assembly with the new link to the 0.75 Bolt instead of the 1.25 Bolt. If you check the properties again, you will notice the change.

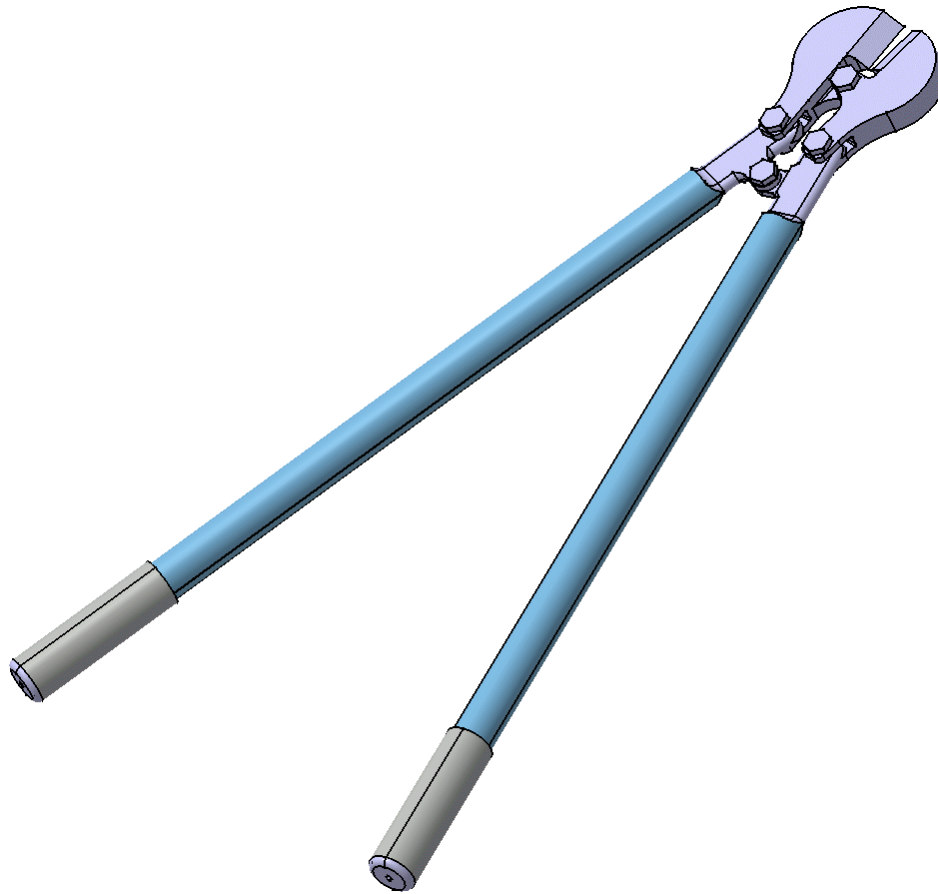
Double-select the *Bolt Cutters* branch of the tree. You are returned to the main assembly and the constraints are updated, which positions the bolt in the correct location. Remember, you used a contact constraint between the bottom face of the bolt head and the outside face of the handles.

Since you wanted the modified bolt to be saved as a different part, you had the extra step of renaming the part and the instance. This procedure works well as long as you remember to save the part while in the Part Design workbench in order to update all of the links. You could also have modified the part outside the assembly.

Note: The New From option in the File pull-down menu is another method of creating a duplicate 1.25 Bolt. You would then change the name to 0.75 Bolt, modify its size, and insert it in the Bolt Cutters assembly.

Insert the 1.25 Bolt and constrain it to the pivot location between the two heads.

Add two more bolts and constrain them into the proper locations. This will finish the assembly. Hide all of the constraints to clean up the assembly.

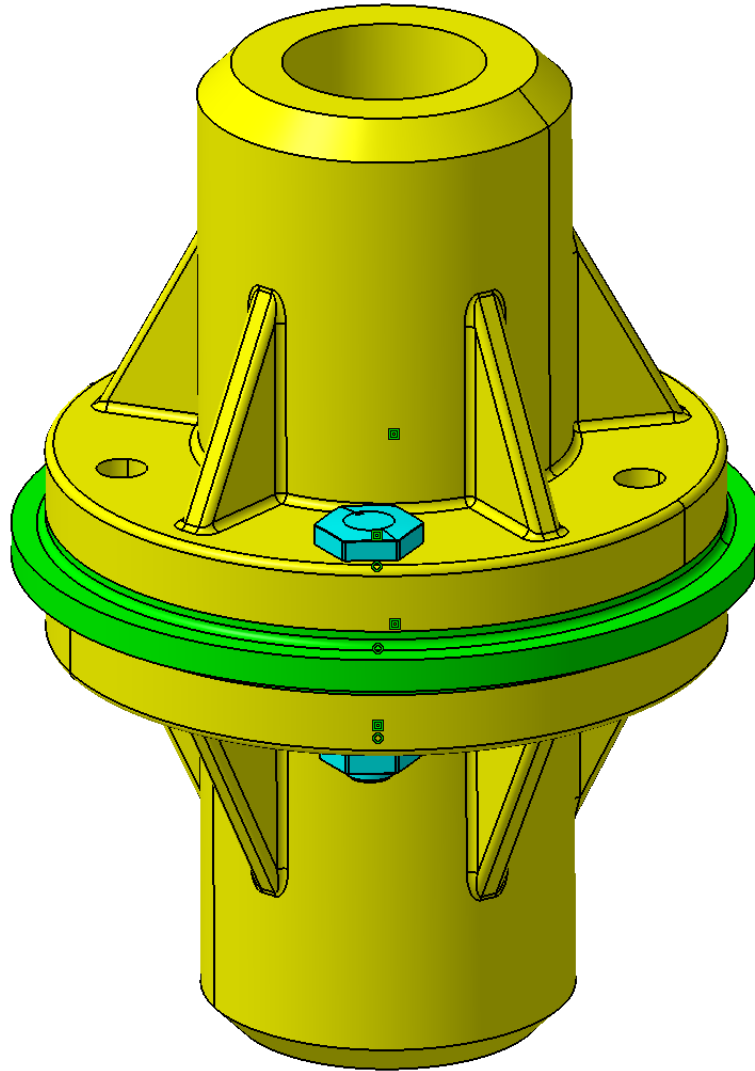


Save and close the document.


Reusing Patterns

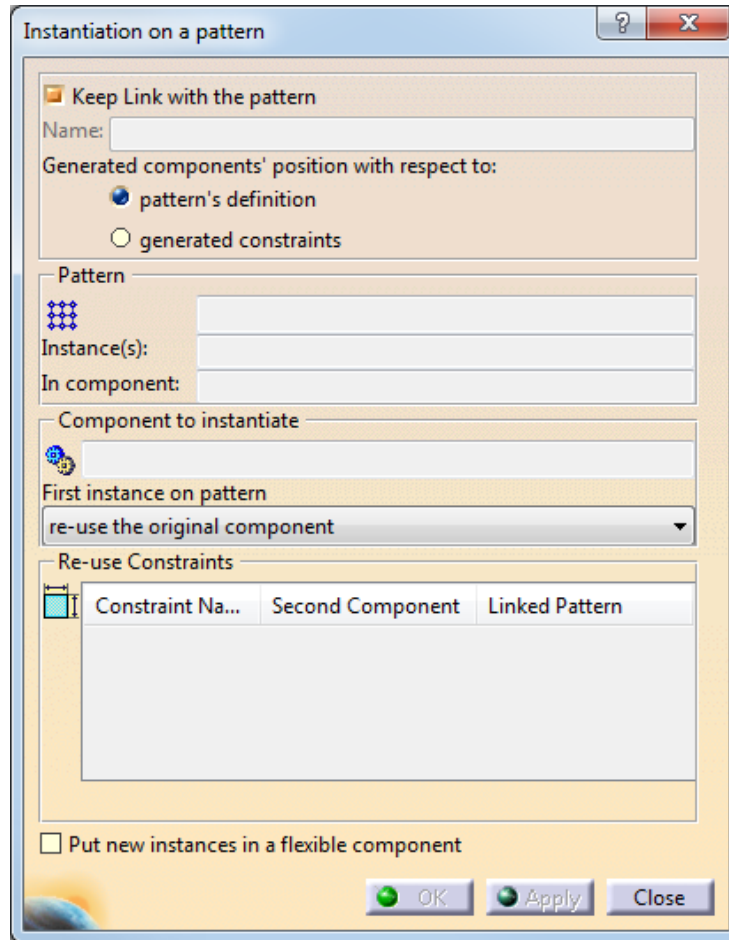
The reuse pattern option allows you to use an existing pattern within a part to help position and constrain components. This is very useful for constraining fasteners to patterns.

Open the Reuse Pattern document located in the *Reuse Pattern* directory. You should see a couple of casings being attached with a bolt and nut. You are going to put the rest of the bolts in the assembly.



Expand the *Products* branch and expand the *Casing.1* branch in order to see the patterns available. The *CircPattern.2* is the one you are going to reuse

Select the **Reuse Pattern icon**.  This icon will allow the user to reuse any pattern used in a part design. The *Instantiation on a pattern* window appears.



Keep Link with the pattern Updates the number of instances as the pattern is modified and creates a link back to the pattern

Generated components' position with respect to

pattern's definition Positions the instances with respect to the pattern's definition

generated constraints Positions the instances and generates constraints

Pattern Contains the pattern that is going to be used

Instance(s) How many instances the pattern contains

In component The component that the pattern exists in

Component to instantiate The item you are going to use in this pattern

First instance on pattern

re-use the original comp... The original component is used and others are added

- create a new instance* The component is copied and then placed into the pattern locations
- cut & paste the original...* The component is deleted by being cut and then pasted into the pattern locations

Re-use Constraints Allows you to specify which constraints will be used

All All constraints are reused

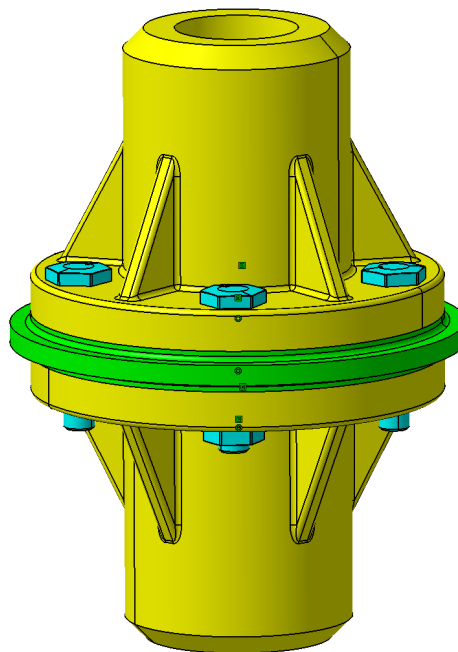
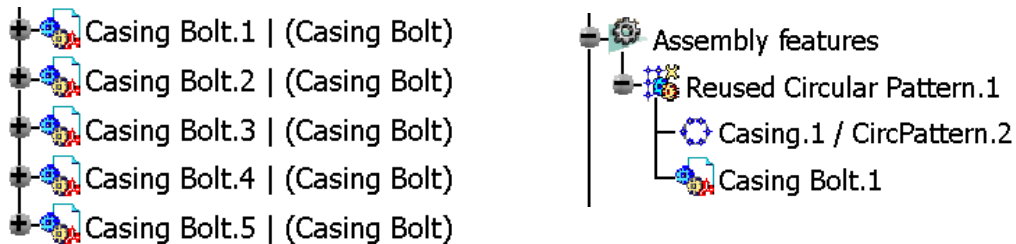
Clear No constraints are reused

Put new instances in a flexible component Places all new instances into its own flexible assembly

Make sure the *Keep Link with the pattern* option is on and select the *pattern's definition* option.

Select *CircPattern.2* from the specification tree and select the *Casing Bolt.1*. The available constraints appear in the *Re-use Constraints* area but you will not be using them.

Choose the *re-use the original component* option for the *First instance on pattern* and select **OK.** Four more instances of the bolt appear in the tree along with an *Assembly features* branch showing the link to the pattern.



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

Select the **Undo** icon to undo the creation of the bolts.

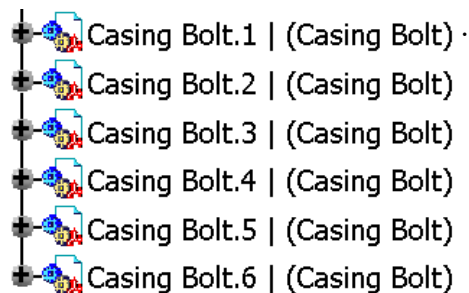


Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears.

Select the same pattern and the bolt again. This time you will change the *First instance on pattern* option.


Change the *First instance on pattern* option to create a new instance and select *OK*.

Notice that the bolt is instanced five more times instead of four. This is because you told it to create a new instance of the bolt instead of reusing the one you selected. The original instance still exists along with the five for the pattern.



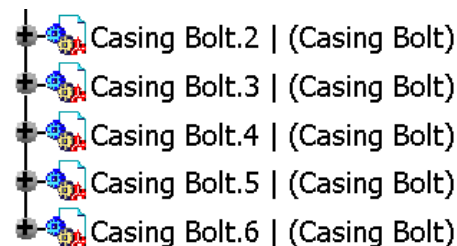
Select the **Undo** icon to undo the creation of the bolts.



Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears..


Select the same pattern and the bolt again. You will change the *First instance on pattern* option again.

Change the *First instance on pattern* option to *cut & paste the original component* and select *OK*. Notice that the bolt is instanced five more times but the original one no longer exists.



Select the **Undo** icon to undo the creation of the bolts.

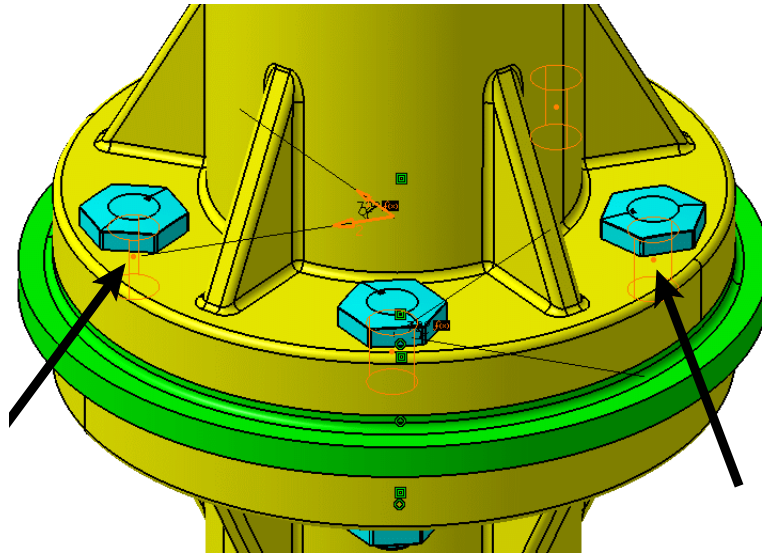


Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears.




Select the same pattern and the bolt again, make sure the *re-use the original component* option is selected and select **OK**. You get four more instances of the bolt.

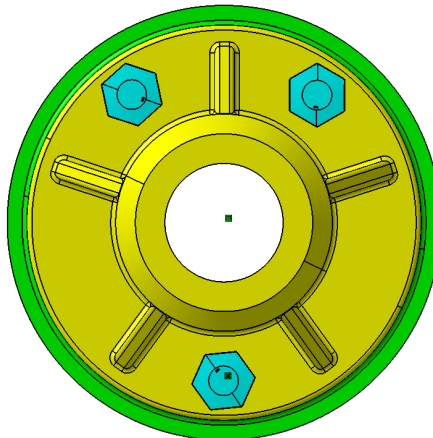
Double select on the *PartBody* of *Casing.1*. This makes that part the active component.


Double select on *CircPattern.2* and remove the two instances shown below and select **OK**. This will modify the pattern to have only three holes.




Double select on the *Reuse Pattern* product. This makes the top level assembly the active component and 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.

-  Casing Bolt.1 | (Casing Bolt)
-  Casing Bolt.2 | (Casing Bolt)
-  Casing Bolt.5 | (Casing Bolt)



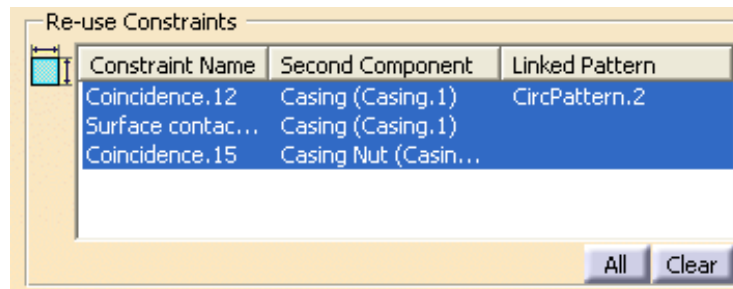
Select the **Undo** icon to undo the change to the pattern and to undo the instancing of the bolts.  You will now reuse the pattern without linking to it.

Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears.

Turn off the *Keep Link with the pattern* option. The generated constraints option automatically turns on. This time, instead of selecting the pattern and the bolt, you will just select the constraint that is common to both.

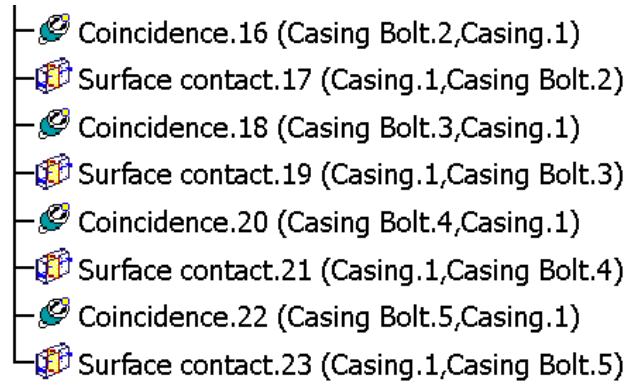
Select the *Coincidence.12* constraint. This is an alternative way of defining the information. By selecting a constraint that is between a component and a pattern, it will automatically use the pattern and the component that the constraint is associated with. This method is available whether you are keeping a link to the pattern or not.

Look at the *Re-use Constraints* area in the window. This shows all of the constraints associated with the component and allows you to specify which ones you want created with each new instance. Notice that only the *Coincidence.12* constraint is linked to the pattern. This means that this is the only one that would change based on the location of the instance in the pattern. The other two constraints will be associated with the same elements as the original.




Select the *Coincidence.12* and *Surface contact.13* constraint in the window. You will have to use the Ctrl key to select both. You do not want the other coincidence constraint since it is tied to the nut and will cause the new instances to be over constrained since the nut is not being replicated. The surface contact constraint is fine since the surface it is using on the casing is the same surface for all of the instances. The coincidence with the casing will change to be at each location of the pattern due to the link with the pattern. You always have to be careful about which constraints you want to reuse.

Select OK. The bolts appear with the two constraints for each one. Notice that there is not an *Assembly features* branch since you do not have a link to the pattern.

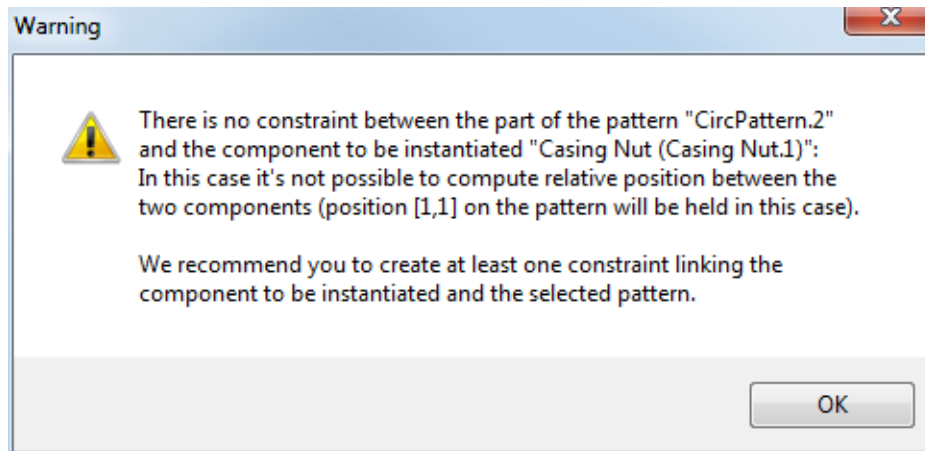


If you were to modify the pattern as you did before, the two bolts would still exist but their coincidence constraints would have an error. This is due to them being driven by constraints and not the definition of the pattern.

Select the Reuse Pattern icon.  The *Instantiation on a pattern* window appears.

Make sure the Keep Link with the pattern option is turned off. The generated constraints option automatically turns on.


Select the CircPattern.2 and the Casing Nut.1. A *Warning* window appears. Since the nut does not have a constraint that involves the pattern, it is recommending that you create one in order for the reuse pattern to work correctly.



Select OK. Notice that you cannot reuse the pattern.

Select Close. You will modify the coincidence constraint of the nut to use the centerline of the hole of the casing instead of the centerline of the bolt since the hole in the casing is created with the pattern.

Modify Coincidence.15 to use the centerline of the hole in Casing.1 instead of the bolt. Make sure you use the hole in *Casing.1* since that is where the pattern that you are using is defined.

Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears.

Make sure the **Keep Link with the pattern** option is turned off. The generated constraints option automatically turns on.

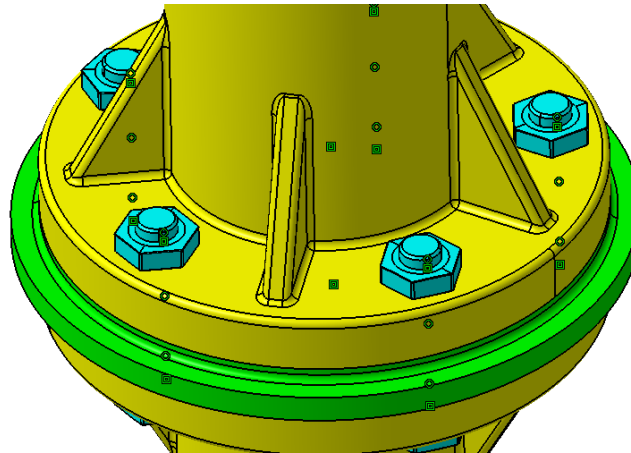
Select the **CircPattern.2** and the **Casing Nut.1**. This time you do not get a warning since a constraint exists between the nut and the pattern.

Change the **First instance on pattern** to use the **cut & paste the original component** option. You want the operation to create five new instances without leaving the original.

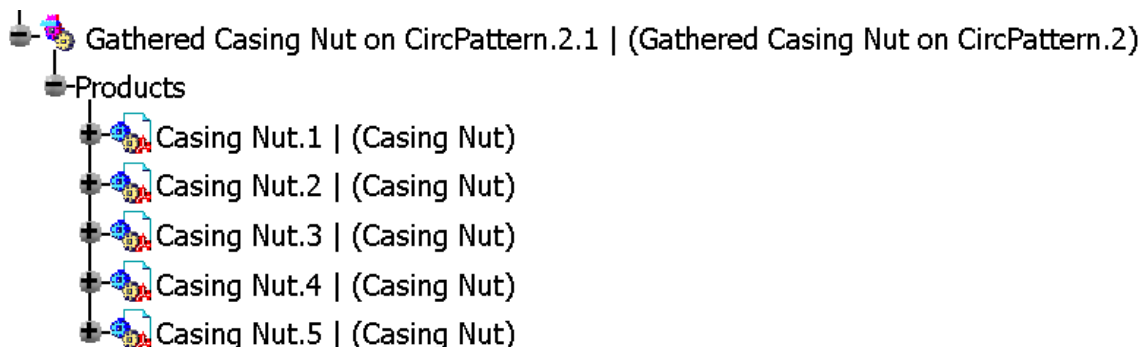
Select **All for the Re-use Constraints**. This ensures that both of the constraints will be generated with each instance.

Turn on the **Put new instances in a flexible component** option. This will put the generated instances in their own component and it will be flexible. Flexible components will be discussed in greater detail later in the course.

Select **OK**. The nuts appear. The assembly was rotated around for the picture shown below.



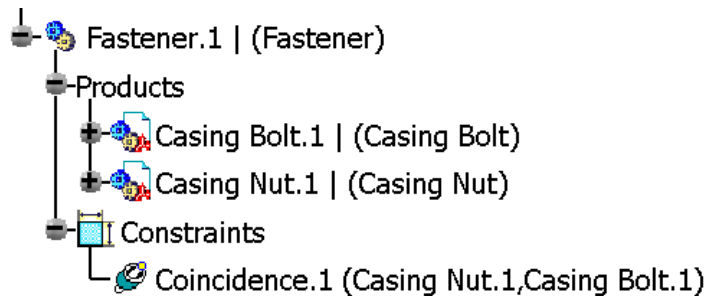
The nuts are all put into their own component and the component is flexible (pinkish purple gear).



Save and close your document.

Open the **Reuse Pattern2** document located in the *Reuse Pattern* directory. This will have you replicate the bolt and nut by having the bolt and nut be in a sub-component. This way you can replicate both parts by reusing the pattern once.

Notice that the assembly contains a *Fastener* component which contains the bolt and nut and it is constrained to the pattern in the casing.

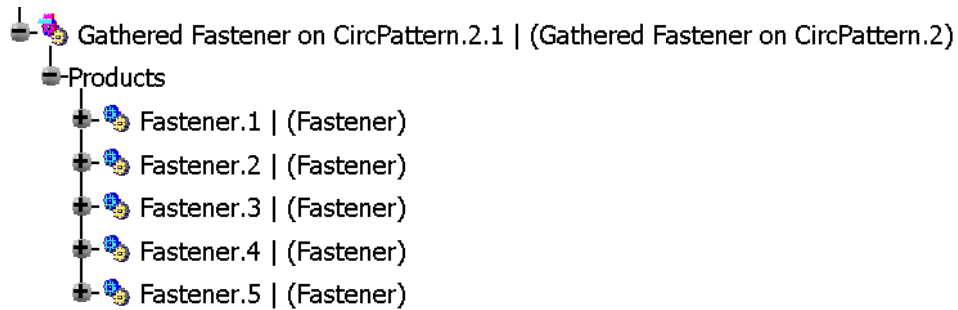


Select the **Reuse Pattern** icon.  The *Instantiation on a pattern* window appears.

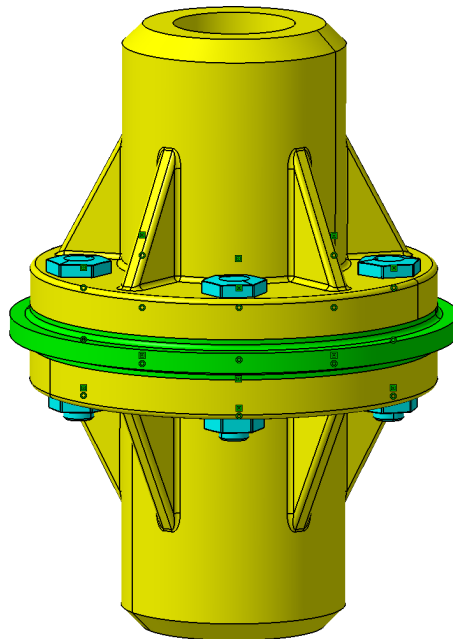
Make sure the *Keep Link with the pattern* option is turned off and the *Put new instances in a flexible component* is turned on.

Select the *CircPattern.2* in the *Casing.1* instance and select the *Fastener.1* component. You are going to want to use all of the constraints since these are used to constrain all aspects of the fastener.

Change the *First instance on pattern* to use the *cut & paste the original component* option and select *OK*. The *Fastener* component is replicated and put under a flexible component.



This provides another method of replicating the bolt and nut. This method created a component that contained both entities and then reused that component with the pattern. Previously you reused the pattern once for the bolt and once for the nut. It just depends on how you want your product structured.



Reusing patterns can be very beneficial, however, the right circumstances need to exist in order for it to be used effectively. You have to have a common component that you want replicated at locations defined by a pattern of a part. If you are having it generate constraints you must make sure that the constraints you choose are valid for each new instance. If you choose to have it link to the pattern then you will have an assembly feature in your product which some PLM solutions do not handle very well (This is discussed later in the course). However, if these circumstances are met then you can save a lot of time reusing a pattern.

Save and close your document.