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Introduction

CATIA Version 5 Basic Concepts

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

- Managing models (opening, closing, saving, etc)
- Solutions, Workbenches and Toolbars
- Tools, Options
- Tools, Customize
- Manipulation
- View Toolbar
- Specification Tree
- Modifying properties
- Applying materials to parts
- Creating axis systems
- Searching
- Measuring

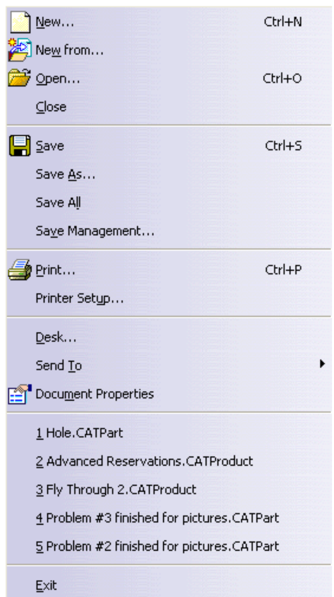
Model Management

This section will discuss how to manage your models including creating, opening, closing and saving.

Creating a new document

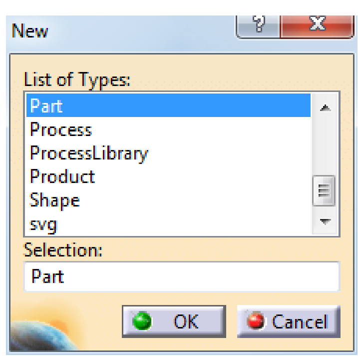
This allows you to start a new document. For this class you will normally start a Part document. This does not close any documents that are already opened, it only creates a new window with the document.

Select the *File* pull down menu.



Select *New*. A *New* window appears. This allows you to create a new document. The document that will be created depends on the selection you make, either an analysis, drawing, part, product or other. There are many different types of documents that you can create in CATIA. You can also use the New icon in the bottom toolbar or Ctrl+N to create a new document. 

Select *Part*.

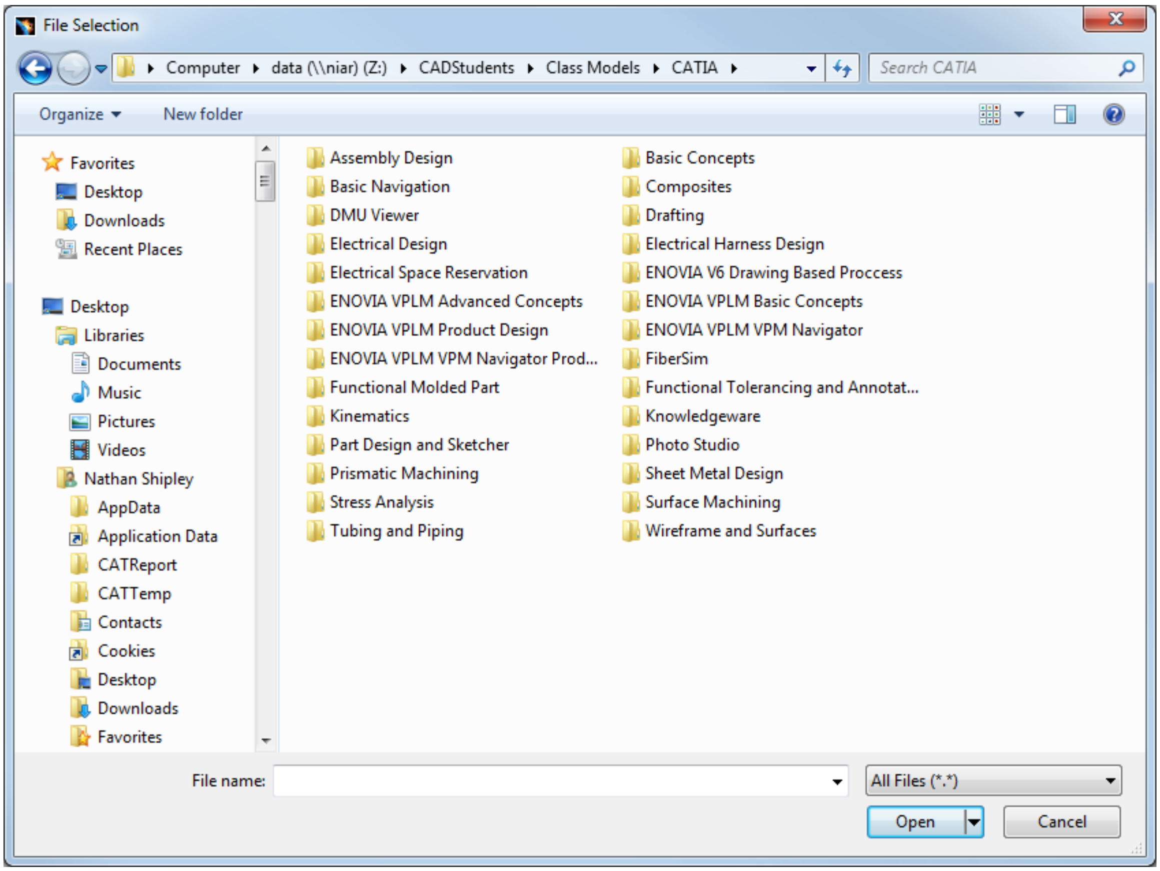


Select *OK*. This will create a new part. Depending on your settings, you might get a *New Part* which will allow you to give the part a name.

Opening an existing document

This allows you to open a document that has been previously saved. This does not close any documents that are already opened, it only opens a new window with the document.

Select the Open icon in the bottom toolbar.  This allows you to open an existing document. A *File Selection* window should appear. You can also use pull down menu *File, Open* or the Ctrl+O to open an existing document.



Find the document that you want to open and select it. You can either open a document by selecting it and then selecting *Open* or you can just double-click on the file and it will automatically open.

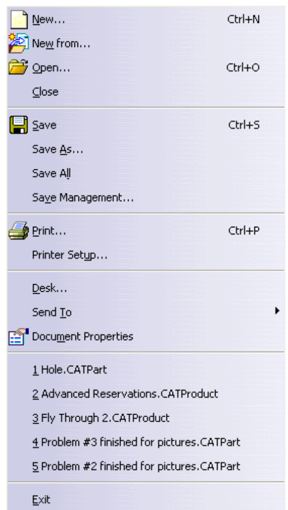
Select *Open*. This should open the document.

Saving a document

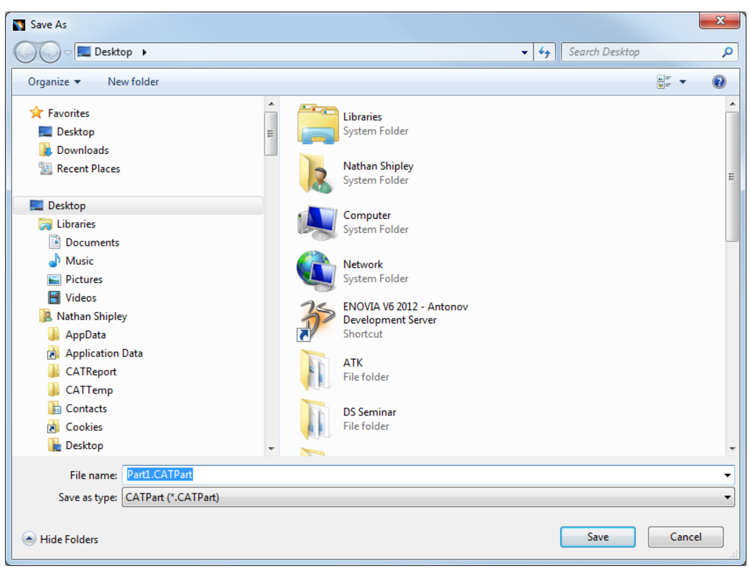
This allows you to save a document that you currently have opened. If you use the save icon, Ctrl+S or the *Save* option under pull down menu *File*, it will save the document with the current name. If this is the first time you saved this document then it will automatically open a *Save As* window allowing you to specify a name for the document. Otherwise it will just save the document with the same name it already has.

If you want to save a document with a different name, you have to use the *Save As* option in the pull-down menu *File*.

Select pull down menu *File*.



Select *Save As*. A *Save As* window should appear. You will need to specify which folder you want to save the document into and the name of the document.



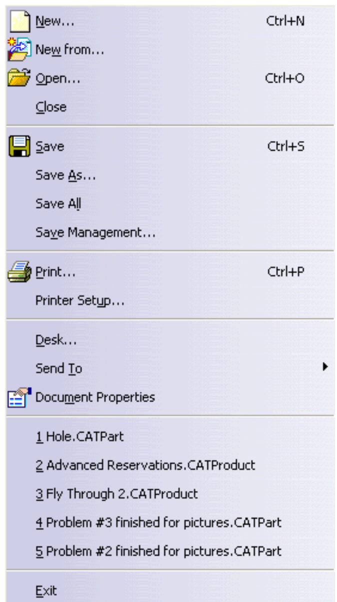
Choose the file location and then enter the file name in the *File name* box.

Select the *Save* button. Alternatively, you may press Enter. The document should save with the new name.

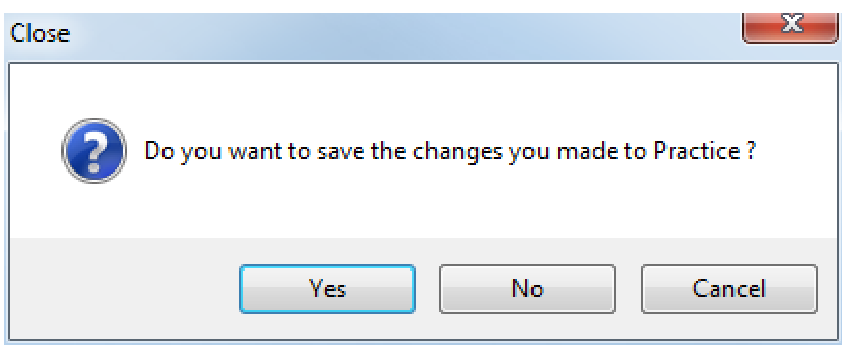
Closing a document

This allows you to close a document with or without saving. If the document has not been saved, CATIA will ask you whether or not you want to save it. It is important to get in the habit of closing your windows when you are finished working with them. Otherwise, you will end up with several windows open at the same time which could slow your system down.

Select pull down menu *File*.



Select *Close*. If the document has already been saved and has not been modified, CATIA will close the current document. If the document has been modified and not saved, CATIA will open the *Close* window.

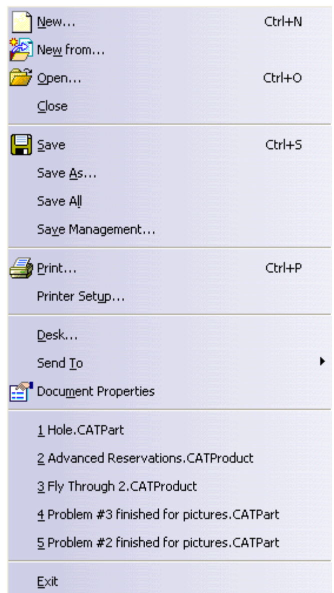


If you want to save the changes then press *Yes*. If the document has previously been saved, it will be saved again under the same name; otherwise, the *Save As* window will appear. If you do not want to save the changes, press *No*. To return to the document rather than closing it, select the *Cancel* button.

Creating a new model from an existing model

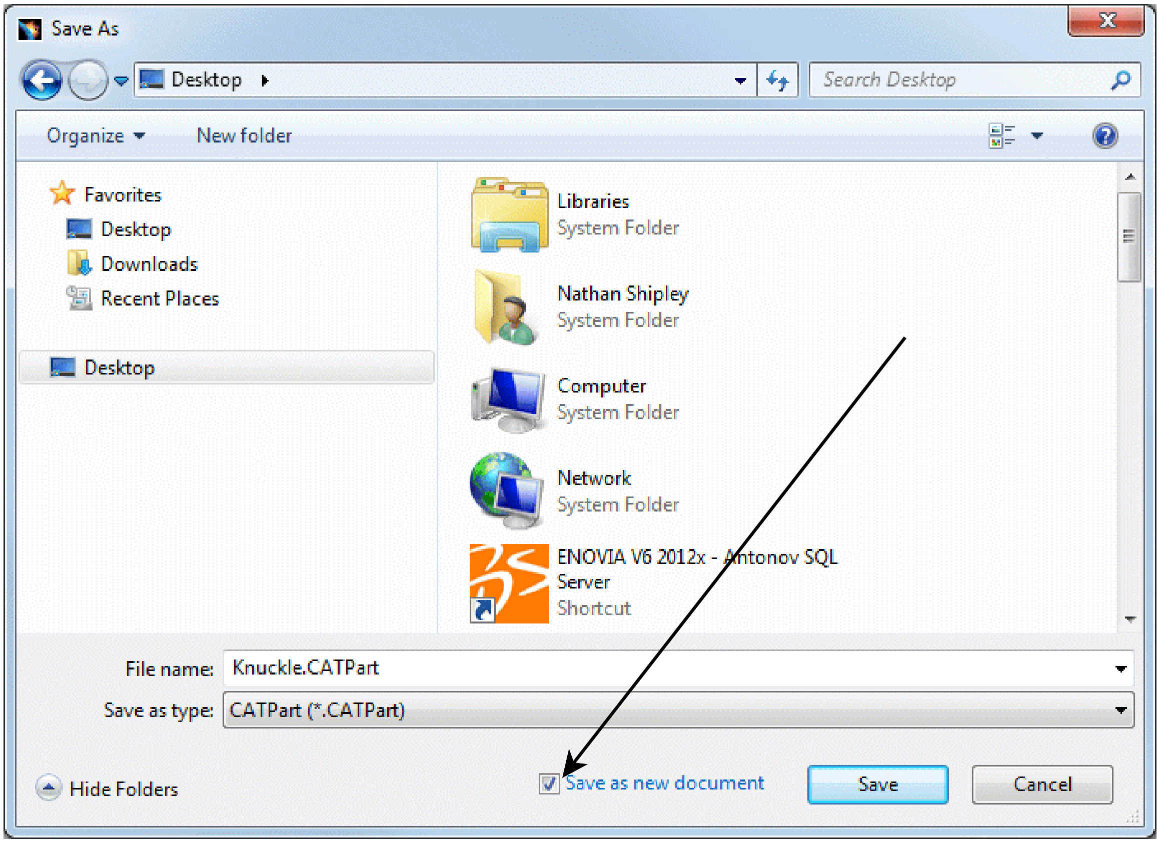
There are two different methods for creating a new model from an existing model. A CATIA model is more complex than a simple text file or Word document. All CATIA documents have a Unique Universal Identifier (UUID). This is basically just an internal number that is assigned to the document when it is created. These UUID's are not seen except within ENOVIA. UUID's are used to help file documents within ENOVIA. To save an existing document as a completely separate document, you will need to give the document a new UUID.

Select the *File* pull down menu.



Select *Save As*. A *Save As* window should appear. You will need to specify which folder you want to save the document into and the name of the document. To save the document as a completely new model so it is no longer linked to the original, you will need to use the *Save as new document* at the bottom of the screen.

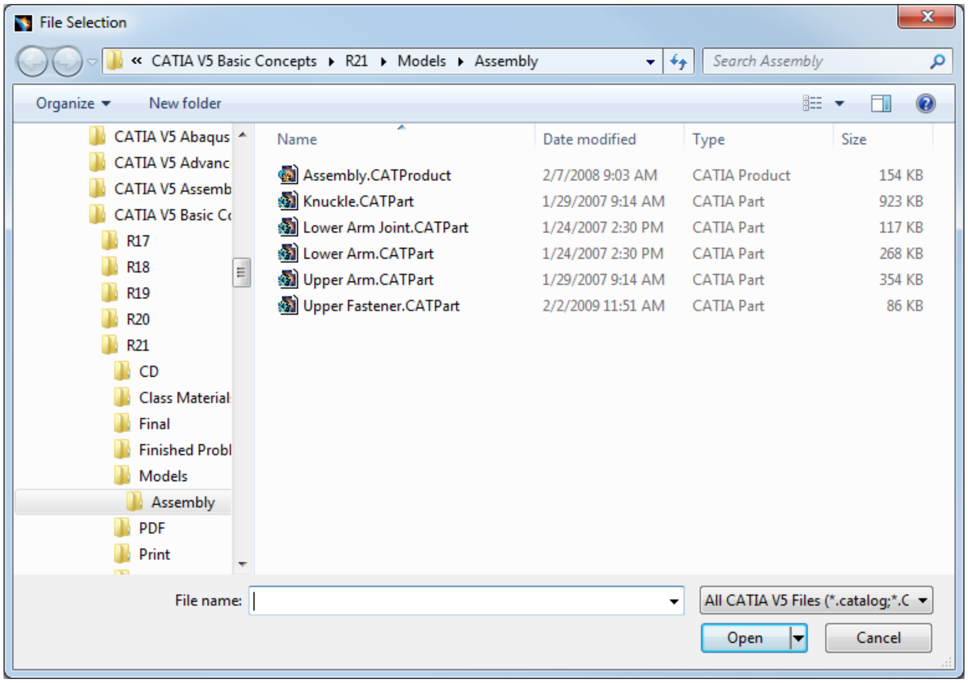
Select the *Save as new document* option as shown. This will specify that the new document needs to have a new UUID as well.



Select *Save*. The model is saved.

The other method is by using the *New from* option.

Select the *File* pull down menu and select *New from*. A file selection menu appears.



Select the document that you wish to open and select *Open*. The document is opened and depending on your settings, you might get the *New Part* window allowing you to specify a name. The document has actually already been assigned a new UUID.

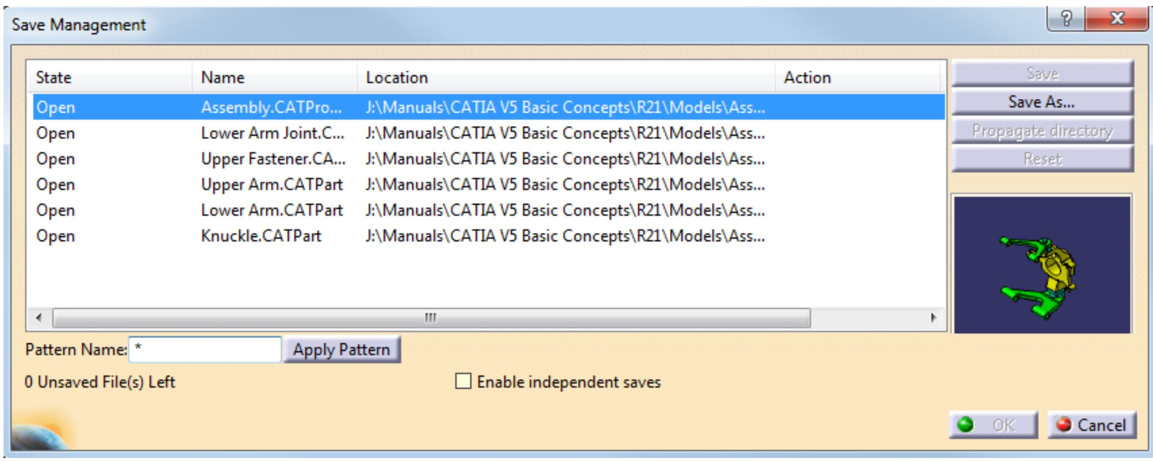
Select *File*, *Save*. The *Save As* window appears. You would generally expect the model to be saved automatically since you didn't change anything. However, since we used the New from option, the UUID of the document has been changed, so you must save it in a different location with a different name.

Save Management

This option is extremely important to understand what components are saved and where they are going to be saved. This is especially important when you are working with assemblies or with a flat file system.

Open the Assembly document located in the *Assembly* directory.

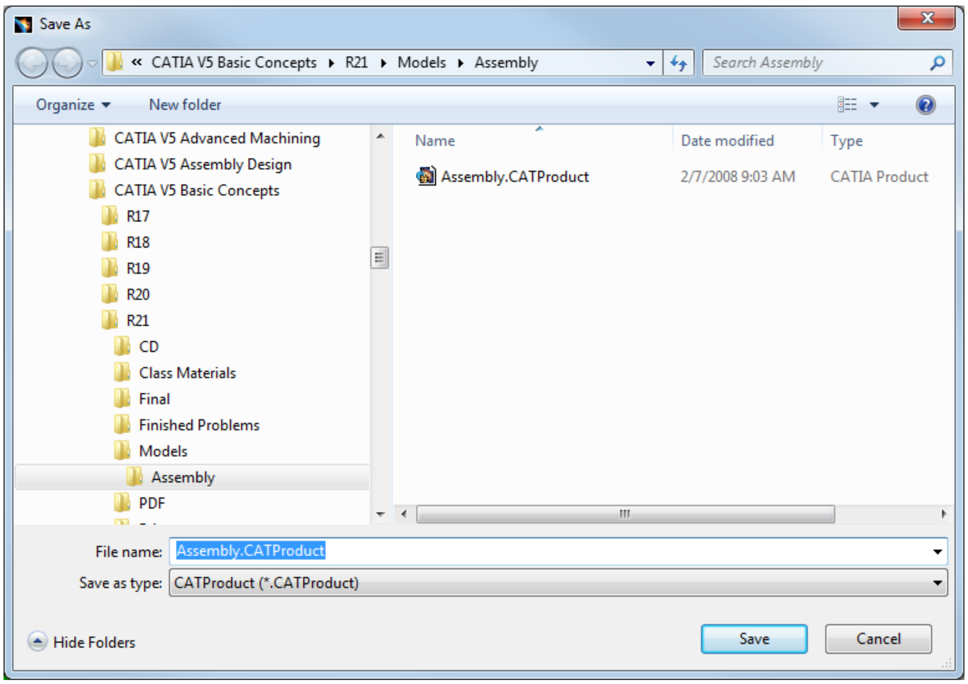
Select pull down menu *File, Save Management.* The *Save Management* window appears. Notice that the assembly and all of the parts that are in it appear in the window.



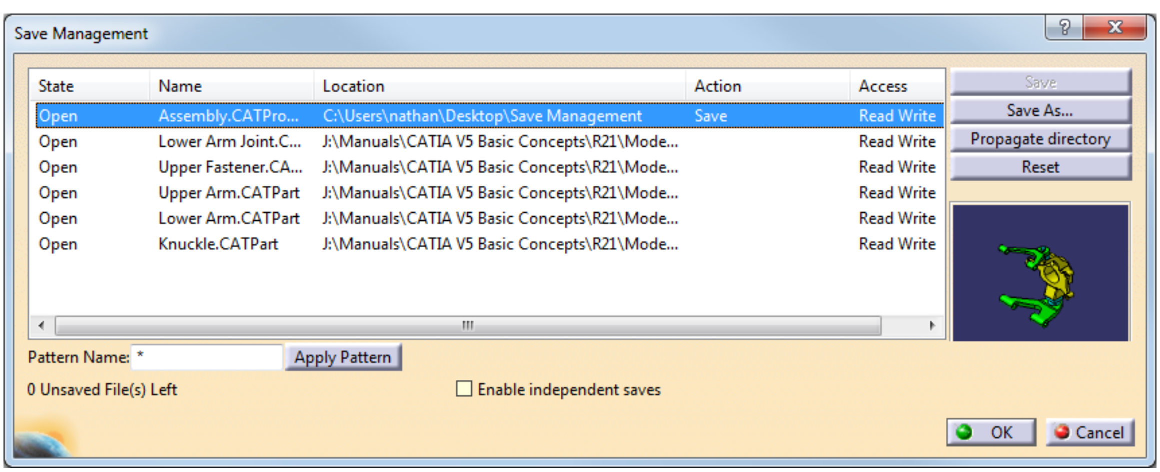
State	Specifies the state of the document. Some of the common states are *Opened, Modified*, and *Read Only*. *Opened* - the document is open but not modified. *Modified* - the document is open and has been modified. *Read Only* - the document is read only and cannot be saved in the current directory.
Name	Name of the document
Location	Current location of the document
Action	Specifies the action that will take place when you select *OK*. Some common actions are *Save* and *Save Auto*. *Save* - the document is saved with the current *Name* in the current *Location*. *Save Auto* - occurs when a higher level assembly is saved. The modified components of that assembly are tagged with *Save Auto* and will be saved with the current *Name* and in the current *Location*.
Access	Shows the type of access you have to the documents
Save, Save As	Forces a save action on the specified document
Propagate directory	The directory that the higher level assembly is saved in is propagated to all of the components
Reset	Resets all items to their original state before any changes were made

0 Unsaved File(s) Left	Specifies the number of unsaved files
Pattern Name	Specifies a naming pattern to use on the selected entities
Apply Pattern	Applies the pattern specified
Enable independent saves	Allows you to save items individually, without the *Save Auto* action occurring

Select on *Assembly* and select the *Save As* button. This will bring up the *Save As* window.

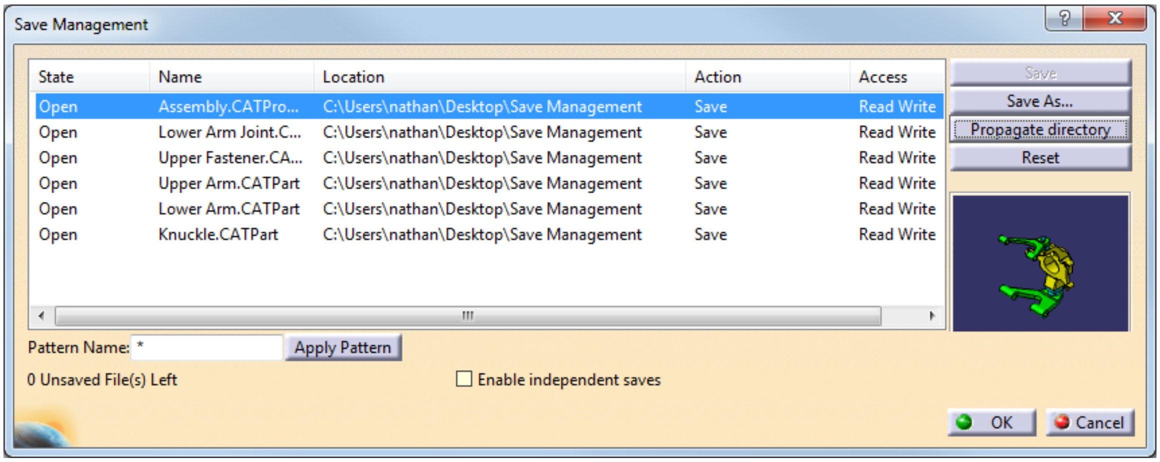


Save the model on the desktop in a folder called Save Management. This will return you to the *Save Management* window. Notice that the *Assembly* automatically gets tagged with a *Save*. If you said *OK* at this point, only the assembly would be saved, without all of the parts.



You could save each of the parts in this same manner, but there is a short cut.

Select the *Propagate directory* button. Notice that all of the parts will now be saved in the Save Management folder on the desktop. It is a good idea to save assemblies with all of their parts in order to make it easier to manage the links between the assembly and all of the parts.



Select *OK*. The assembly and all of its parts are saved in the *Save Management* folder on the desktop.

Close your document.

Manipulation

The following are actions that can be performed using the mouse when trying to view your parts or sketches.

Three button mouse

Center the display	Select and release the middle mouse button on the location that you want to be centered and it will move to the center of the display
Pan	Select and hold the middle mouse button and you can move your display around by moving the mouse
Rotate	Select and hold the middle mouse button and then select and hold either the first or third mouse buttons and you can rotate the display around by moving the mouse. You should see a rotational ball appear for reference. Both buttons will be held down simultaneously.
Zoom	Select and hold the middle mouse button and then select and release either the first or third mouse buttons and you can zoom in or out by moving the mouse up or down. Only the middle mouse button will be held down.
Rotate and Zoom	While on an geometrical entity you can press and hold the Shift key and then press the middle mouse button to perform a rotation and zoom using a view point control.

Two button mouse

If you only have a two button mouse to work with then you can do the same things, but you will have to use the keyboard in conjunction with the mouse.

Pan	While holding down the Alt key, select and hold the right mouse button
Rotate	While holding down the Alt key, select and hold the right mouse button and then either select and hold the left mouse button or the Ctrl key.
Zoom	While holding down the Ctrl and Alt keys, select and hold the right mouse button

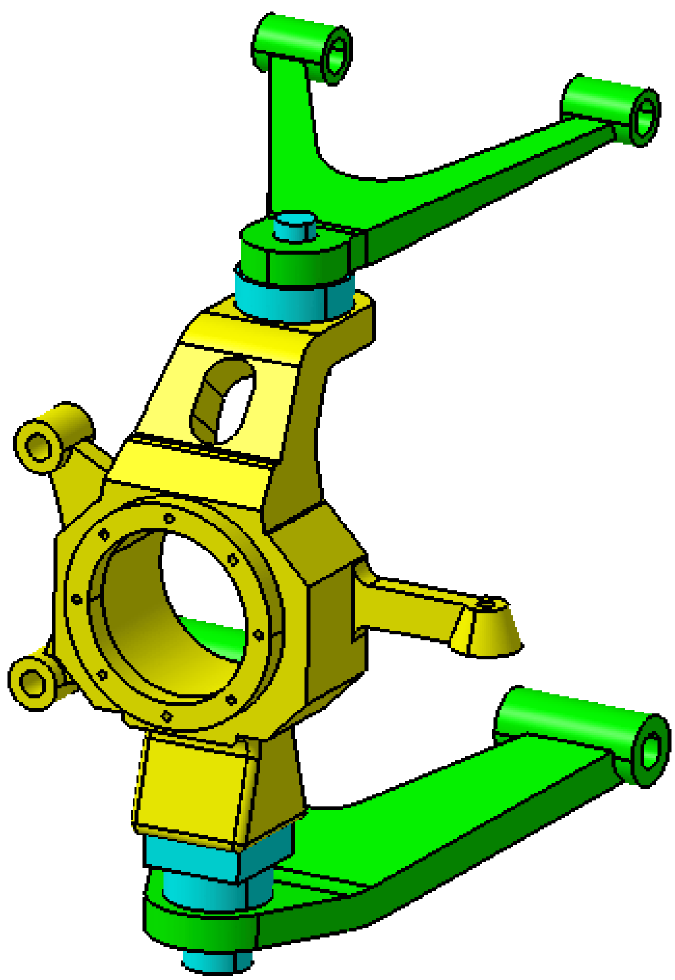
SpaceBall or SpaceMouse

They can be used to pan, rotate and zoom as long as the correct driver is installed.

Keyboard

Pan	Press and hold the Ctrl key and select the arrows to pan up, down, right or left
Rotate around the vertical	Press and hold the Shift key and select the left or right arrow
Rotate around the horizontal	Press and hold the Shift key and select the up or down arrow
Rotate around the normal	Press and hold the Ctrl and Shift keys and select the left or right arrow
Zoom In	Press and hold the Ctrl key and select the Page Up key
Zoom Out	Press and hold the Ctrl key and select the Page Down key

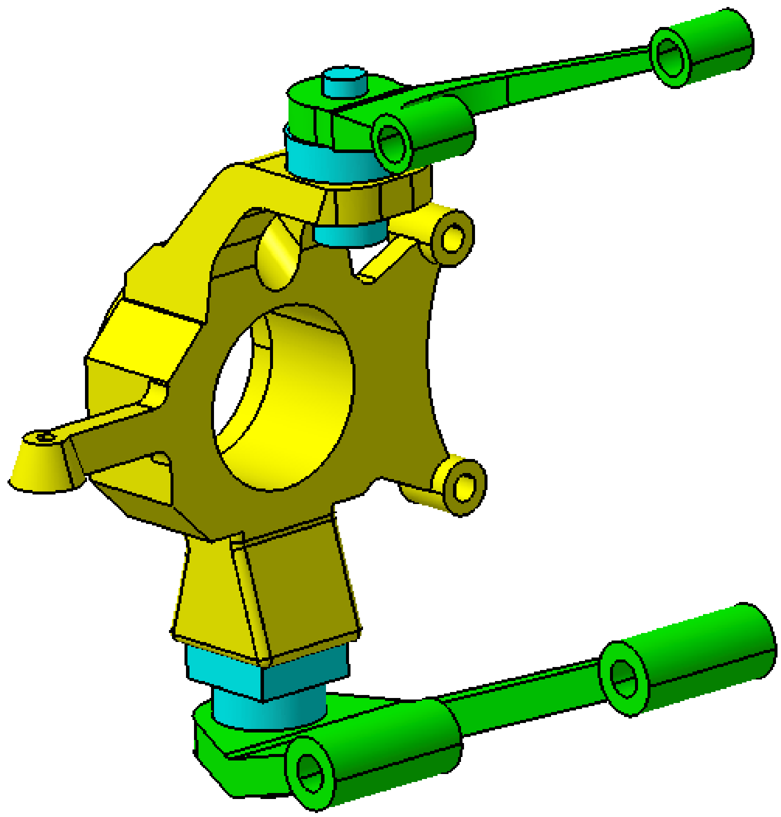
Open the Assembly document from the *Assembly* directory.



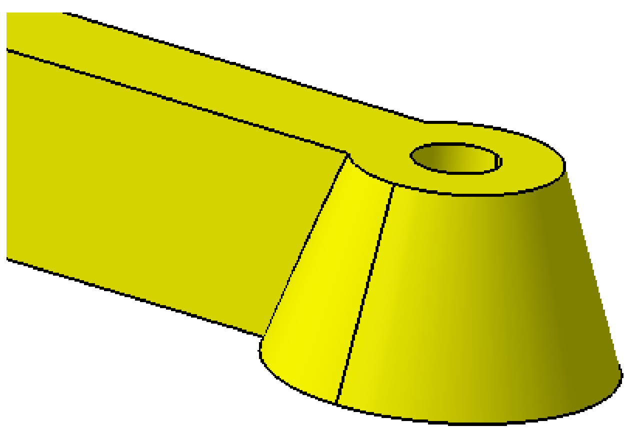
You will practice manipulating with this model.

Switch to the Assembly Design workbench if you are not already there. Remember, you can do this by selecting the Start pull down menu and then selecting *Assembly Design* from your favorites or from the *Mechanical Design* solution.

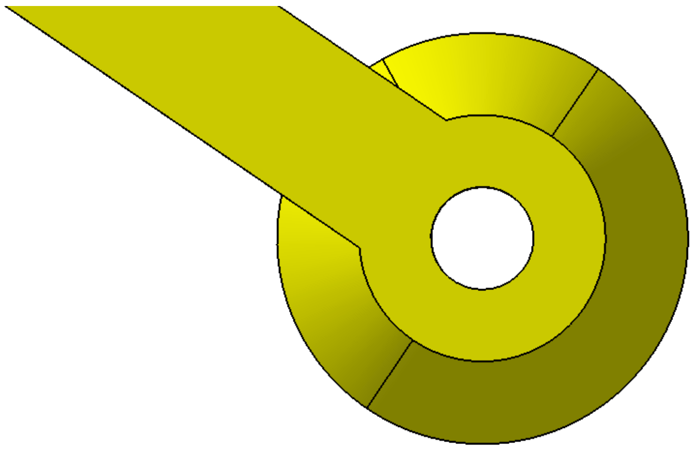
Rotate the model around as shown.



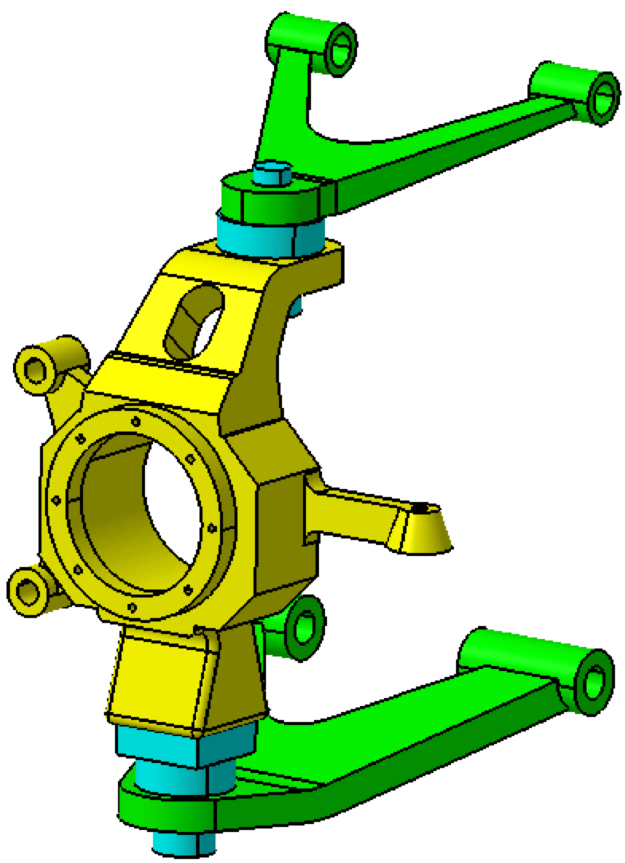
Rotate and zoom the model as shown. You may want to adjust the center of rotation by using the middle mouse button so that it will be easier to zoom and rotate the model as shown.



Rotate and zoom the model as shown. You may want to adjust the center of rotation to be somewhere close to the hole in the model by clicking and releasing the middle mouse button.



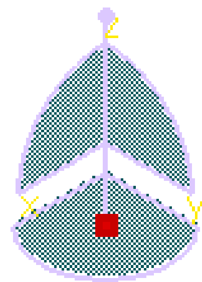
Manipulate the model as shown.



Manipulating with the mouse is very important within CATIA V5. You will want to practice this until you have it mastered.

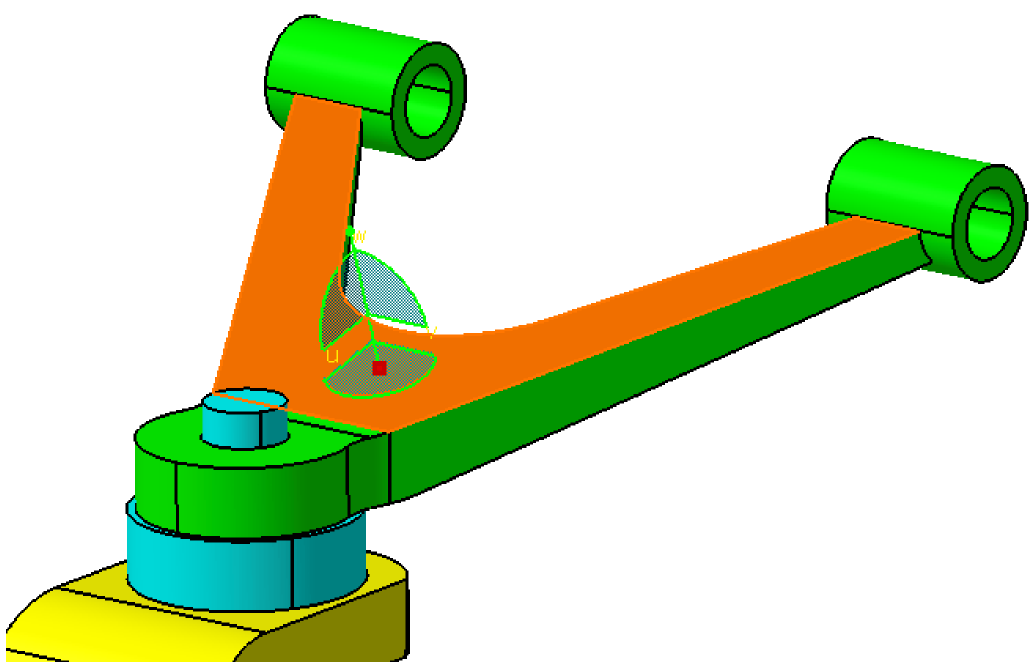
Compass

You can also manipulate a model with the compass. The compass can be used to move parts around within an assembly as well.



You can select on and drag the line and arc segments of the compass in order to translate and rotate the model respectively. You may also select and drag the vertex on the top of the compass to manipulate the model around.

Select and hold the red dot in the middle of the compass with the first mouse button. Drag the compass and drop it onto the face as shown.

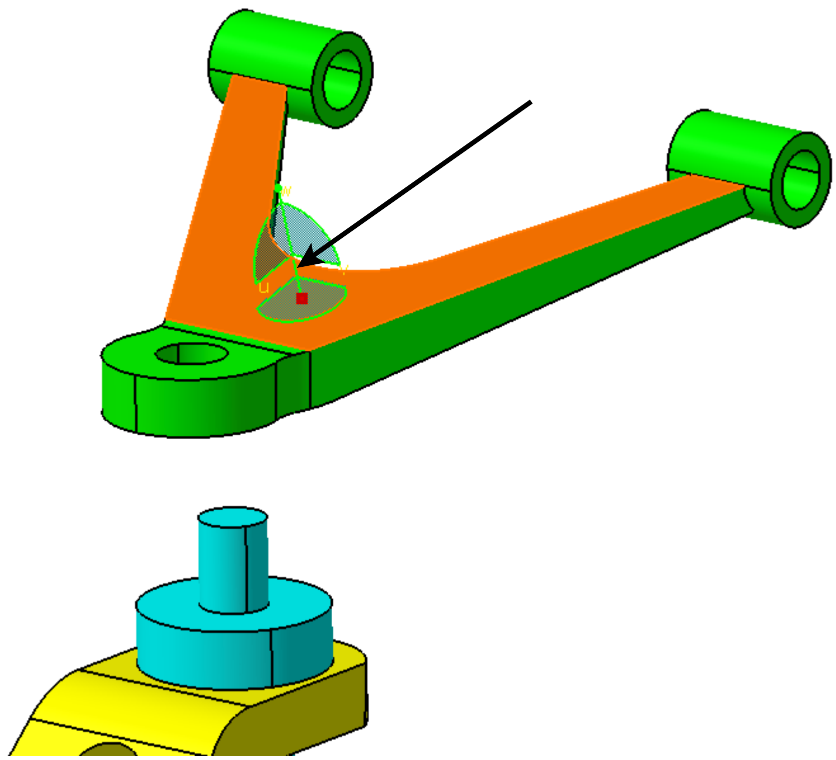


The compass should snap to the face and turn green. When the compass is green, that means that it is attached to whatever is selected.

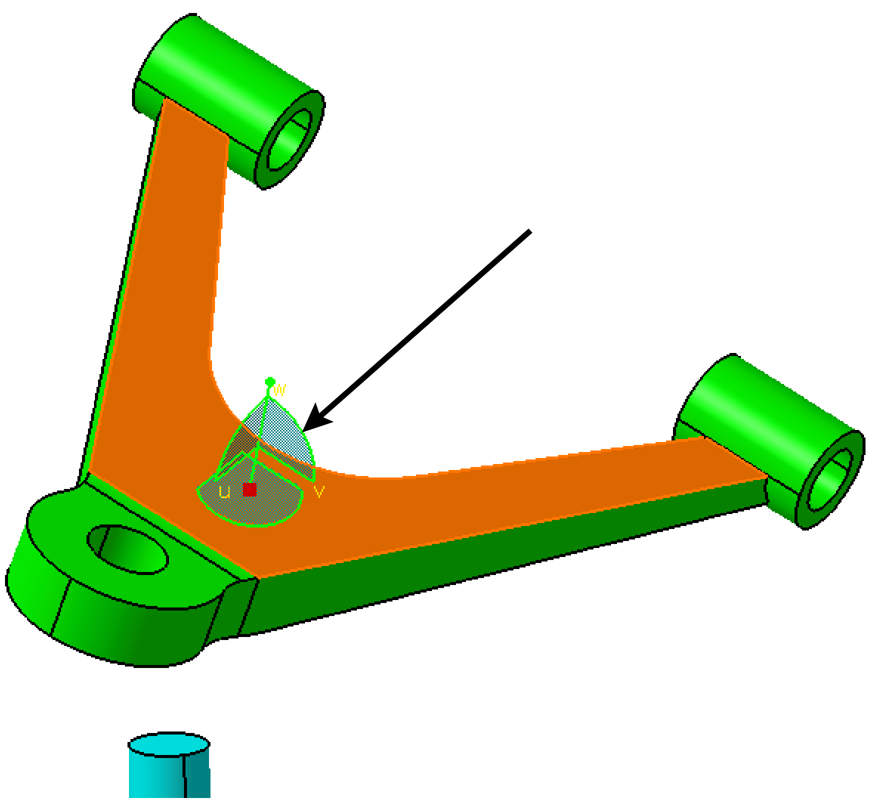
Select out in space. The compass turns white. It is no longer attached to anything.

Select the same face that the compass is sitting on. It should turn green again.

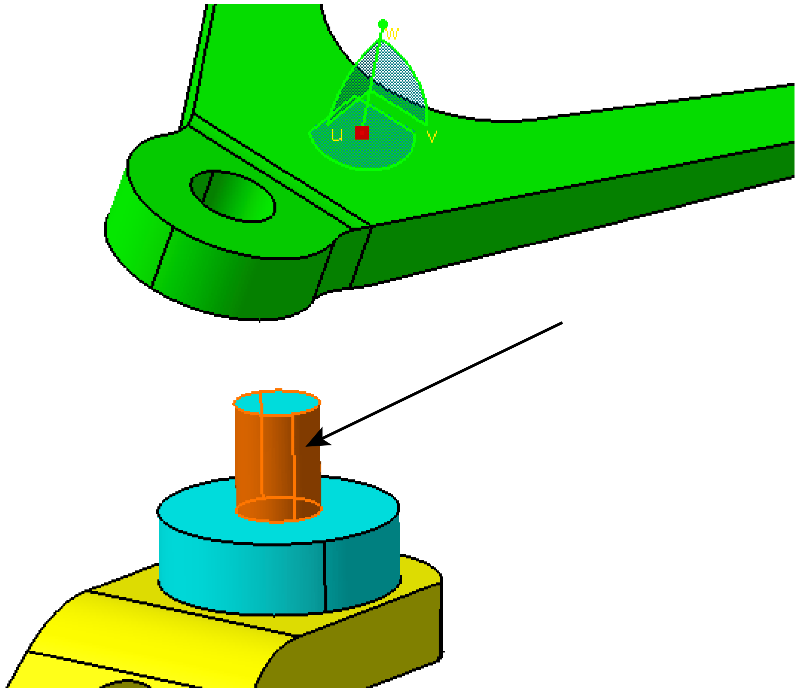
Select the *w* axis line as shown below and drag the axis up as shown. The Upper Arm should move with the axis.



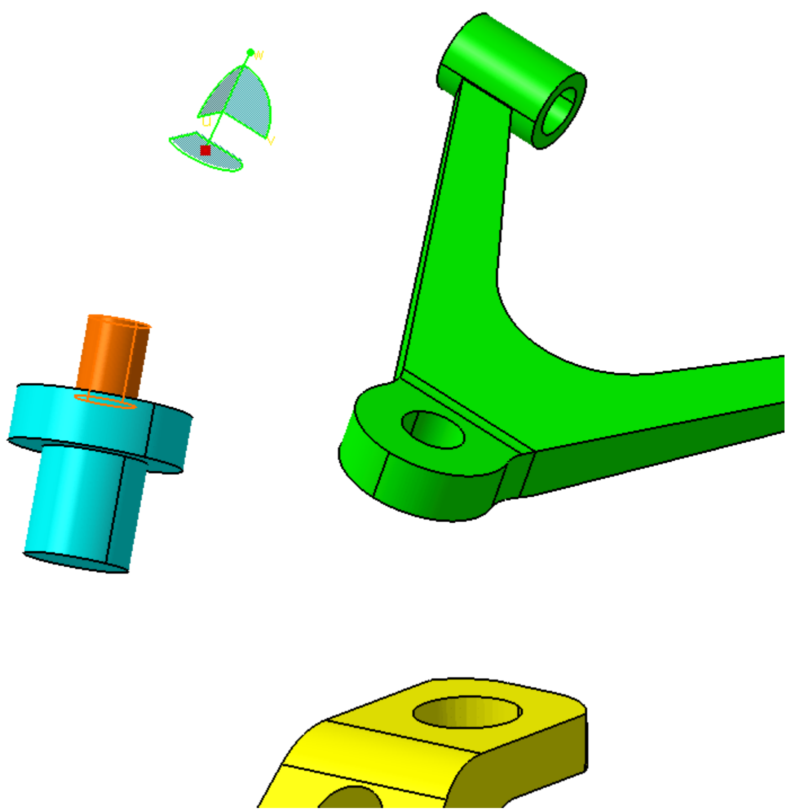
Select the arc as shown and drag the compass to rotate the Upper Arm around as shown.



Select the Upper Fastener as shown. The compass will now be attached to the Upper Fastener, even though it is still located on the Upper Arm.



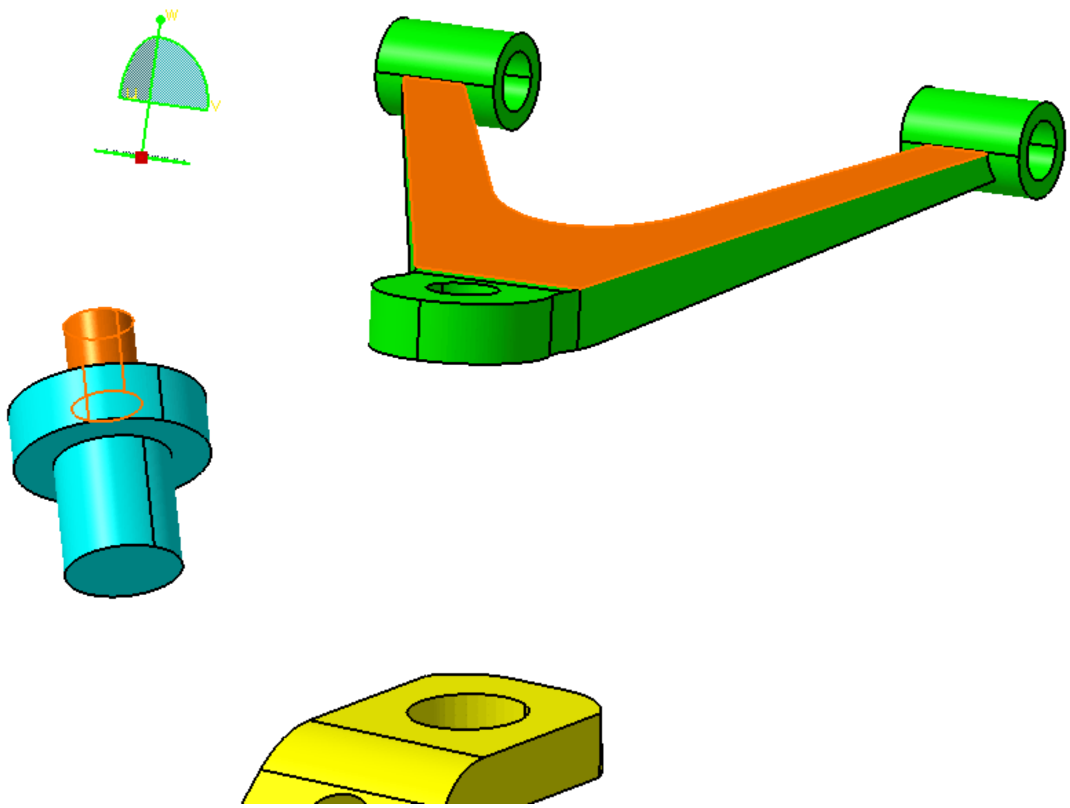
Move the compass around and notice that the Upper Fastener moves with the compass.



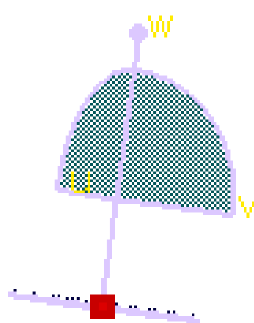
You can also select multiple parts to manipulate with the compass.

Select the face on the Upper Arm that you had previously selected. Hold down the Ctrl key and select the face on the Upper Fastener that you had previously selected.

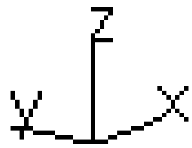
Move the compass around. Notice that both of the parts move with the compass now.



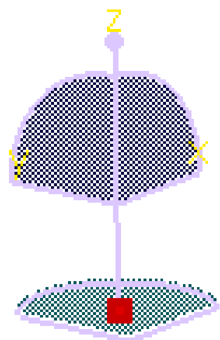
Select the red dot in the center of the compass and drag and drop it into its location in the upper right hand corner of the display. Notice that the compass maintains the same orientation and is labeled u, v, w instead of x, y, z. This is because the compass is still in the local coordinates of the parts that you just moved.



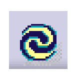
Drag the compass down to the lower right had corner of the display and drop in on the axis system.



The compass will switch back to the global coordinates of the assembly.



The compass makes it very easy to manipulate individual parts within an assembly.

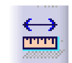
Select the Update All icon in the bottom toolbar.  The assembly snaps back together since it has assembly constraints on it that specify how it should be oriented.

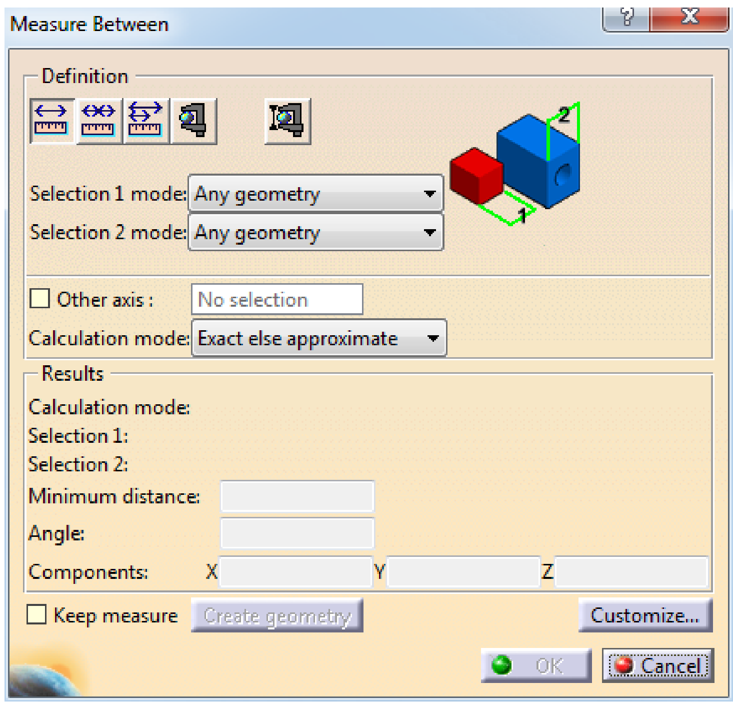
Measurements

Being able to analyze your models is very important. It is essential that you are able to determine if a part meets tolerance as well as where one part is located in relation to another. There are several methods that will be investigated for analyzing your models.

Measure Between

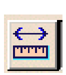
The measure between option allows you to measure the distance between any two elements. This option can be used for a variety of element types.

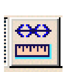
Select the Measure Between icon from the bottom toolbar.  The *Measure Between* window appears.



There are a number of different measurement options available under the measure between icon.

Definition

 Measures distance and angle between defined selections

 Sets the second selection as the first selection for the next measure

 Fixes the first selection so that you always measure from this item

 Switches to the measure item option

 Measures the thickness of an object

Selection 1 & 2 Mode

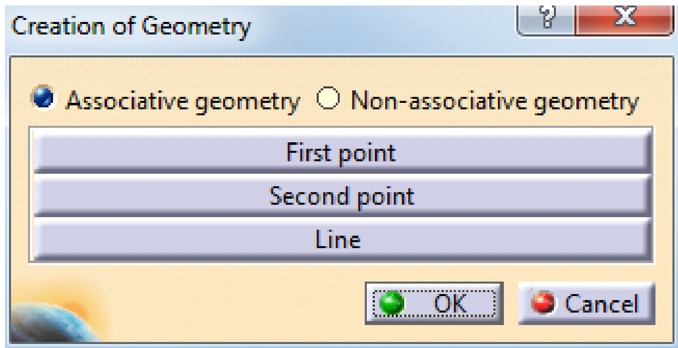
Any geometry	Measures the distance and angle between defined geometrical entities (points, edges, surfaces, etc.)
Any geometry, infinite	This is the same as *Any geometry*, except that it will extend the geometry infinitely and measure from the extension
Picking point	Measures the distance between points selected on defined geometrical entities
Point only	Measures the distance between points
Edge only, Surface only	Measures the distance and angle between edges and surfaces respectively
Product only	Measures between entire products
Picking axis	Measures between an element and an infinite line, perpendicular to the screen
Intersection	Measures the distance between intersection points, between two edges, or an edge and a surface. In this case, two selections are necessary to define an intersection.
Edge limits	Measures the distance between endpoints or midpoints of edges
Arc center	Measures the distance between the centers of arcs
Coordinate	Measures the distance between coordinates entered
Center of 3 points arc	Measures the distance between the centers of 3 point arcs

Other Axis Define what axis system you want to measure against

Calculation mode

Exact else approximate	Gives an exact measurement when it can otherwise it gives an approximate measurement
Exact	Gives only an exact measurement
Approximate	Gives only an approximate measurement

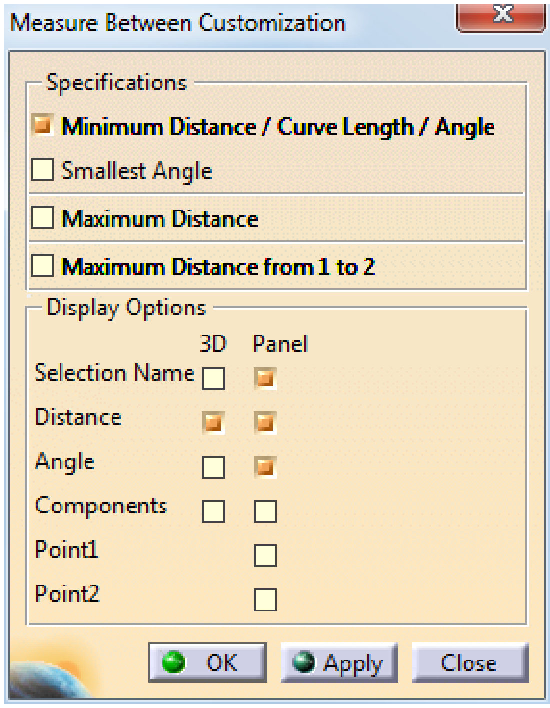
Results	This section varies according to what information you choose to display. Selecting the *Customize* button at the bottom of the window will allow you to change what shows in the window.
Keep Measure	This option will keep the measurement in the specification tree, as well as on the part
Create geometry	Creates corresponding geometry for the type of measurement you are performing. An example is shown below.



Associative geometry	Creates the geometry so that it is associated to the model
Non-associative geometry	Creates the geometry so that it will be a datum feature

The other options specify the actual geometry to be created.

Customize



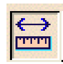
Specifications

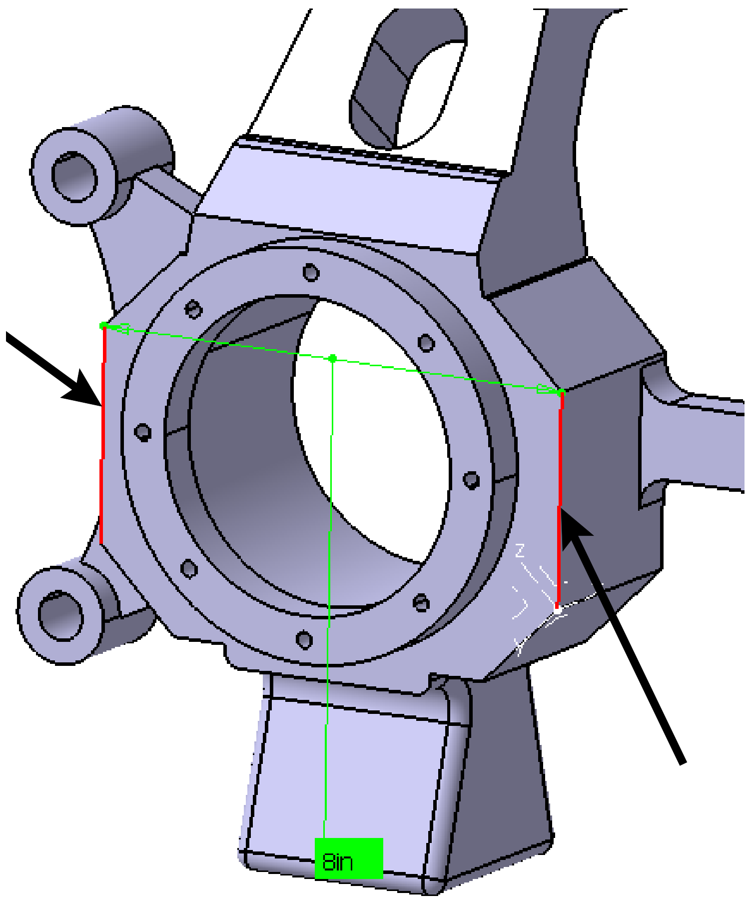
Minimum Distance/Curve Length/Angle	Measures the minimum distance between two selected elements and the angle
Smallest Angle	Measures the smallest angle independent of where you select the elements
Maximum Distance	Measures the maximum distance from selection 1 or 2
Maximum Distance from 1 to 2	Measures the maximum distance from selection 1

Display Options *3D* refers to the items appearing in the graphical workspace, whereas *Panel* refers to the items appearing in the *Measure Between* window

Selection Name	Displays the name of the elements selected
Distance	Shows the distance between the elements
Angle	Shows the angle between the elements

Components	Displays the X, Y, and Z components, relative to the axis, between the two selections
Point 1 & Point 2	Shows the X, Y, and Z location points of the element selected, where you selected it

Make sure the Measure Between option  **in the window is selected and *Selection 1 mode* and *Selection 2 mode* are set to *Any geometry*. With the *Keep measure* option turned off, select the two edges as shown below.** This will create a distance measurement as shown in the picture.

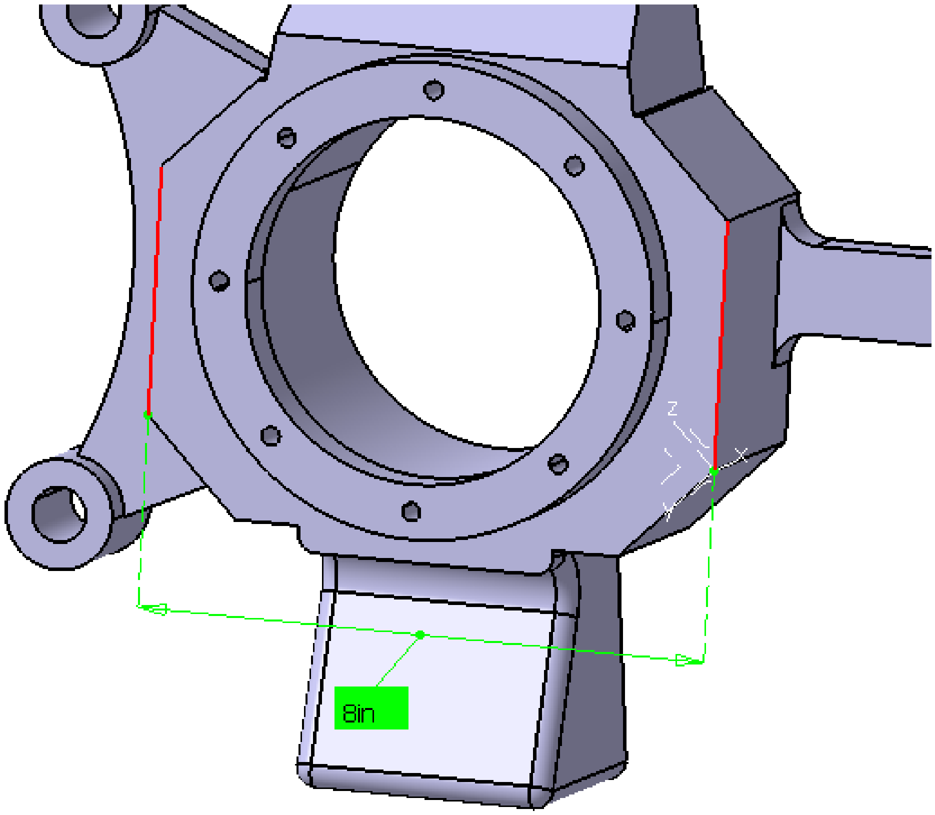


The position of the dimension can be changed.

Press and hold the first mouse button while on the measurement value and drag the text. This will allow you to move the dimension text but not the actual dimension.

Press and hold the first mouse button while on the dimension line and drag the dimension. This will allow you to move the dimension lines creating extension lines. You can select anywhere along the dimension line to move it.

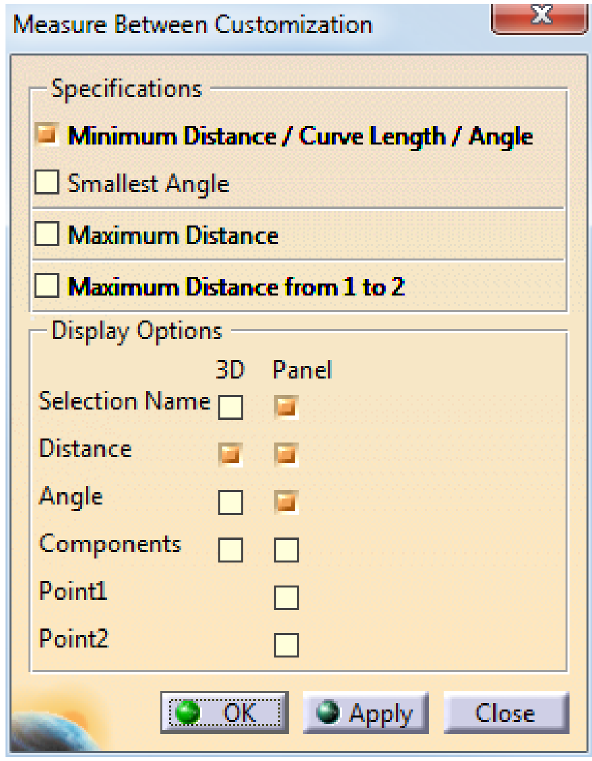
Move the dimension as shown below.



By dragging the dimension off of the part, it is easier to see everything. In larger assemblies you will find this to be a necessity.

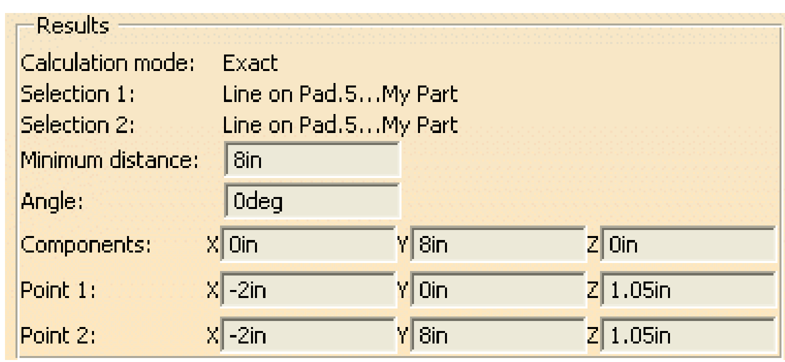
Whenever you measure your items, you can get a lot of information displayed in the *Results* area. Most of the time the distance and angle are enough, however, you can show more or less information.

Select the *Customize* button in the *Measure Between* window. The *Measure Between Customization* window appears.



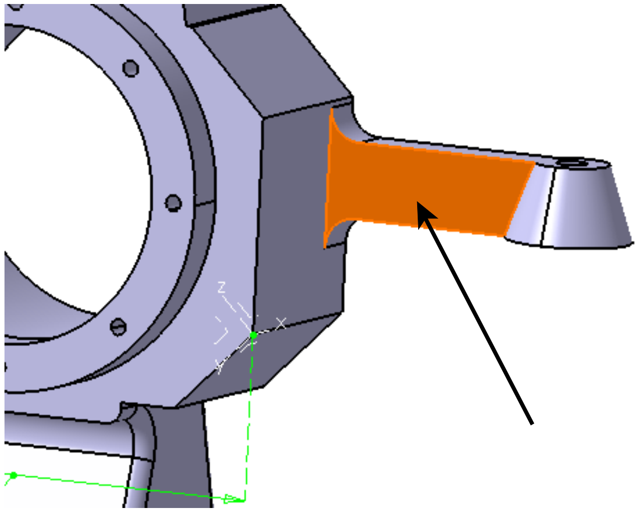
You can toggle the options on and off by selecting the boxes.

Turn on the *Components*, *Point 1* and *Point 2* display options under *Panel* and select *Apply*. You should notice that the additional information is shown in the *Measure Between* window. If you turn on the *Components* option under *3D* then the component information would appear in the graphical workspace.

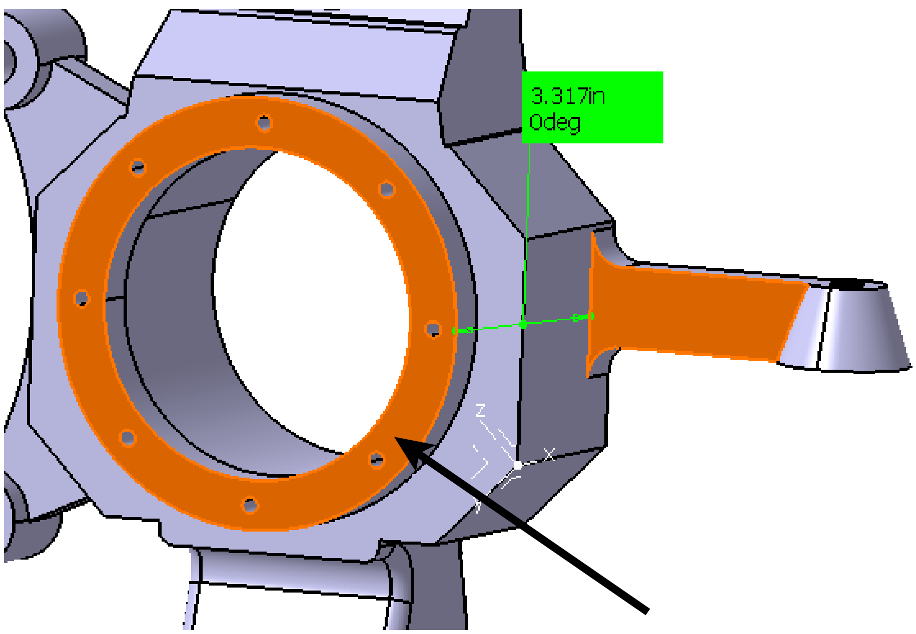


Turn off all of the display options except the *Distance* and *Angle* options under *3D* and *Panel* and select *OK.*

Select the face as shown.



Select the second face as shown. The measurement respects the limits of the faces and is actually measuring diagonally.

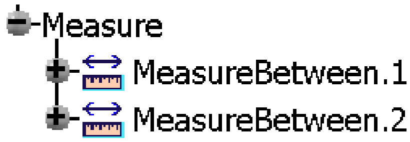


Notice the disappearance of the previous dimension. This is because the *Keep measure* option is turned off.

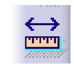
Select the *Keep measure* option. This option will keep the measurement after it has been created.

Create the first measurement again. This time, both measurements will remain. The measurements will also appear in the specification tree under the *Measure* branch.

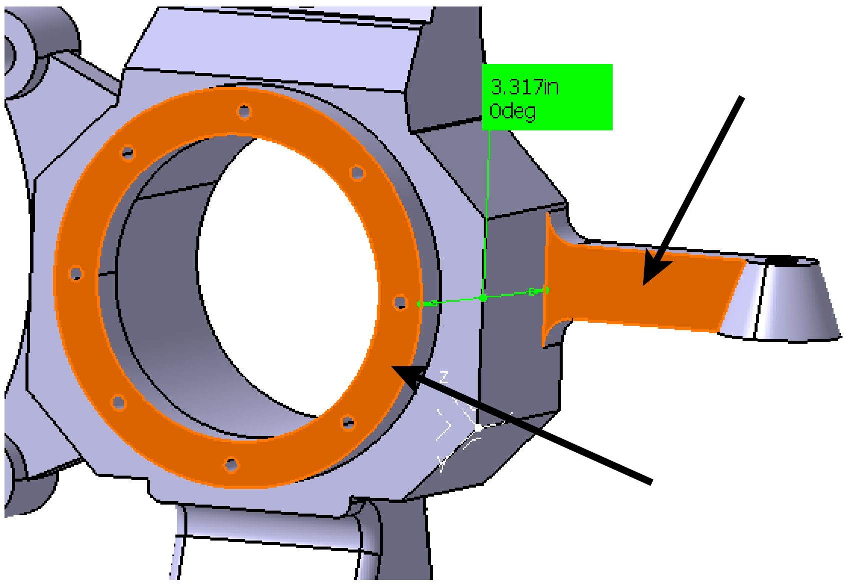
Expand the tree as shown below.



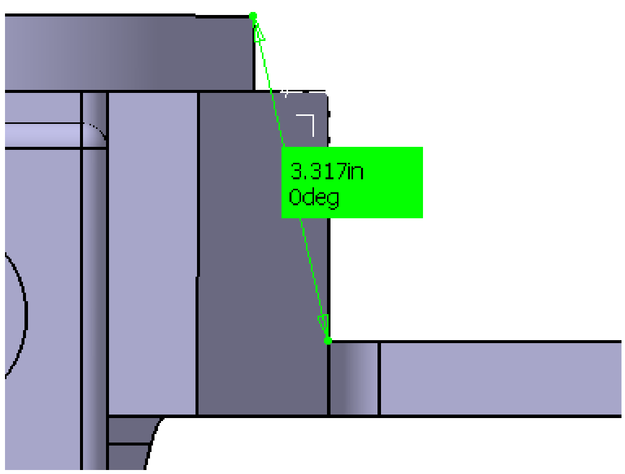
Select *Cancel*. Both of the dimensions will disappear. If you wanted to keep the dimensions, then you would select *OK*.

Select the Measure Between icon again. Turn off the *Keep measure* option. 

Select the two faces again as shown.

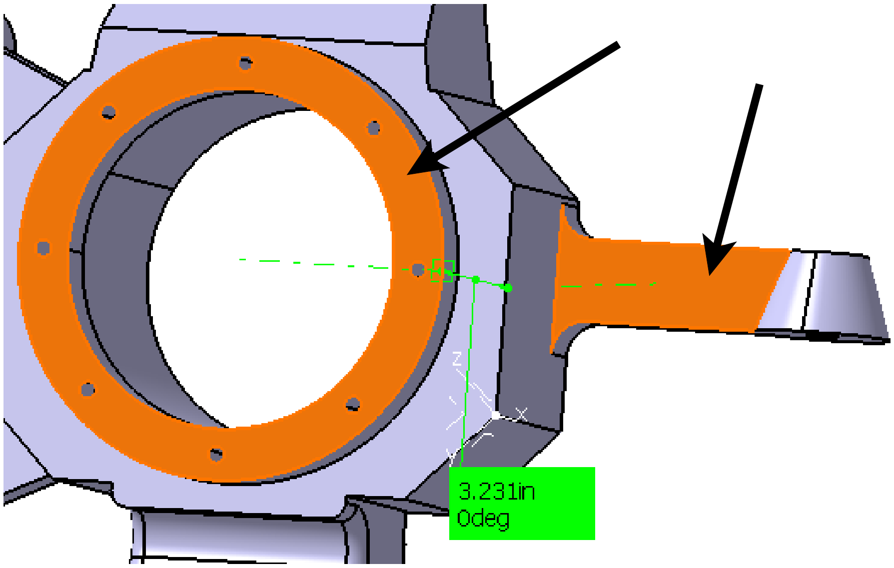


You are trying to get the normal distance between the two faces. However, as mentioned earlier, it is respecting the boundaries of the planar faces. This is not exactly what you want.

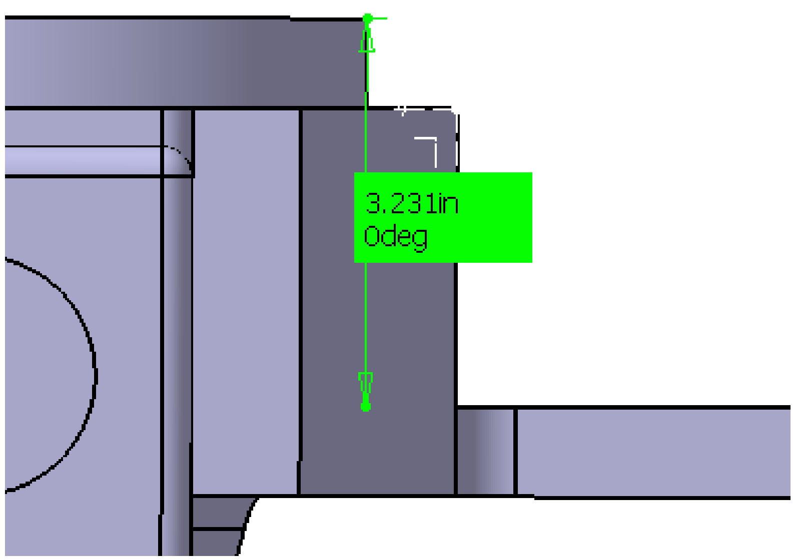


With the model rotated, it is much easier to see that this is not the measurement that we wanted.

Change both of the selection modes to *Any geometry, infinite* and select the same two faces. The dimension appears as shown below. This time it is the actual distance between the two faces. The *Any geometry, infinite* option extends the two planar faces out infinitely and then measures between them.

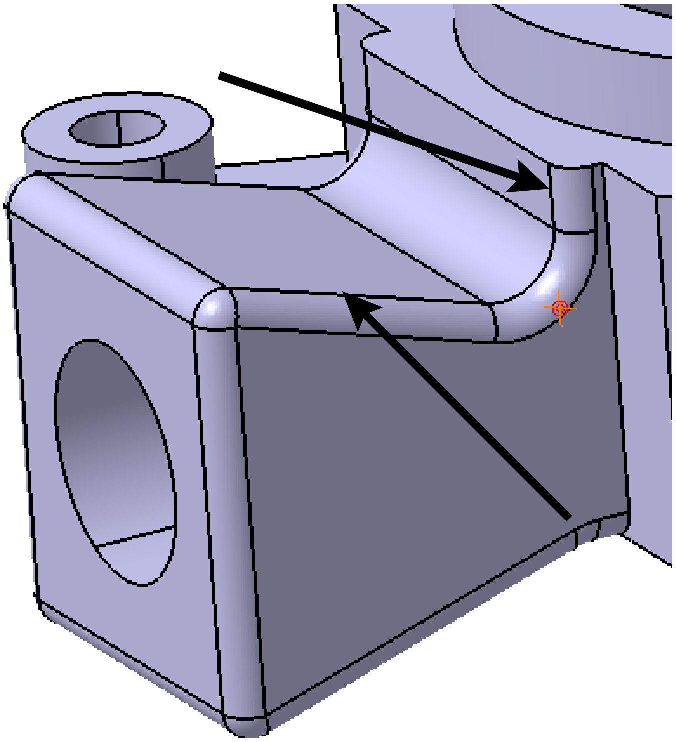


With the model rotated, it is easy to see that you are now getting a normal measurement between the two faces.

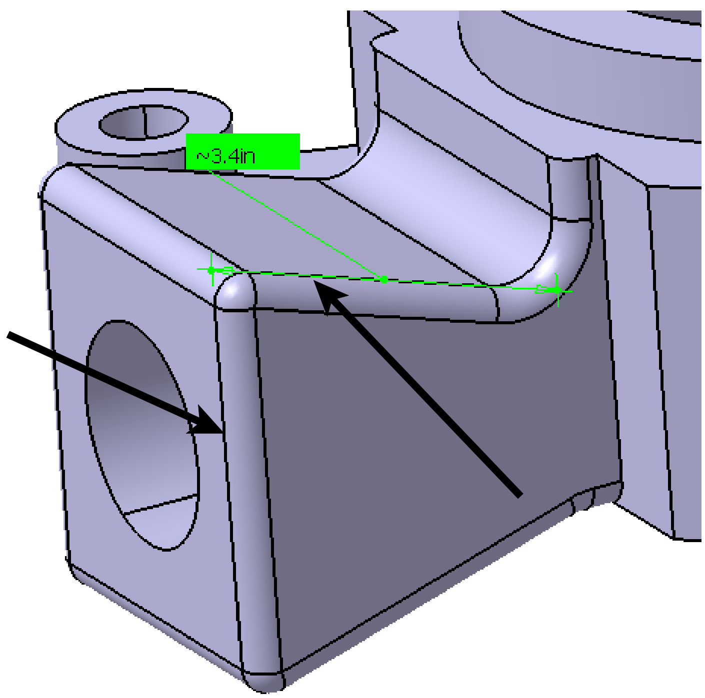


Change both of the selection modes to *Intersection.*

Select the two lines as shown below. This will compute the intersection point between the two elements and use that for the measurement. The intersection point will appear after you select the two elements.

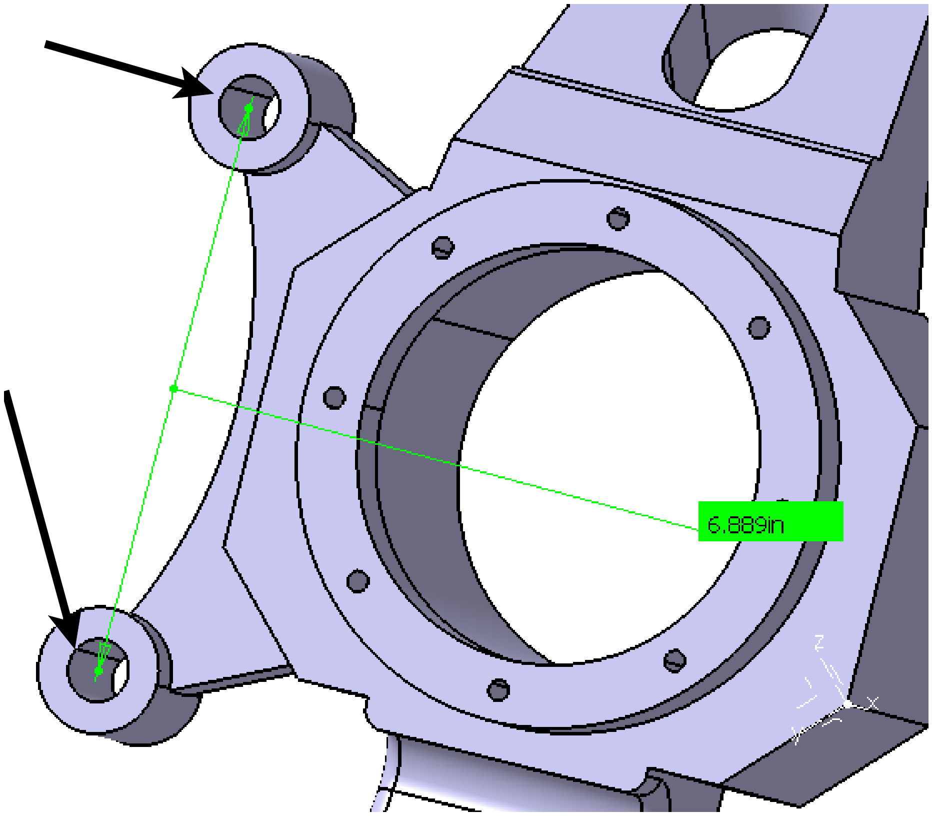


Select the two lines as shown below to identify the second intersection point.



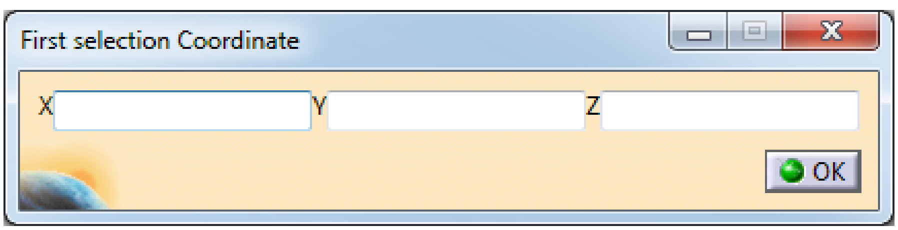
The dimension appears between the two intersection points. The measurement appears with a ~ symbol because it is an approximate measurement. Any time CATIA has to compute a location it will consider the measurement to be an approximate.

Change both of the selection modes to *Arc center* and select the two arcs as shown.



Notice that the dimension appears between the two arcs' center points.

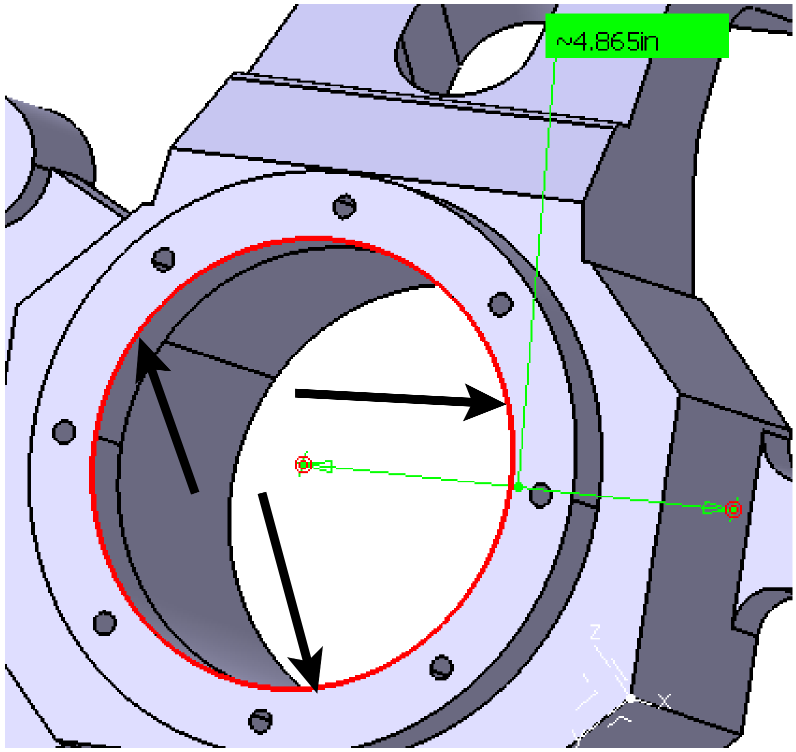
Change *Selection mode 1* to *Coordinate*. The *First selection Coordinate* window appears.



Enter 0, 0, 0 for *X, Y* and *Z* respectively and select *OK.*

Change *Selection mode 2* to *Center of 3 points arc*. You will need to specify three points that will be used to compute an arc and then the center of that arc will be used in the measurement.

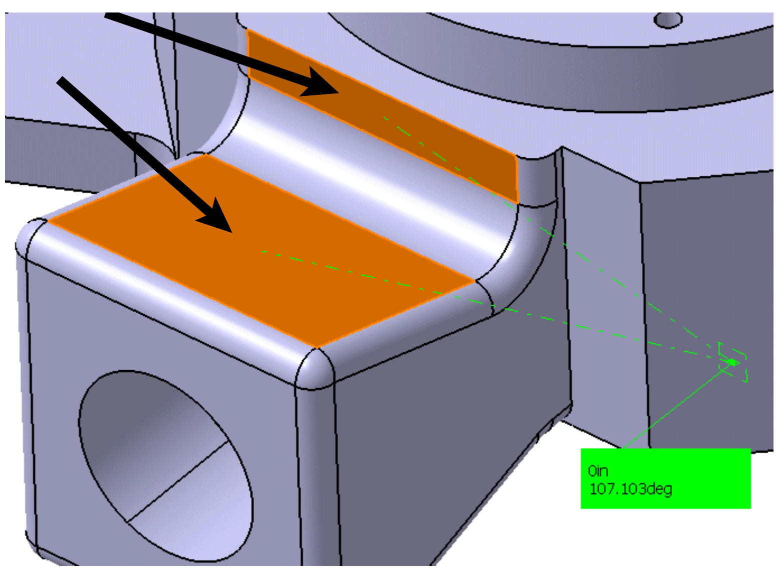
Select the three locations shown below. These are approximate locations.



The measurement is an approximate since it had to compute an arc.

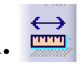
Switch the selection modes back to *Any geometry, infinite*.

Select the two faces as shown below.



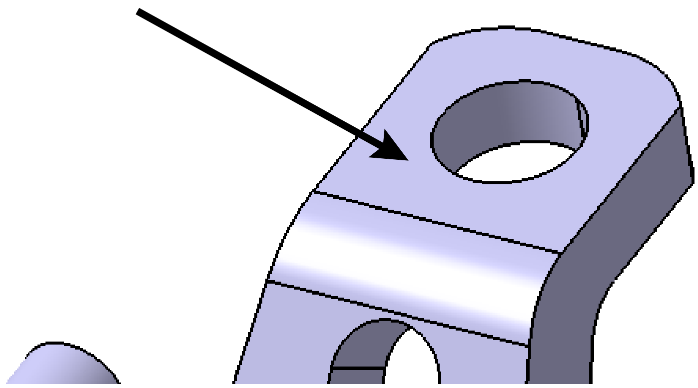
For this measurement, you are more interested in the angle than the distance.

Select *OK*. The measurement disappears since you did not have *Keep measure* turned on.

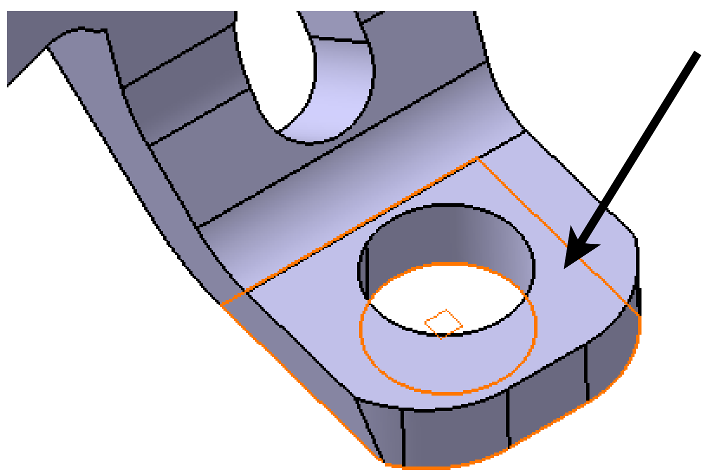
Select the Measure Between icon again and turn on the *Keep Measure* option. 

Select the Measure Between in Chain Mode icon from the window.  This option will allow you to measure in chain mode.

Select the face as shown.

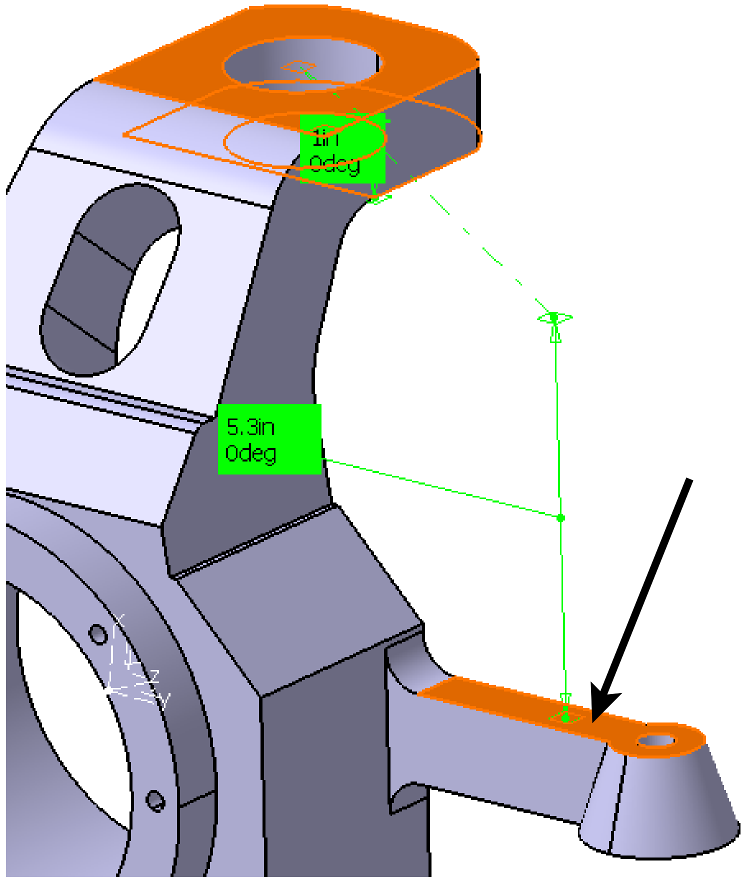


Rotate the model around and select the face as shown.

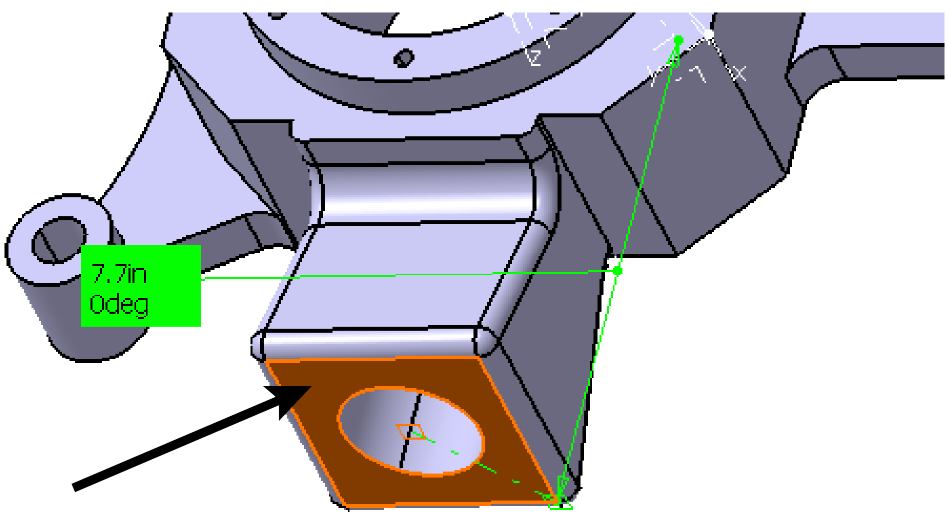


The measurement is created.

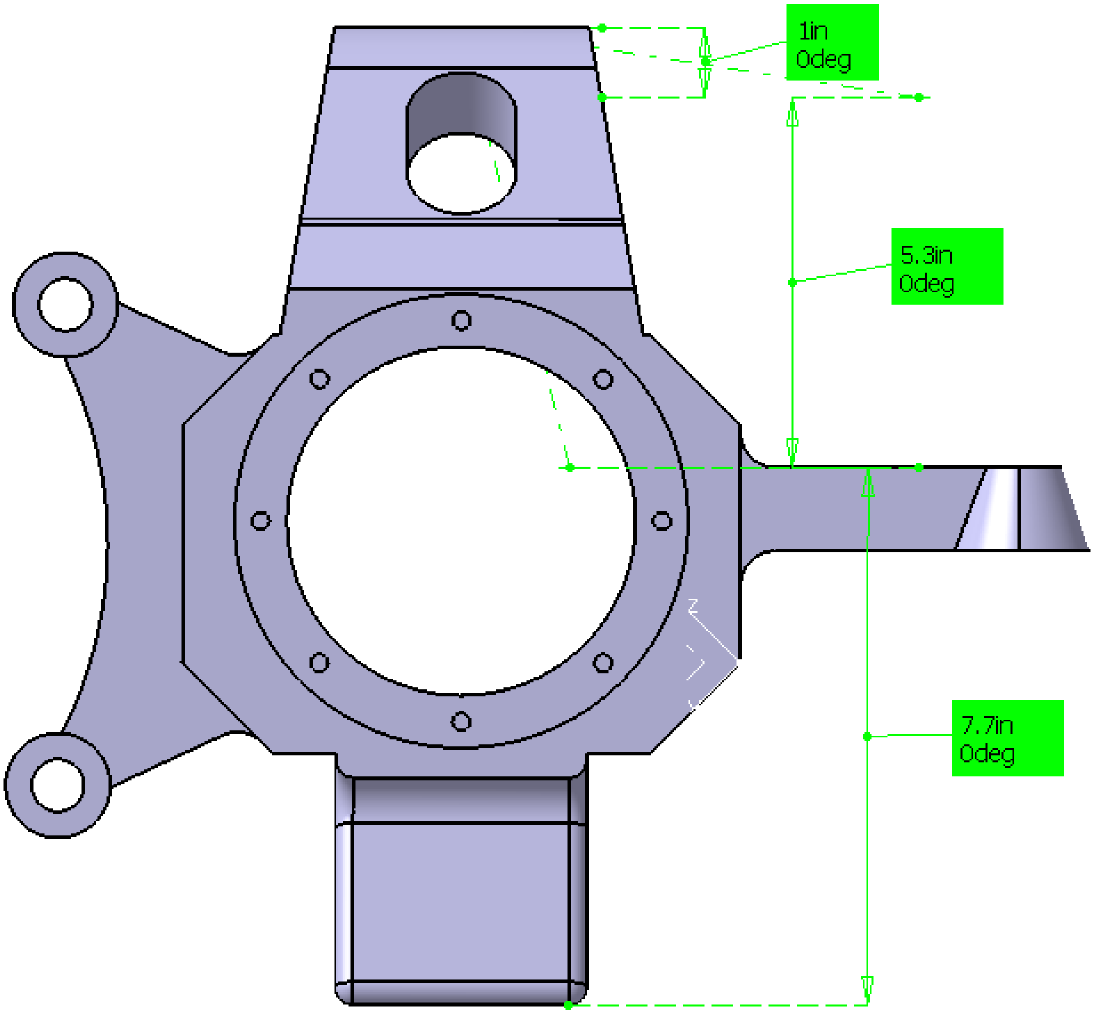
Rotate the model back around and select the face as shown. The measurement is created from where the first measurement ended.



Rotate the model around and select the face as shown. Again the measurement is created from where the previous measurement ended.



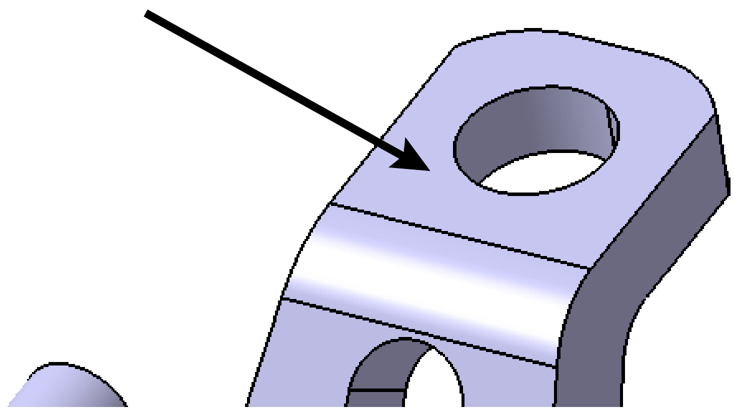
Notice that each measurement uses the previous measurements second selection for its first selection. The measurements have been moved around for clarity.



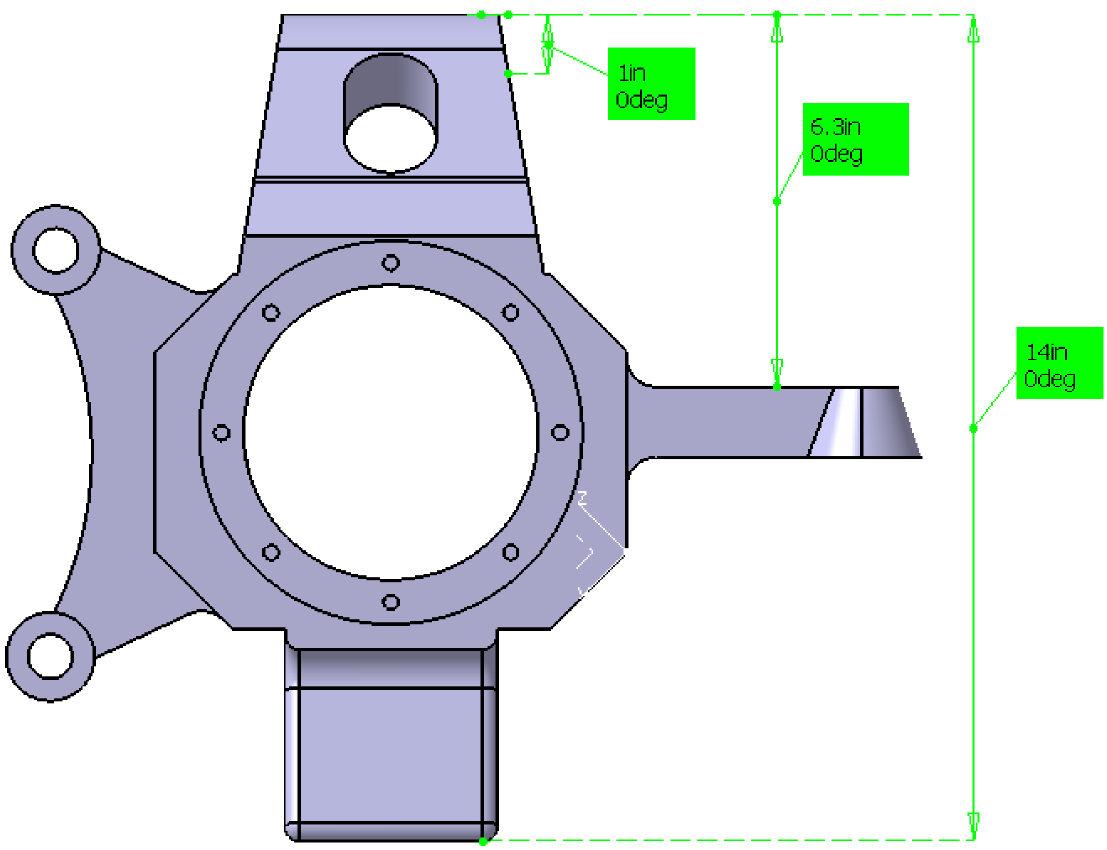
Select *Cancel*. The measurements disappear.

Select the Measure Between icon and select the Measure Between in Fan Mode option from the window.   This option will allow you to make all of your measurements based off of the first selection. This option is perfect for taking measurements from a datum.

Select the face as shown below.



Select the same three faces that you selected before when measuring in chain mode. The measurements should appear as shown. Again the measurements have been moved for clarity.



Select *Cancel*. The measurements disappear.

You will now create some measurements using the measure item option.