Tutorial 4

Assembly Structure and Drawing

Purpose:

To give the student the basic knowledge related to the creation of a constrained assembly structure and on how to make an assembly drawing.

Reference:

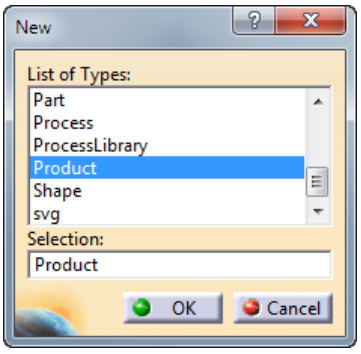
Use the ***04-Bearing_Support_Assy.pdf*** file, the related part files and the ***R20-D_Size.CATDrawing*** file.

1 – Launch CATIA®

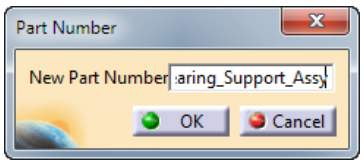
- If a product file is automatically created, keep it and skip Step *2*.

2 – Create a new product

- Use ***File>New*** to launch the ***New*** dialog box.
- Using the scroll bar, select ***Product*** in the list.



- In the ***Part Number*** dialog box, replace *Product1* by *Bearing_Support_Assembly.*



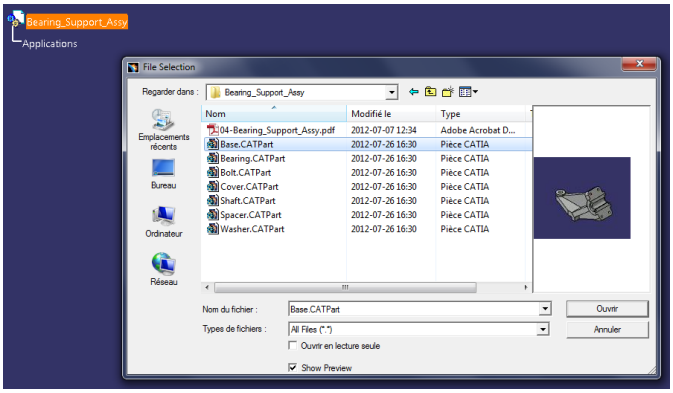
- Click the ***OK*** button to close the dialog box.

3 – If necessary, organize the environment

- If more information is necessary about this, review Tutorial 1.

4 – Load the first part alone, to set up the product coordinate system

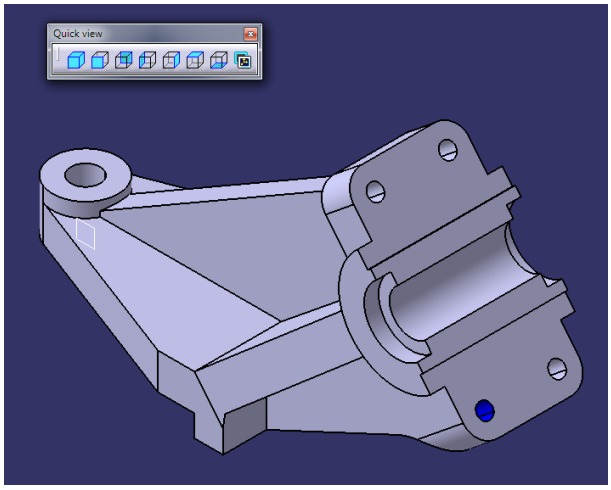
- Click the ***Existing Component*** tool icon  to load a part that has already be modeled and that is stored in a dedicated folder. After clicking, nothing happens until the user clicks the name of the product, at the top of the ***Product Specification Tree***. Then, a dialog box opens to allow the user to browse and select the part(s) that has (ve) to be inserted in the product.



- At this moment, one part only must be selected in order to give the user the opportunity to control the way the assembly will be displayed using the different view presets.
- Browse to locate the folder named ***Bearing_Support_Assy*** and select the *Base.CATPart* file and click on the ***Open*** button (Sorry for the illustration! A French version of Windows is used…)
- Pull the ***Quick View*** toolbar 

in the working area and make some visualization trials. The product reacts the same way the part reacts when it is open alone in a different window. (Try it if you are not sure).
- It is important to have a first part that realigns properly when the ***Quick View*** tools are used. If it is not the case, you will struggle when you will work in the assembly

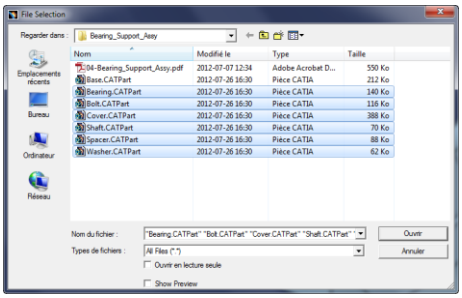
and you will struggle again when will come the time to extract assembly and detail views in the drawing. It is worth the time to make some tests at first to save time later on.



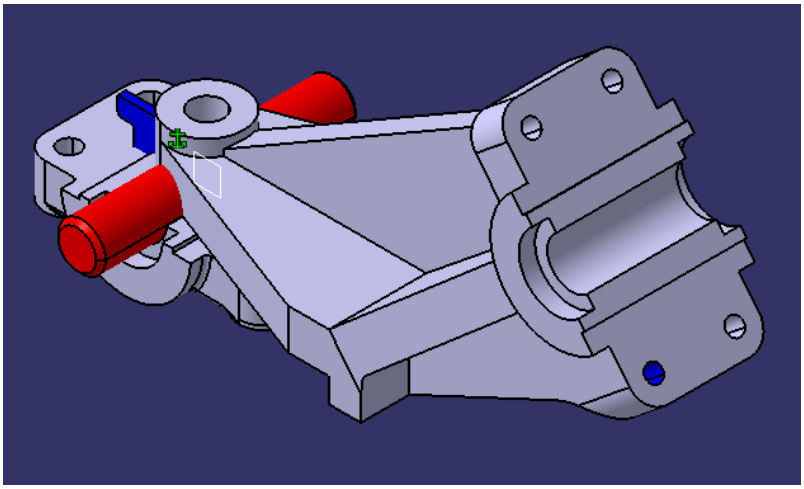
- Pull the ***Constraint*** toolbar  in the working area.
- Click the ***Fix Component*** tool icon  and select the base to make it the reference for the assembly.

5 – Load the other reference parts of the assembly

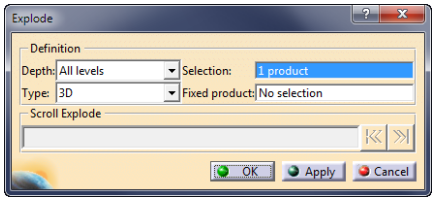
- Click the ***Existing Component*** tool icon  once again to select the rest of the parts.
- Click in the selection area to make it active and then, key in CTRL+A to select all files present in the folder.
- Press the CTRL key alone and unselect the *Base.CATPart* file that is already present in the product and the .pdf file that cannot be open in CATIA® and that will return an error message if you try to open it. Click on the ***Open*** button.



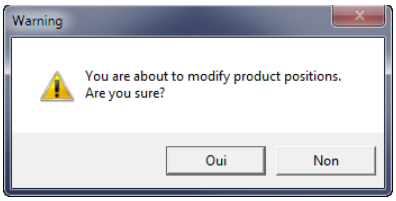
- All parts will appear in the product but will be superimposed since they all have been modeled close to the origin of their respective .CATPart file.



- Click the ***Explode*** tool icon  to move all the parts from their original location. Accept all default values by clicking on the ***OK*** button.



- A warning message will appear. Confirm by clicking on the ***Yes*** button (*Oui* in the illustration…).



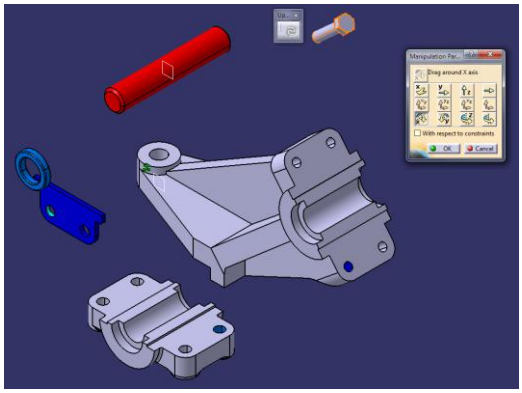
- All reference parts are now easy to see in the product structure. Note that the assembly needs an update. This is due to the displacement of the base that was already fixed. Click the ***Update all***  tool icon to relocate the base to its original location.

6 – Relocate the reference parts

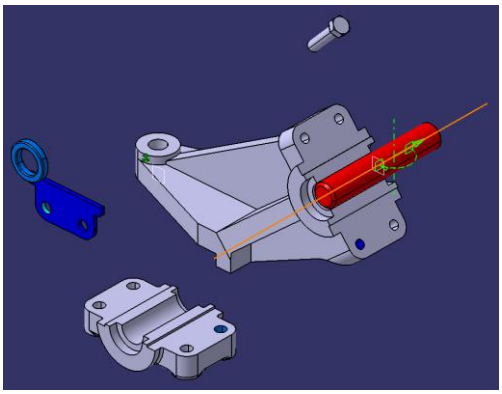
Since some parts may be difficult to see because they are hidden by other parts or are inside other parts, it is a good practice to move the hidden parts in order to identify them. Some tools are available to the user to accomplish this task.

- First, click the ***Manipulation*** tool icon  .
- In the dialog box that appears, click to select a linear translation, a rotation or a planar translation and then select and drag any part to a new location.

- Orient the bolt in order to relatively align it with the axis of the hole in the base part.

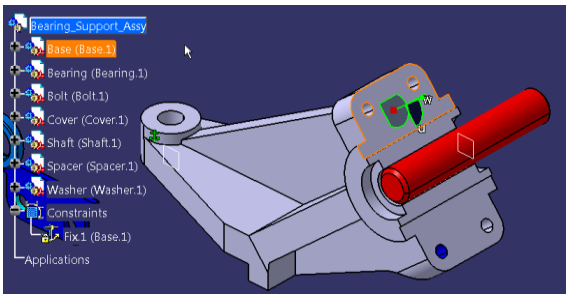


- Now, click the ***Snap*** tool icon  .
- Click the red shaft cylindrical face. A reference axis will appear. Then, click the base part cavity's cylindrical face that will receive the shaft. A location axis will appear allowing the user to flip the part. Click anywhere in the ***blue*** working area to complete the operation. The shaft will be properly aligned in its cavity, but no constraint will be created.



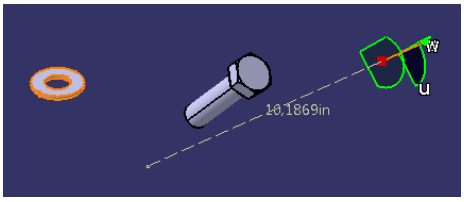
- Next, bring the mouse cursor over the ***red*** square found on the ***Compass***. The cursor will change for a four-pointing arrow icon.

- Click the ***red*** square and drag the ***Compass*** over one of the base part's inclined faces. Drop it.

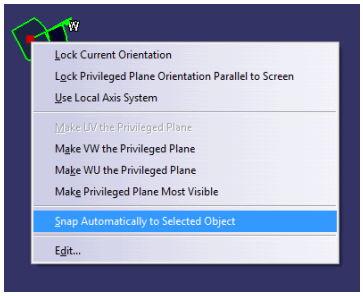


- The edges of the ***Compass*** turn ***green*** and the *Base* item is highlighted in the ***Product Specification Tree***. The part is now selected and ready to be moved. Click to select any ***green*** edge of the ***Compass*** and move the cursor to create a linear or rotative displacement. Since the base part is moved, the ***Update all*** tool icon  will turn on. Click the icon to update the product and return the base to its original location and click anywhere in the ***blue*** working area to deactivate the part.

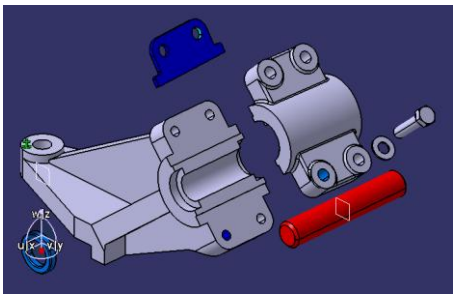
- Click the *Washer* item in the ***Product Specification Tree***. The ***Compass*** will turn on once again. Click any edge to drag the washer to a new position. The washer will move but the relative position between the washer and the ***Compass*** may make the operation more complex to visualize.



- The ***Compass*** could be dragged and dropped to any new part that needs to be moved in the product, but to make the user's life easier, it is possible to preset this behavior by accessing the ***Compass***' context menu and selecting the ***Snap Automatically to Selected Object***. To access the context menu, just bring the mouse cursor over the ***Compass***' ***red*** square and right-click.



- From now on, every time a part will be selected either directly (in the working area) or by its representation in the ***Product Specification Tree***, the ***Compass*** will relocate on the selected part making future transformation much easier.
- Roughly align all the reference parts. Make sure the cover hole having a ***blue*** inner cylindrical face is aligned to the corresponding base hole. Assembly constraints will align the parts but a 180° flip is always possible. Making the rough alignment will take a few minutes but will prevent a number of frustrations…



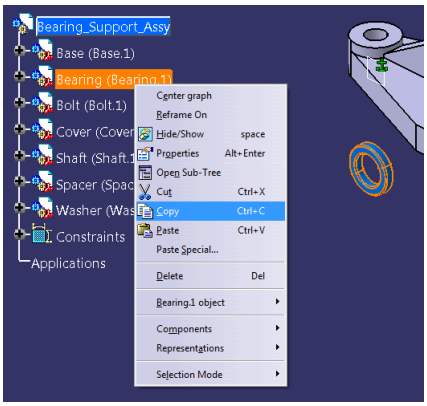
- Access the ***Compass***' context menu to unselect the ***Snap Automatically to Selected Object*** option and drag the ***Compass*** onto the ***XYZ*** tripod located in the bottom right corner of the screen. This will relocate the ***Compass*** to its original location.

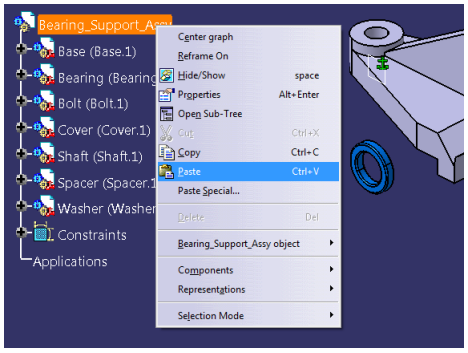
- Save the product.

7 – Loading the missing parts

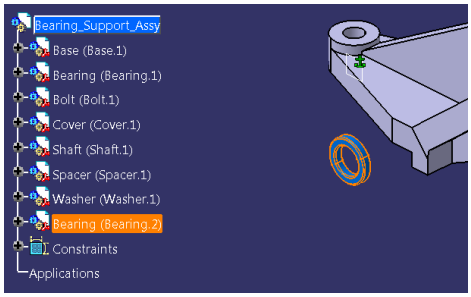
Some parts are still missing in the assembly. They could be loaded by using the ***Existing Component*** tool  , shown previously. This will load new instances of the selected parts in the product and would insert them at the exact same location where its reference part was when it was first loaded. Other options can be used to create new instances of a part already loaded.

- Bring the mouse cursor over the bearing item in the ***Product Specification Tree***. Right-click to access the context menu and select the ***Copy*** option.
- Move the cursor over the product name, at the top of the ***Product Specification Tree***. Right-click to access the context menu and select the ***Paste*** option.

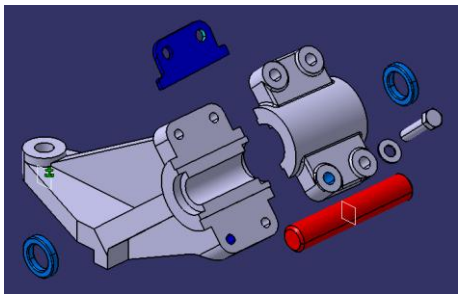




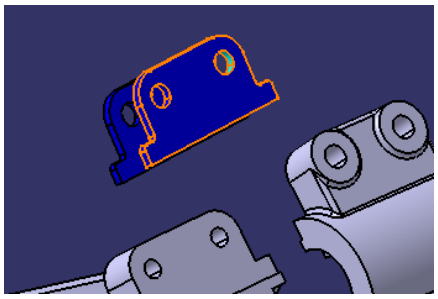
- Note that a new bearing instance is created at the end of the tree (Bearing.2). This new instance is superimposed to the first one (Bearing.1).



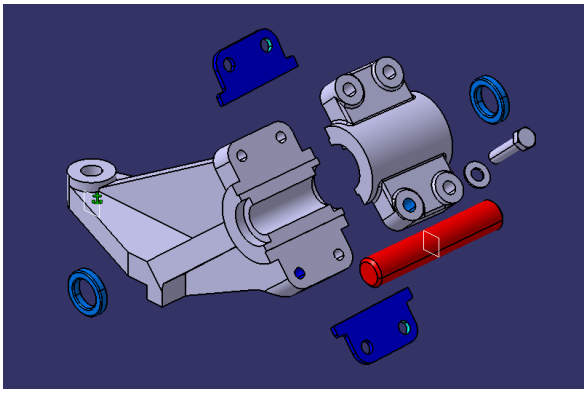
- Use one of the means shown previously to relocate the second bearing on the other side of the support.



- Click the ***Fast Multi Instantiation*** tool icon  and select the spacer to create a second instance. The new spacer is automatically offset from its reference. Using this tool is definitely the optimal way to insert missing parts in an assembly since it is possible for the user to double-click the icon at first to keep the tool active and then make new instances of any missing part.

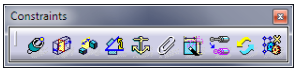


- Use one of the means shown previously to relocate the second bearing on the other side of the support.

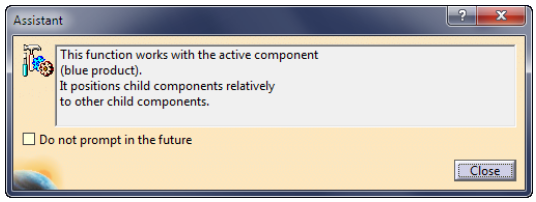


- It is not necessary to create new instances for the washer and the bolt. A different approach will be used to insert the missing parts.

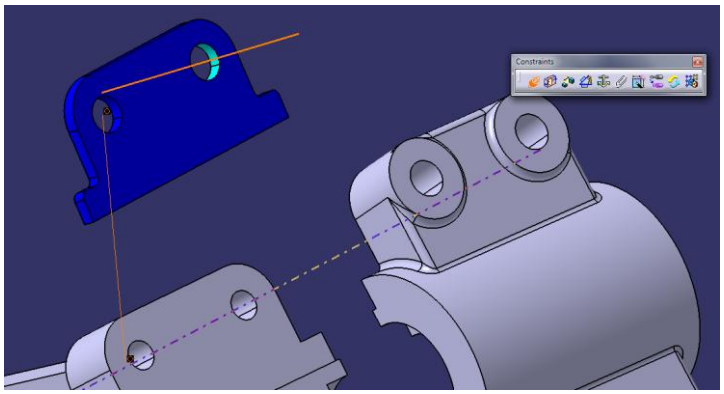
8 – Creating assembly constraints

- Pull the ***Constraints*** toolbar 

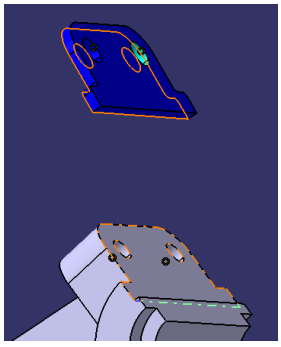
in the working area. Now that more than one part is present in the product, all constraining tools are available to the user.
- Double-click the ***Coincidence Constraint*** icon  to keep the tool active.
- The ***Assistant*** dialog box will pop-up to warn the user about the fact that constraints are created in the active product or sub-product only. Read and understand the message and click the ***Do not prompt in the future*** checkbox and close the dialog box.



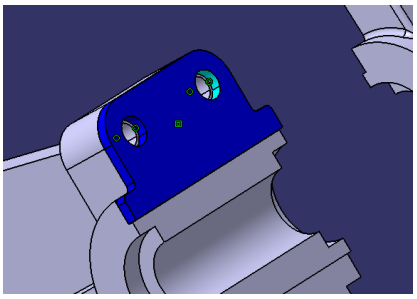
- Create two coincidence constraints between the two top spacer holes and the two corresponding base holes. Select the cylindrical face of a hole to access its axis. If the axis does not show up, zoom in to make the selection. It is a graphical limitation.



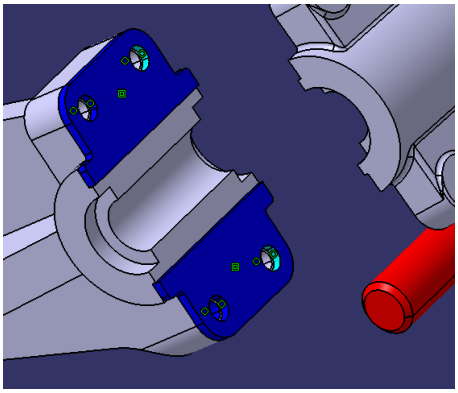
- Click on the ***Coincidence Constraint*** icon  once again to deactivate the tool.
- Click the ***Contact Constraint*** tool icon  and select the bottom face or the spacer and the base face that will receive it.



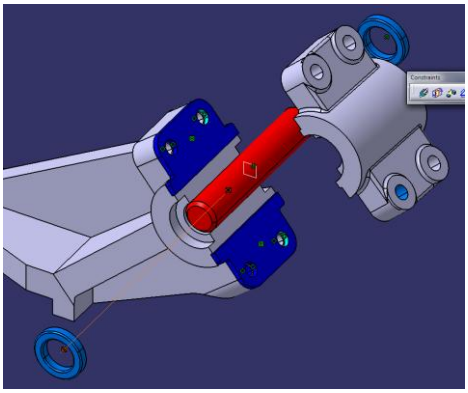
- Click the ***Update All*** tool icon  to force the product's regeneration and bring the spacer to its final location.



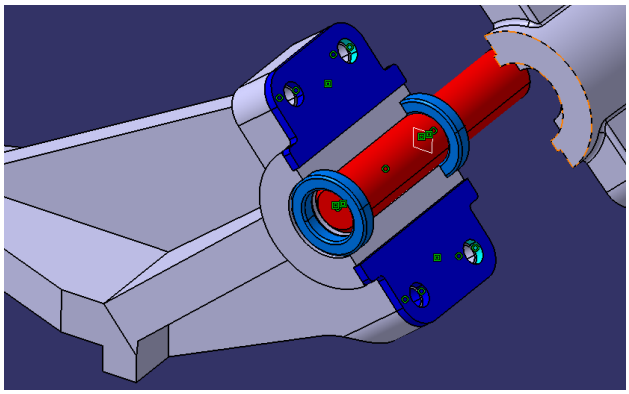
- Repeat the previous operations to constrain the bottom spacer.



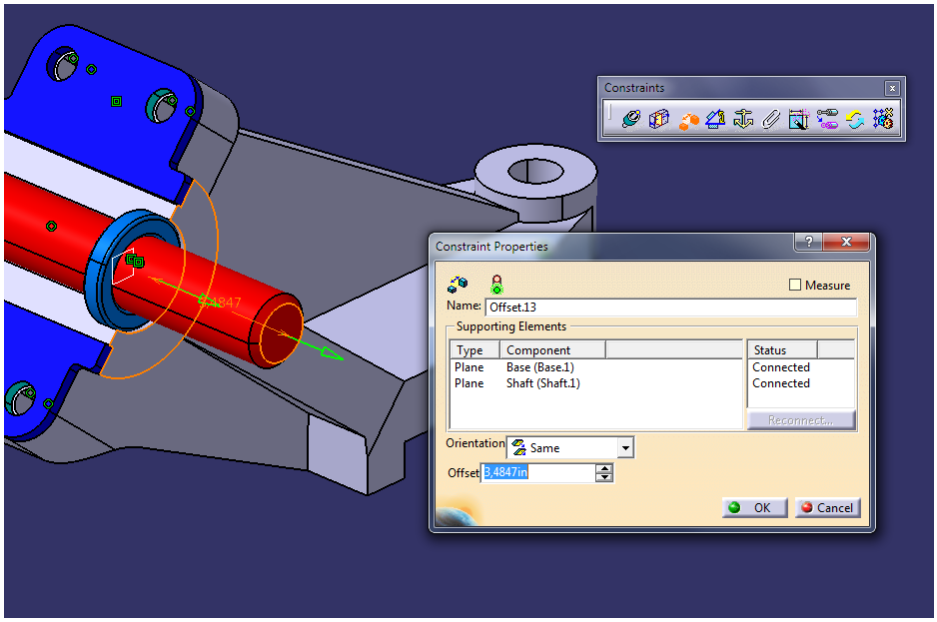
- Use coincidence constraints to bring the shaft and the two bearing aligned on the base cavity's axis. Update the product.



- Use contact constraints to locate the two bearings in their seats. Update the product.

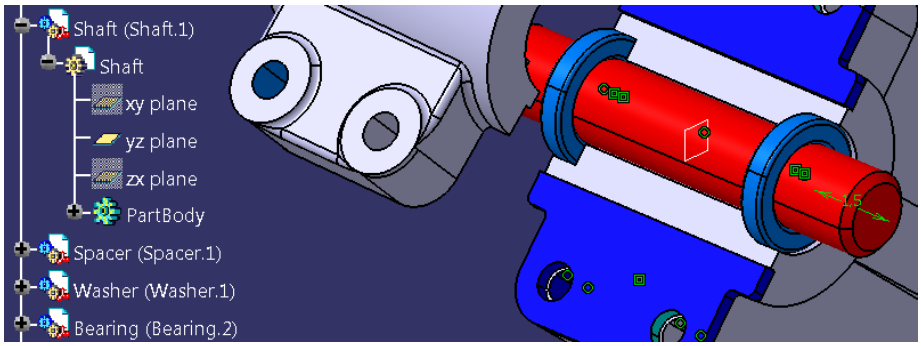


- Click on the ***Offset Constraint*** tool icon  , rotate about the product and select the base's side face and the protruding shaft's end face. A dialog box will pop-up.

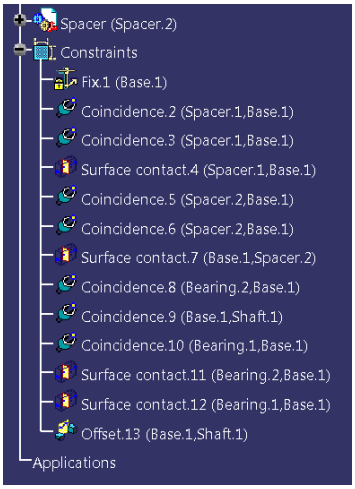


- In the ***Offset*** edition box, a value is highlighted in ***blue***. It may be positive or negative depending on the order taken to select the two offset faces. To decrease the protruding value make sure to keep the same sign on the value inserted. In this scenario *+1.5* inches should be used to center the shaft in the base.

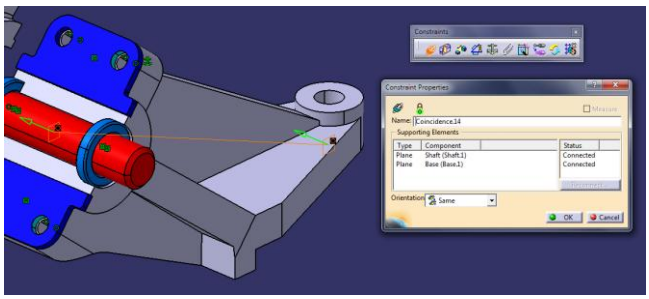
The problem with this approach is that the shaft will no longer be centered if its length is modified. In the original shaft part, the revolution was made symmetrical according to the origin, making its *yz* plane centered on the part.



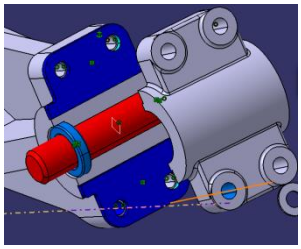
- Click on the *plus* sign found on the ***Constraints*** item's left side in the ***Product Specification Tree***. This will expand the ***Constraints*** group.



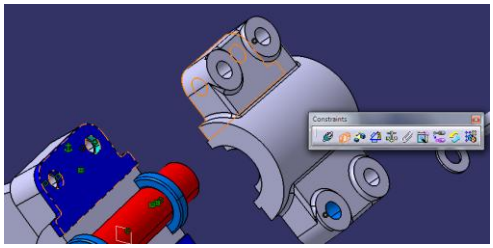
- Select the offset constraint, at the end of the list, and delete it.
- Create a coincidence constraint between the shaft's plane and the base's plane found at the other end of the base. No update should be necessary since the shaft was already centered.



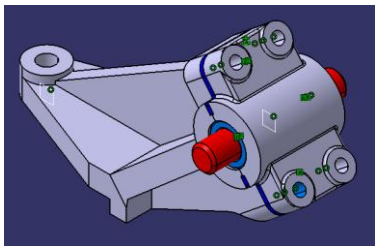
- Create coincidence constraints between two cover holes and their two corresponding holes on the base.



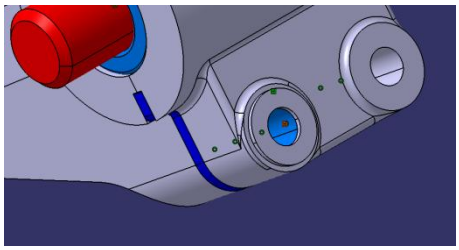
- Create a contact constraint between one of the spacers and the cover's corresponding face.



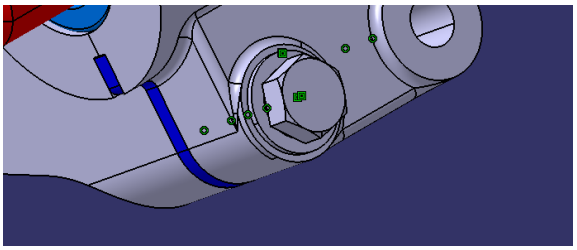
- Update the product.



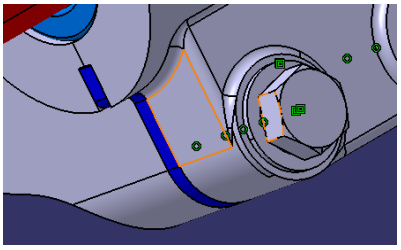
- Use a coincidence and a contact constraint to constrain the washer on the cover blue hole's spotface. Do not forget to zoom in to ease the selection if necessary.



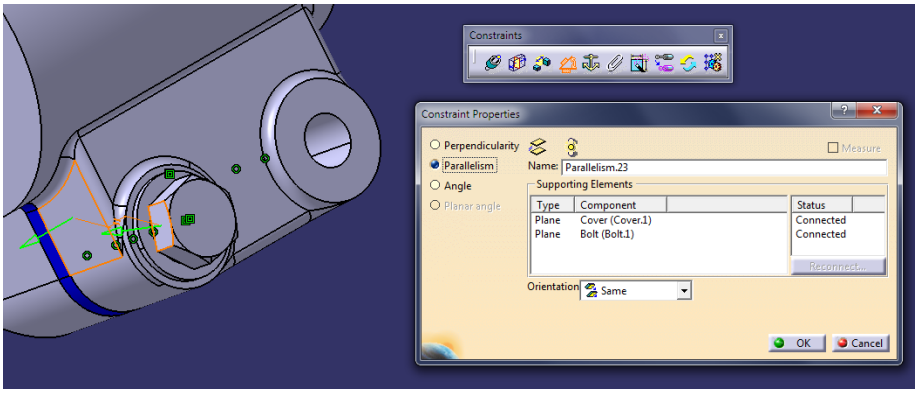
- Repeat the operations for the bolt.



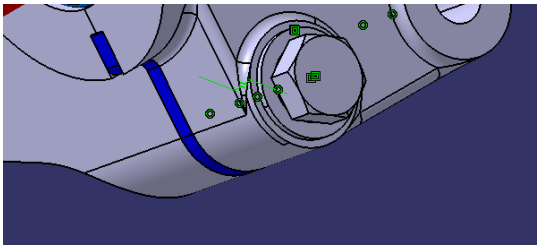
- Click the ***Angle Constraint*** tool icon  and select the cover's side face and the bolt head side's face that is the most parallel to it.



- In the dialog box, check the ***Parallelism*** check box and click the ***OK*** button.

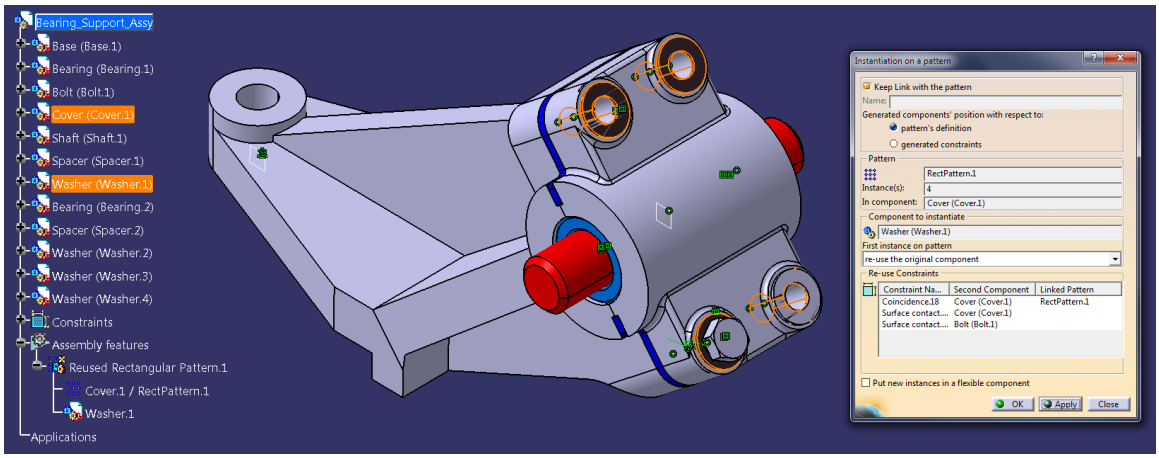


- Update the product. The bolt head is aligned.

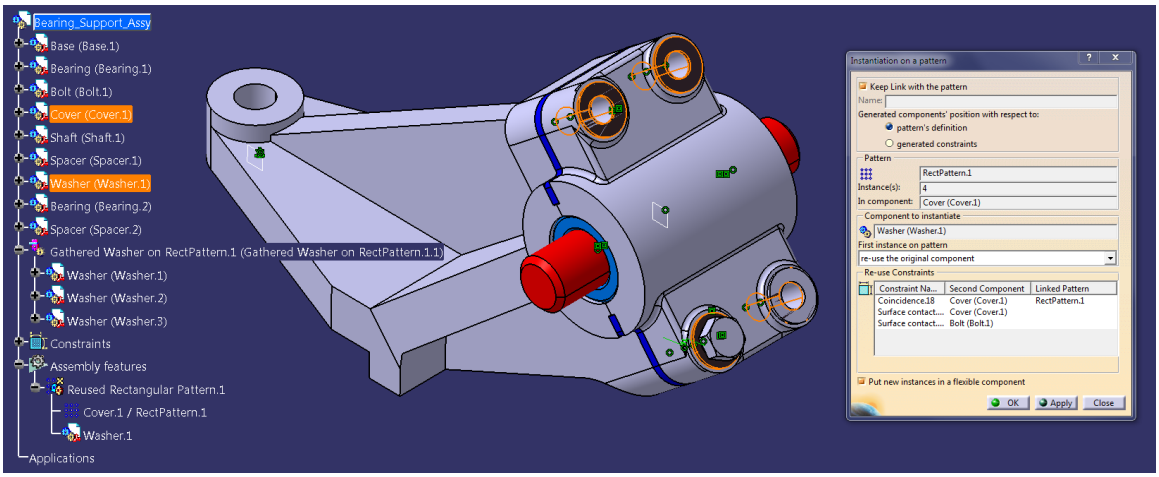


9 – Generate pattern elements

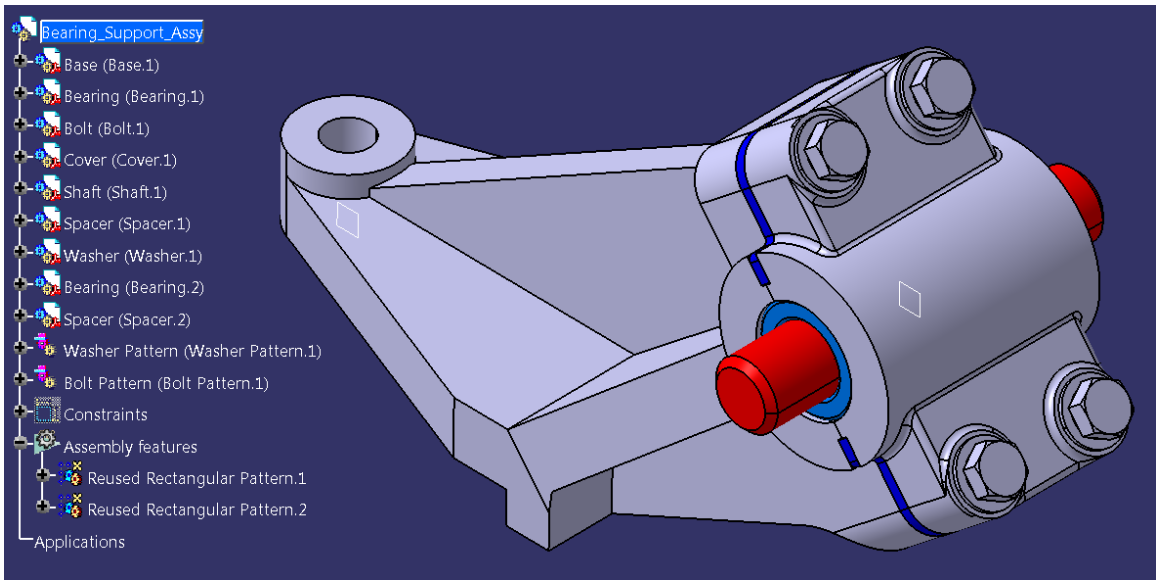
- Click the ***Reuse Pattern*** tool icon  .
- Zoom in and select the washer first. Then, select the inner face of any cover's hole and click the ***Apply*** button; the dialog box will stay open.



- Have a look at the dialog box content. Note also that the three proposed instances and the ***Assembly features*** group that are added at the end of the ***Product Specification Tree***. This approach may be correct for small patterns but with a large number of parts being used in a pattern (parts kept together by a bolt, 2 washers, a nut and a locknut for example…) the tree will rapidly become quite long…
- Click on the ***OK*** button to close the dialog box and click the ***Undo*** tool icon  to delete the repetition.
- Use the ***Reuse Pattern*** tool once again and make the same selections (washer and hole inner face) but this time, click the ***Put new instances in a flexible component*** checkbox, found in the dialog box's lower left corner, before clicking the ***Apply*** button.



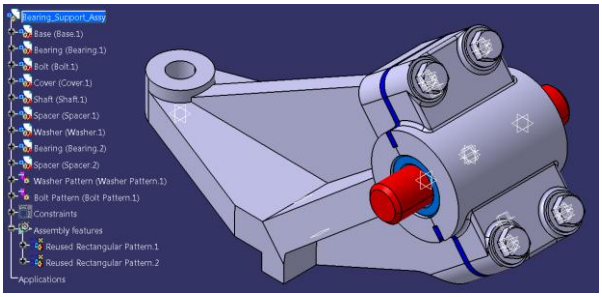
- Note that in this second approach, the three new instances are grouped under a flexible assembly component. The name of this component is usually quite long, taking a lot of space in the working area. It is usually a good practice to edit the name of the component and make it shorter. Using this way, adding three or thirty instances in a pattern will take the same amount of space when the component is collapsed.
- Click the ***OK*** button to close the dialog box and repeat the procedure for the bolt.
- Save the product.



10 – Hiding planes

- To complete, it would be nicer to hide all planes in the assembly. Most methodologies ask to hide all the planes before making a final save on a part, but practice is usually different… Here the correct and permanent way to hide the planes is to access the base and the shaft part and to hide the plane shown in each of them. A quick way to hide all planes temporarily is to hide them through the product. This can be done by using the following ***Power Input***: *t:plane, all.*

By doing this, all planes in all product's parts will be selected. Click the ***Hide/Show*** tool icon  a first time. All planes that were hidden will appear in the product.



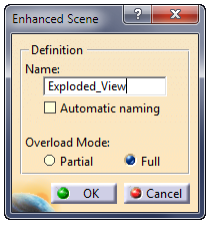
- Repeat the procedure. All planes will then disappear. To save some typing time, you can click in the Power Input area and use the keyboard up arrow to recall the last entry. Remember this trick, you'll probably use it in the first assembly you will build from scratch… and in some others!

11 – Creating a scene

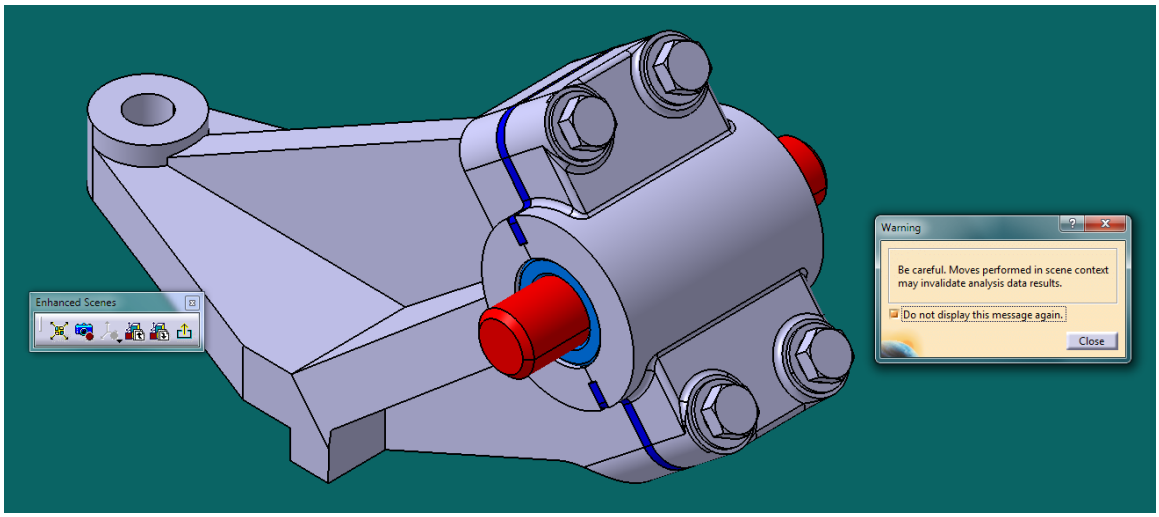
An exploded view will be created on the drawing. In order to make it, it is not possible to explode the assembly in the working environment. A scene will be necessary.

- Click the ***Enhanced Scene*** tool icon  to create a new scene.

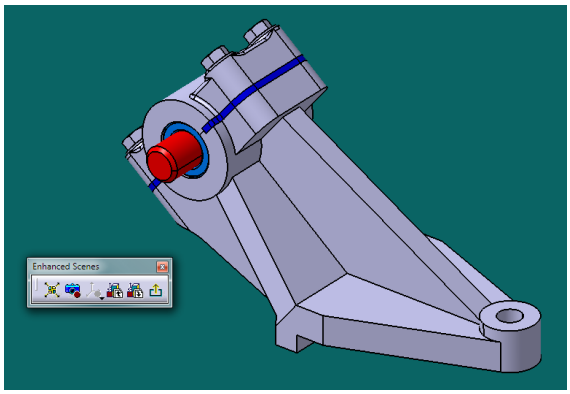
- A dialog box will pop-up. Give a name to the scene, uncheck the ***Automatic naming*** checkbox and set the **Overload Mode** to ***Full***.



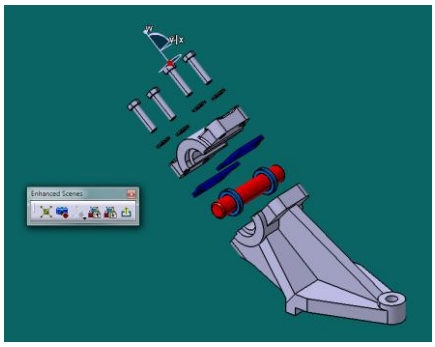
- The product environment will change and a ***green*** background will replace the working area ***blue*** background. A warning message will also inform the user that displacements cannot be reset automatically since assembly constraints no longer exist in a scene. Click the ***Do not display this message again*** checkbox.



- Rotate about the assembly in order to take a view point that will show the back left side of the product. Click the ***Save Viewpoint*** tool icon  to get this viewpoint each time the scene will be accessed.



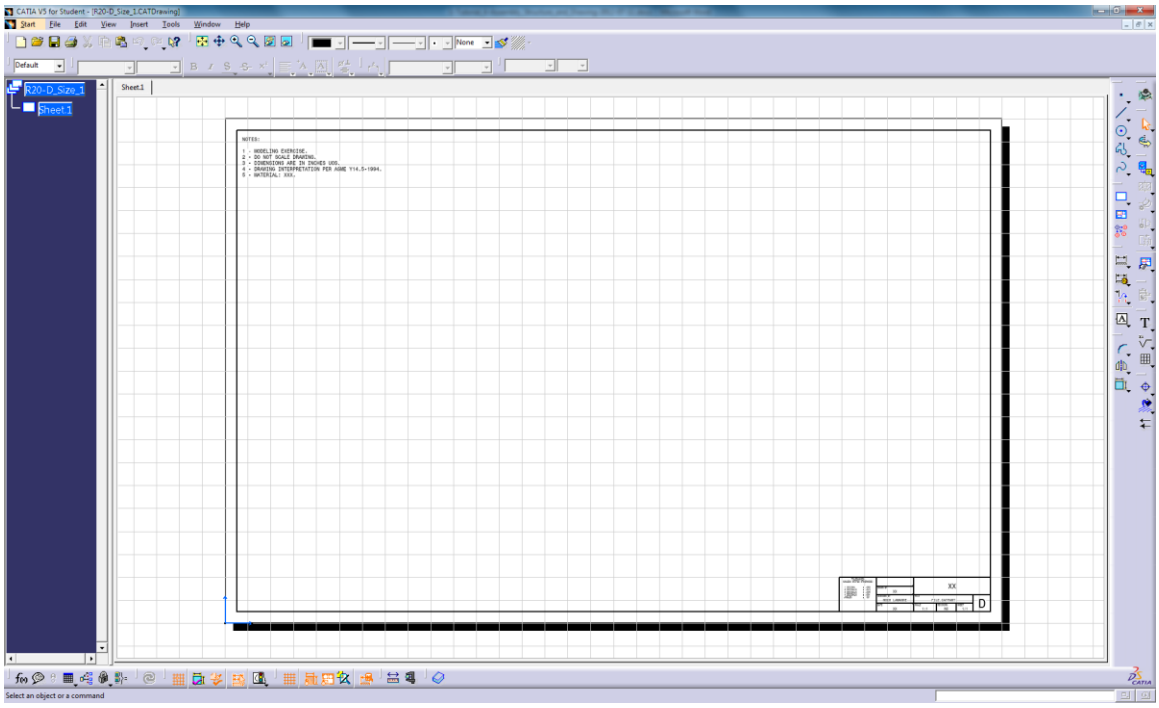
- Move and drop the ***Compass*** to the cover's top face and use the CTRL key to select all assembly items except the base. Move all the parts a certain distance thus avoiding superimposition of parts in the view and then pressing the CTRL key to deselect the shaft and the two bearings. Move the remaining parts again and then stop and deselect the two spacers. Go on until all parts are properly shown.



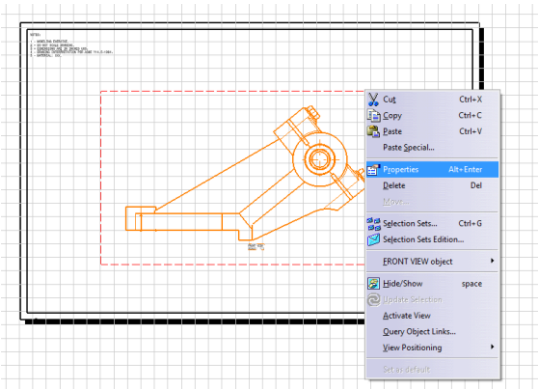
- Exit the scene and save the product.

12 – Creating the assembly drawing

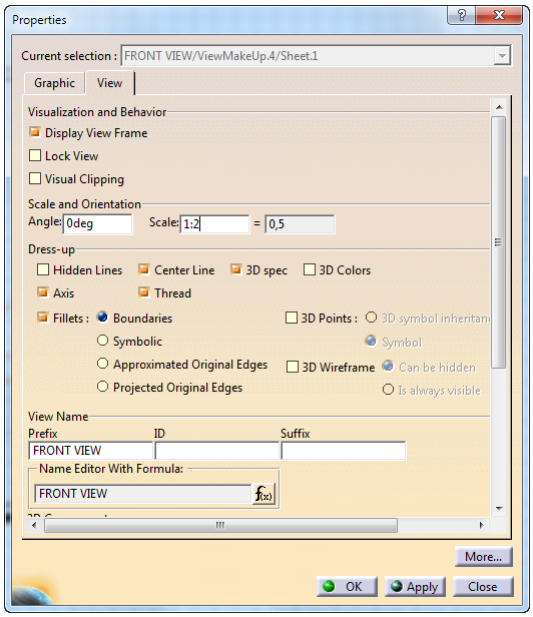
- Use ***File>New From*** and browse to locate the format file named ***R20-D_Size.CATDrawing***. This will create a new drawing file containing a border and a title block.



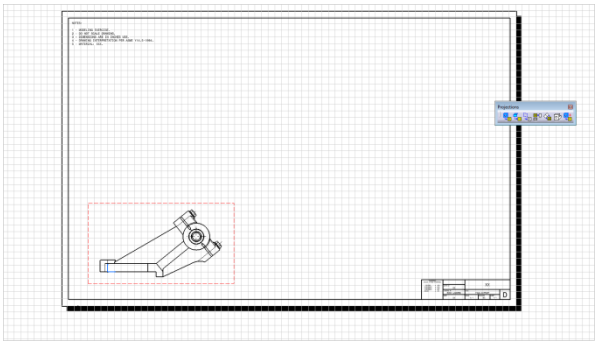
- The same way orthogonal views were extracted in Tutorial 1, click the ***Front View*** tool icon  to extract the first assembly view.
- Bring the mouse cursor over the ***red*** dashed view frame and right-click to access the view's context menu. Select the ***Properties*** option.



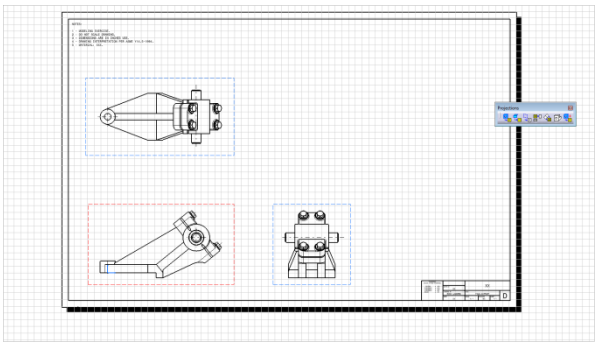
- Set the ***Scale*** to *1:2*.



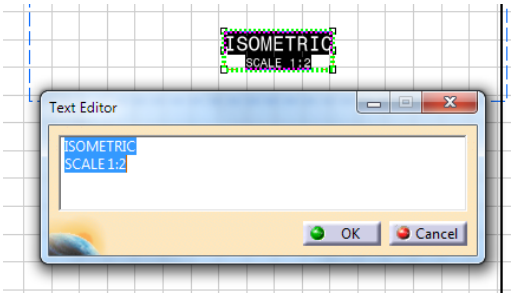
- Click the ***OK*** button, delete the view name from the view and relocate the view in the sheet's bottom left corner.



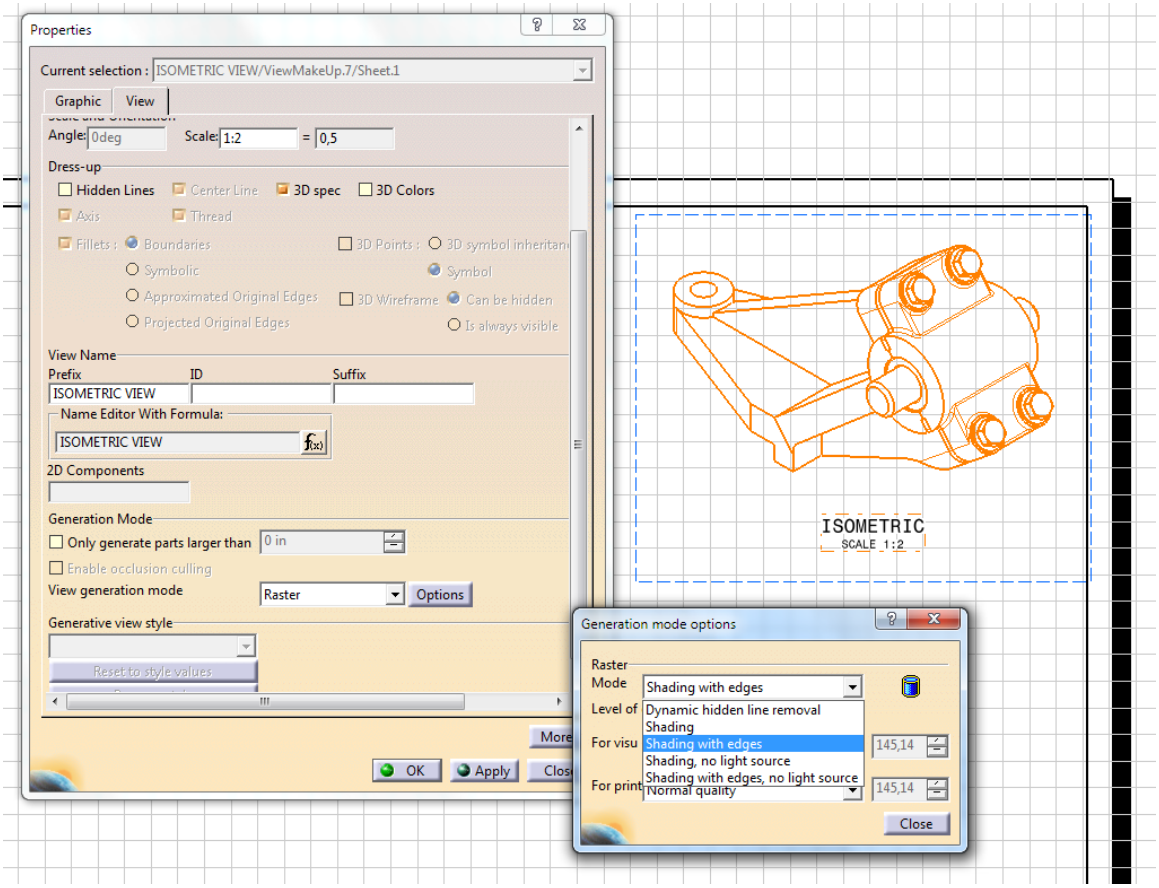
- Click the ***Projection View*** tool icon  to extract a top and a side view. Delete the view names that are automatically created. Note that the two new views are already at a *1:2* scale.



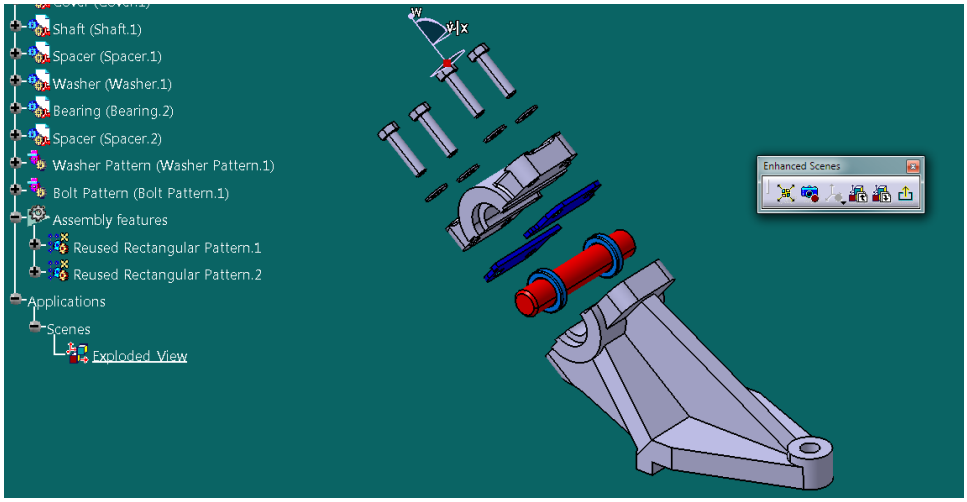
- Click the ***Isometric View*** tool icon  . Switch windows to access the product. Click on the 3-D ***Isometric View*** tool icon  to properly align the assembly and click on any of the parts' faces. A window switch will occur and an isometric view will be created in the drawing. Edit the view name, making the top line *0.25* inch high and stating: ISOMETRIC, in capital letters. Make the second line *0.15* inch high and stating: SCALE 1:2. Make the two texts centered.



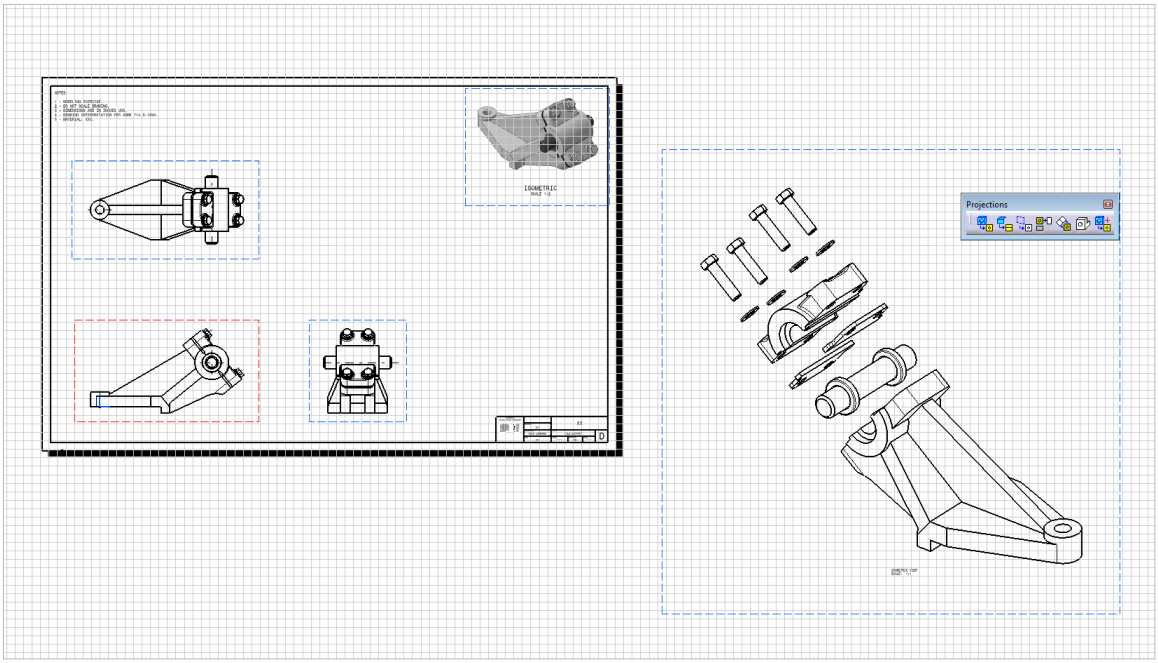
- Edit the view properties' dialog box.
- Change the view scale to *1:2* and click on the ***Apply*** button.
- Scroll down to be able to change the ***View generation mode*** to ***Raster*** and click on the ***Options*** button to set the ***Raster Mode*** to ***Shading with edges***. Click on the ***Close*** and the ***OK*** buttons.



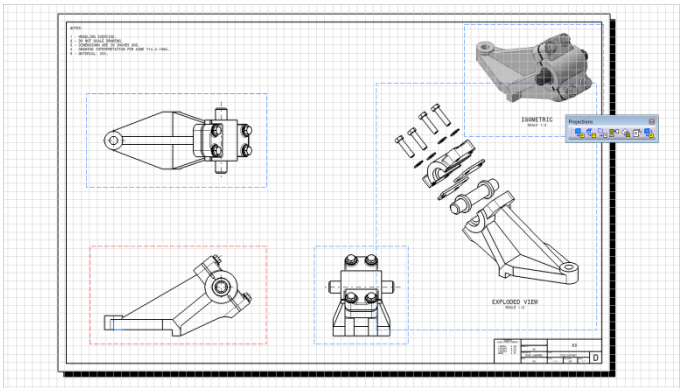
- Locate the isometric view in the drawing's top right corner.
- Switch windows to get back to the assembly.
- Double-click on the *Exploded_View* scene item in ***Product Specification Tree***. This will allow the user to access the scene.



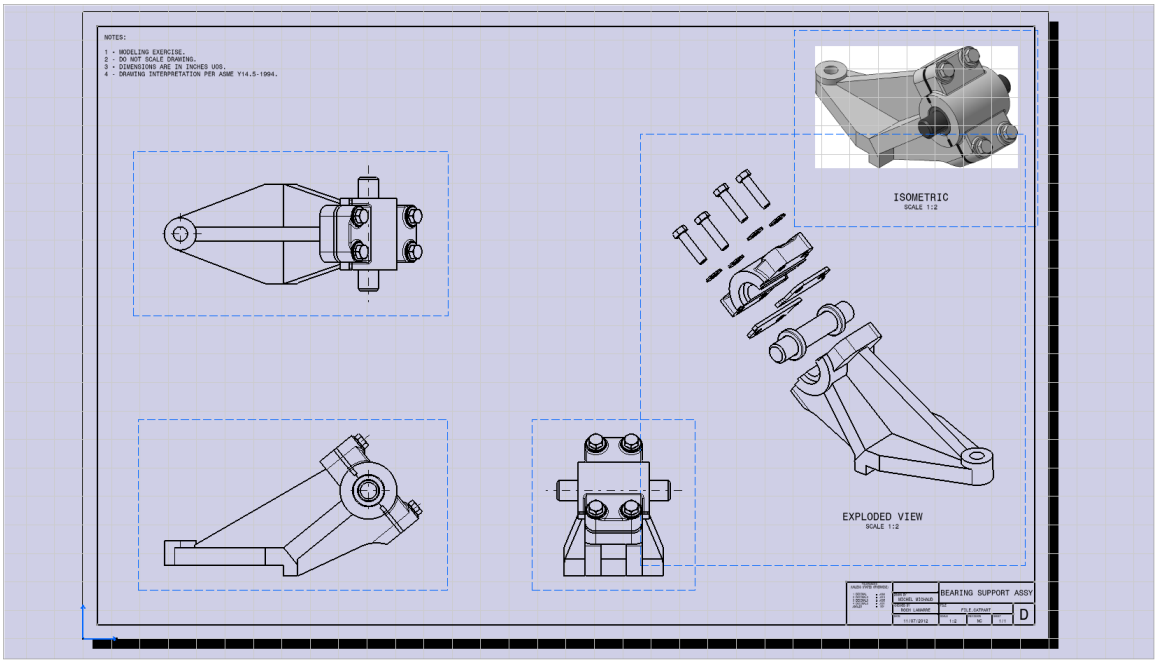
- Switch windows to access the drawing.
- Click the ***Isometric View*** tool icon to extract the last view.
- Switch windows to access the scene.
- Click the scene name in the ***Product Specification Tree***.
- Click any solid's face.
- Complete the view extraction.



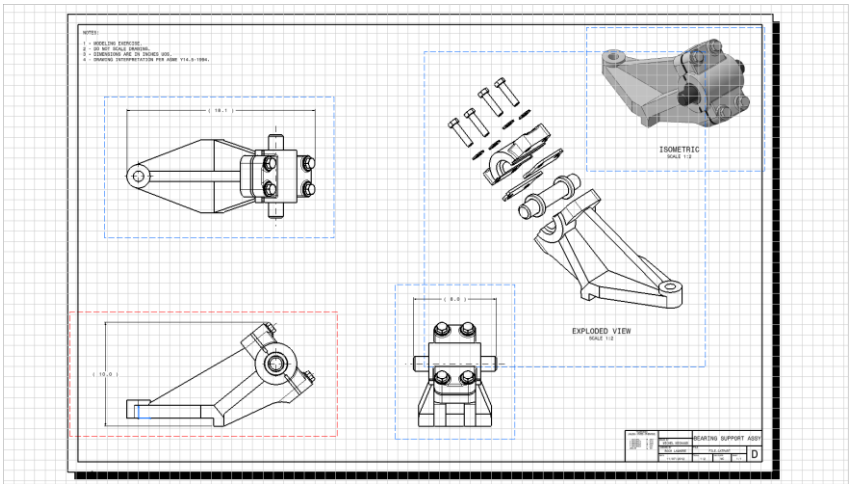
- Edit the view name to make it EXPLODED VIEW and SCALE 1:2, using the same standards as before.
- Access the view properties' dialog box to change the scale to *1:2.*
- Relocate the view in the drawing.



- Use ***Edit>Sheet Background*** to complete the drawing information. Delete the *Material* line in the notes and fill out the title block. When done use ***Edit>Working Views*** to get back to the standard environment.

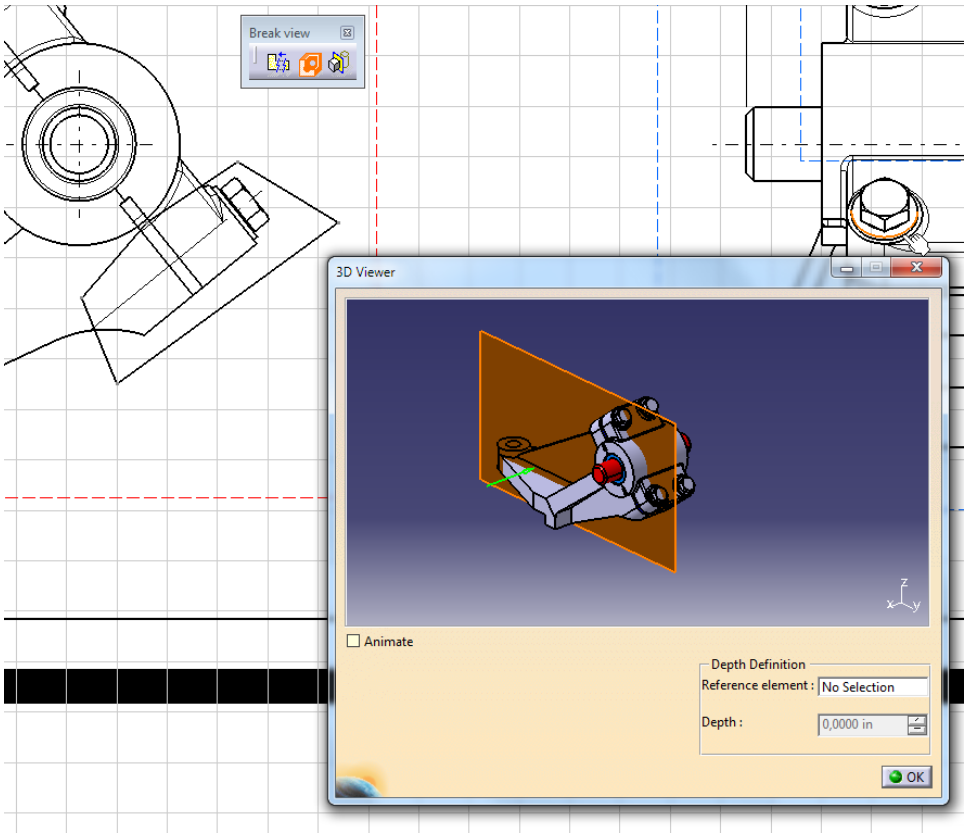


- Save the drawing.
- Double-click on the ***Dimensions*** tool icon  to create the three overall dimensions. Make them *0.150* inch high, use only one decimal place and add parenthesis as prefix and suffix to show they are reference dimensions.

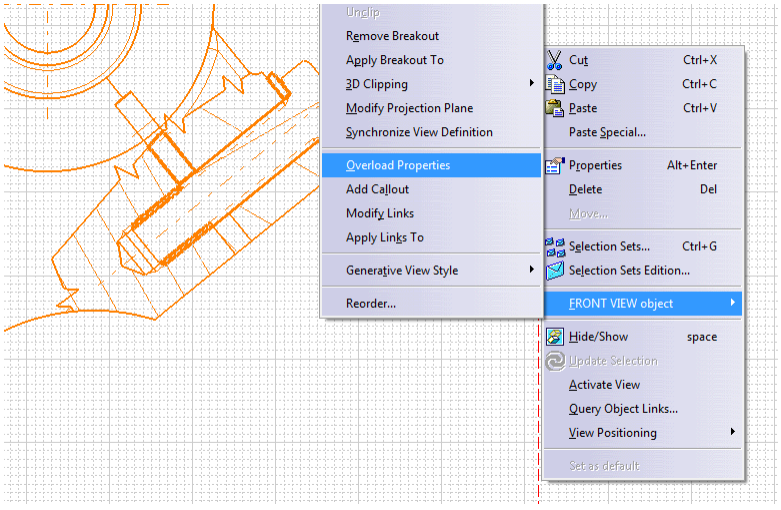


- Expand the ***Break View*** toolbar 

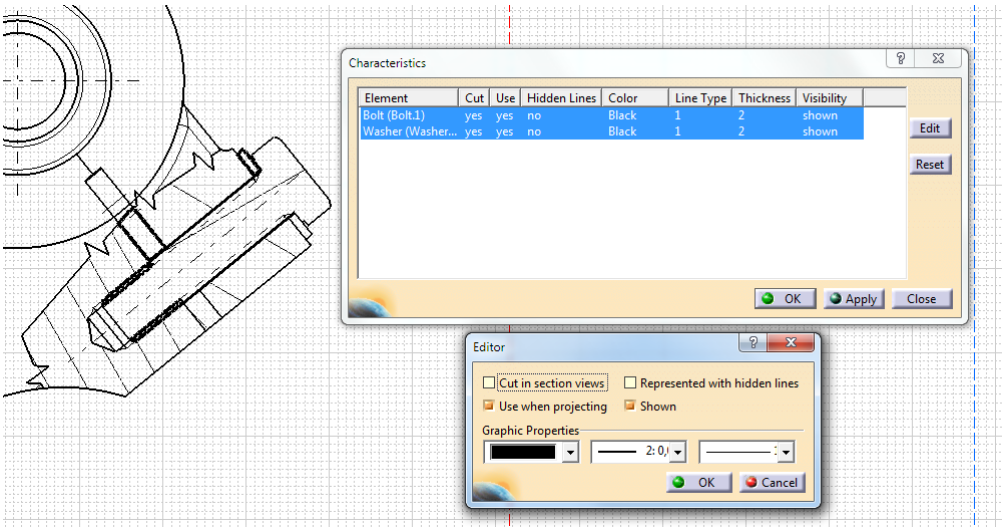
hidden under the ***Broken View*** tool icon  .
- Make sure the front view is active.
- Click the ***Breakout View*** tool icon  and define a breakout region, in the front view, in a way that the bottom bolt will become fully visible. Note that it is possible to rotate about the content in the ***3D Viewer*** the same way it is done in the working area.
- Bring the mouse cursor near the representation of the bottom left washer in the right view and click the ellipse. Selecting a circle or an ellipse in a view projected from the one in which the breakout will be generated is probably the fastest way to align the cutting plane with the axis of a hole.
- Click the ***OK*** button in the ***3D Viewer*** dialog box to complete the breakout definition.



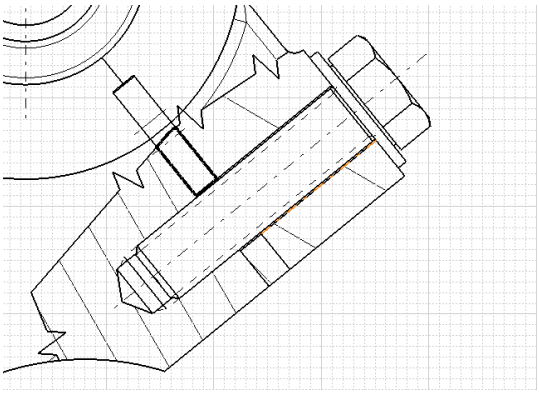
- Since the breakout cut the bolt and the washer and that drafting standards ask to do not cut standard parts, access the view context menu and activate the ***Overload Properties*** option.

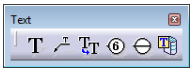


- Select one line on the bolt and one line on the washer. Both instances should appear in the ***Characteristics*** dialog box.
- Using the CTRL key select both instances in the dialog box and click on the ***Edit*** button. The ***Editor*** dialog box will pop-up.



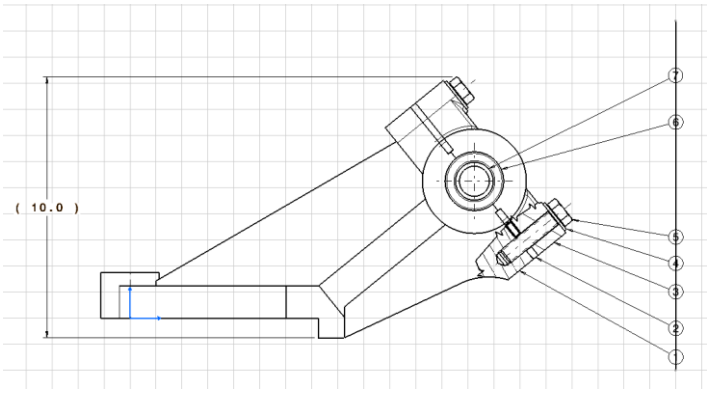
- Uncheck the ***Cut in section views*** checkbox and click the ***OK*** buttons in each of the dialog boxes. The bolt and the washer should no longer be sectioned.



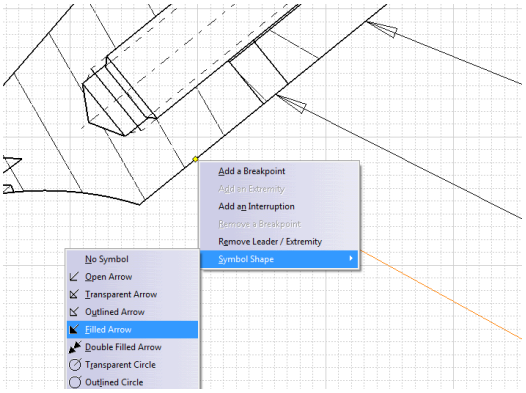
- Create a vertical line to the right of the assembly representation in the front view.
- Expand the ***Text*** toolbar  hidden under the ***Text*** tool icon  .
- Double-Click the ***Balloon*** tool icon and identify each of the assembly components by using a counterclockwise incremental approach. Start at the bottom of the line and center each balloon to the line going up. When aligning balloons vertically or

horizontally, the drawing grid may be used instead as a reference, but it happens that balloons are aligned along one of the assembly's side neither vertical nor horizontal and then, the construction line technique is more practical.

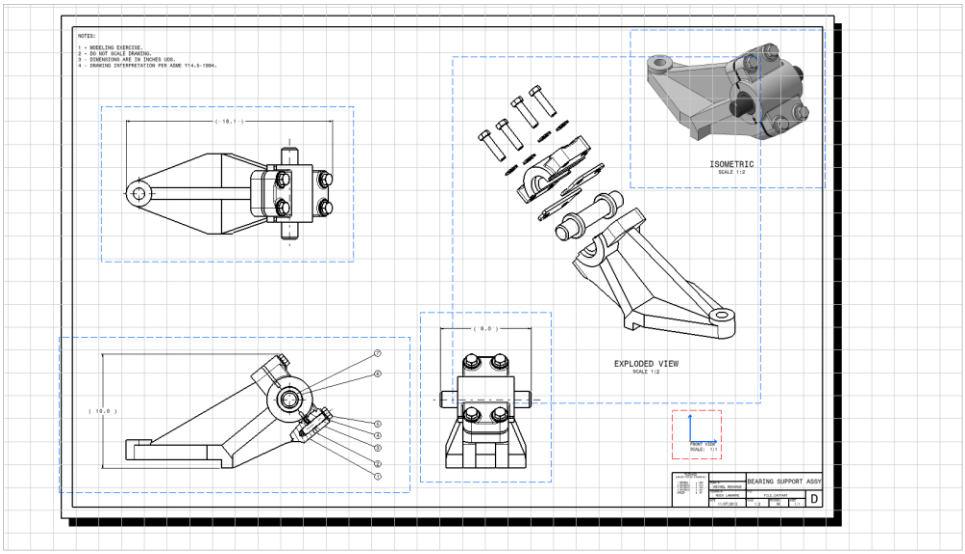
- Hide the line in no-show. It may be useful in the future.
- Make sure all your balloon text is *0.150* inch high.



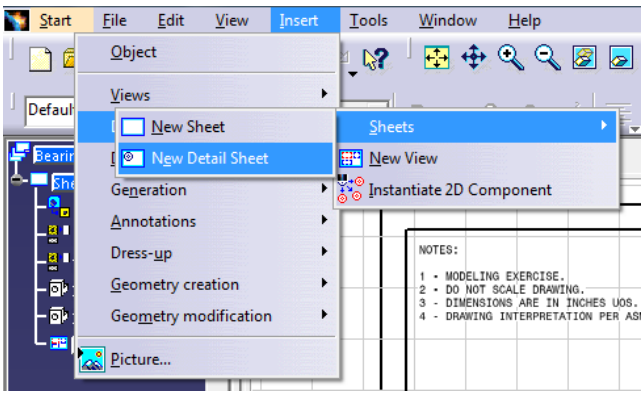
- Zoom in to have a look at your arrow heads: they should be filled.
- If they are not, click on the first leader line and locate the ***yellow*** diamond shape that appears at the tip of the arrow. Bring the mouse cursor over the diamond shape and right click to access the context menu. Select the ***Symbol Shape*** option and choose *Filled Arrow*. Repeat the procedure for all balloons.



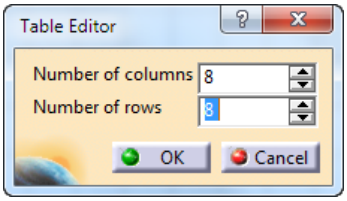
- Click the ***New View*** tool icon  and click about one inch over the title block to locate the view.



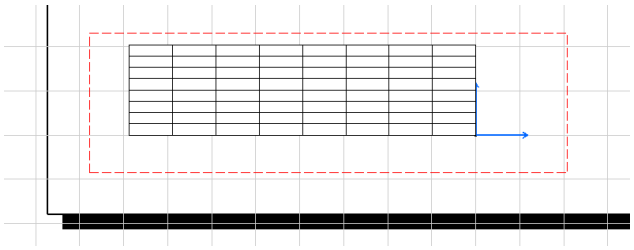
- Use ***Insert>Drawing>Sheets>New Detail Sheet*** to add a detail sheet to the drawing. A view is already present and active in this new sheet.



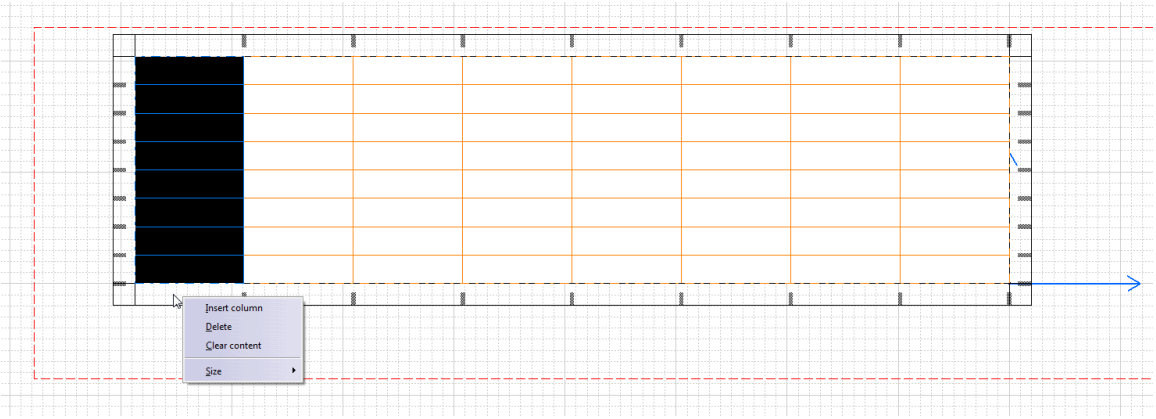
- Click the ***Table*** tool icon  and edit the content of the ***Table Editor*** to have *8* columns and *8* lines.



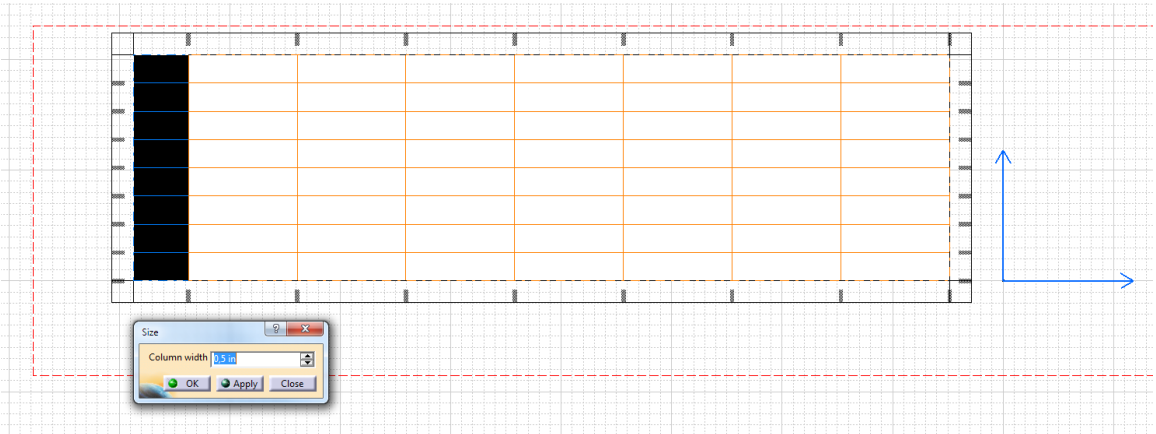
- Click in the view to locate the table. Make sure the table's lower right corner is perfectly aligned to the view origin. This will facilitate its insertion later on.



- Double-click the table. A new frame will appear.
- Bring the mouse cursor under the first column and right click to access the context menu.

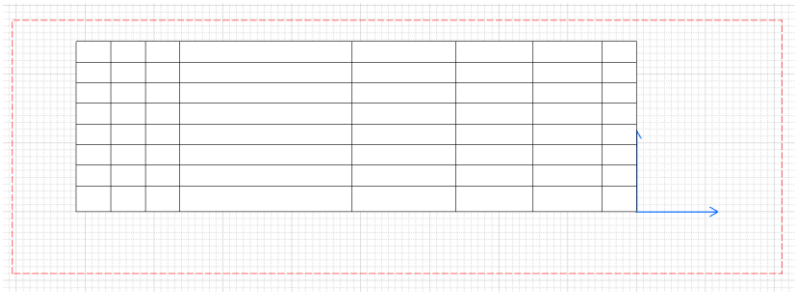


- Select the ***Size*** option and then the ***Set Size*** sub-option.
- Set the first column width to *0.5* inch and click the ***OK*** button to apply the change.

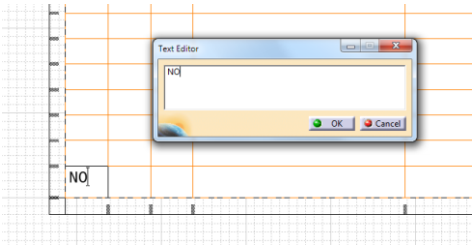


- Set the other column widths to *0.5*, *0.5*, *2.5*, *2.5*, *1.125*, *1* and *0.5* inches.
- Bring the mouse cursor to the left of the bottom line.

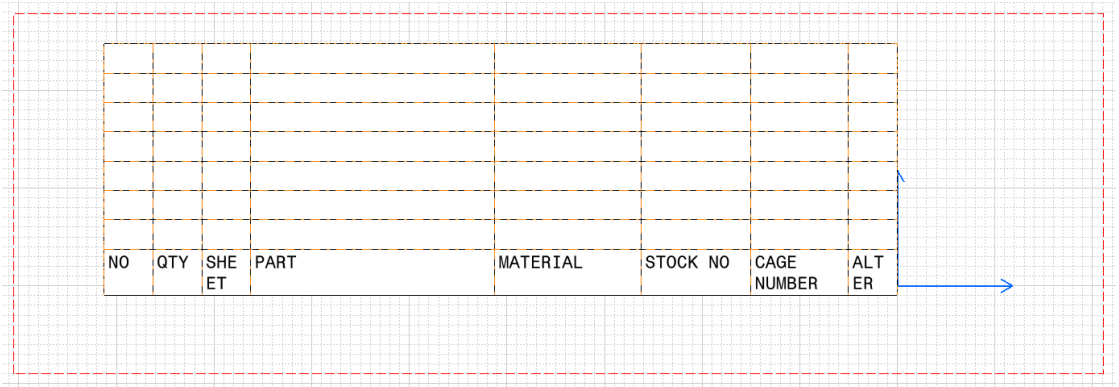
- Using a similar approach, set the bottom line height to *0.375* inch.
- Make all the remaining lines *0.3* inch high.
- Click outside of the table to complete the edition.
- Relocate the table in the view if necessary.



- Double-click the table to activate it.
- Double-click in the bottom left cell. Add *NO* in the ***Text Editor***. Click the ***OK*** button to complete the text creation.

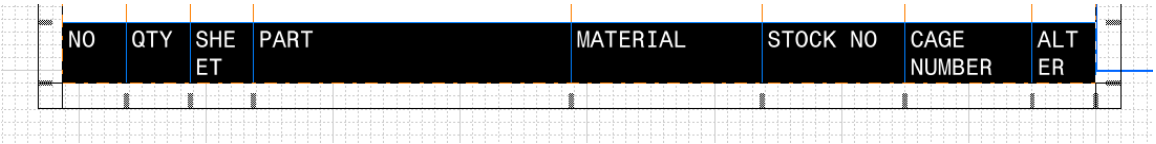


- Add the following texts in the remaining columns: *QTY*, *SHEET*, *PART*, *MATERIAL*, *STOCK NO*, *CAGE NUMBER*, *ALTER.*

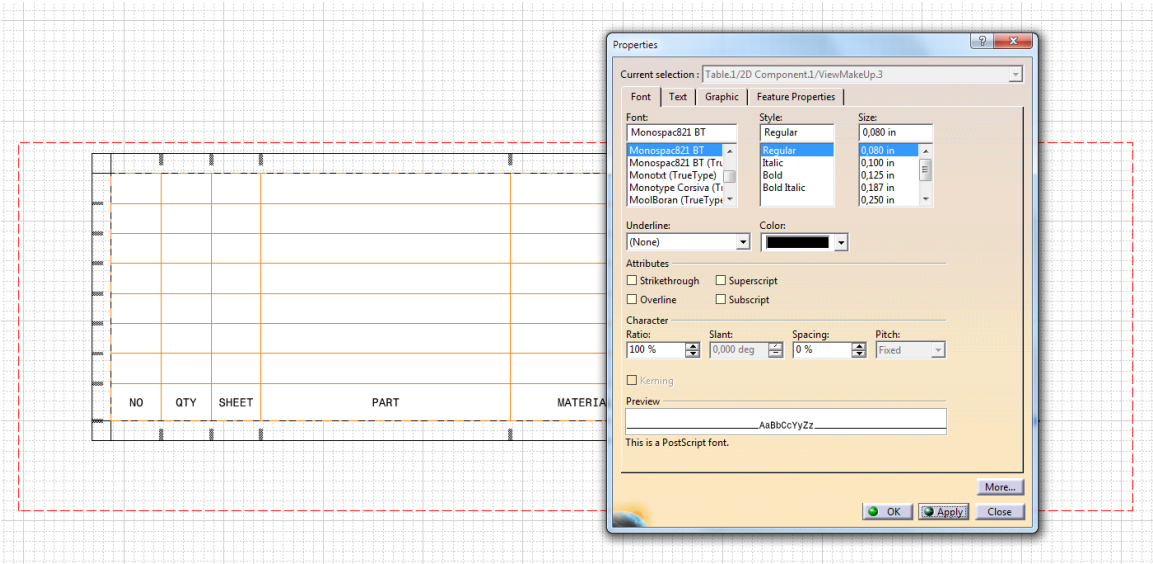


- Keep the table activated.

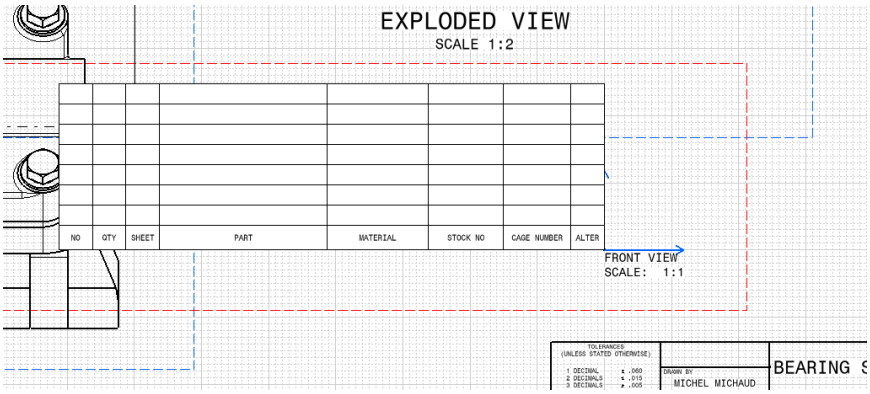
- Click and drag all the bottom line cells. They will become ***black***.



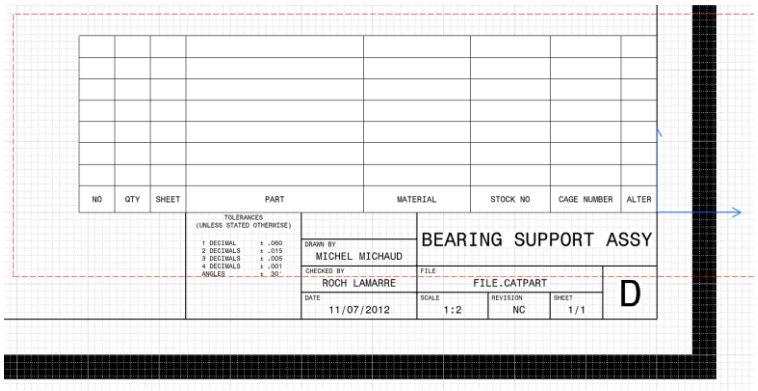
- Bring the mouse cursor over the ***black*** background and right-click to access the context menu. Select the ***Properties*** option.
- In the ***Font*** tab, set the ***Size*** to *0.080* inch. In the ***Text*** tab, set the ***Position Anchor Point*** to Middle Center. Click the ***OK*** button to complete the edition.



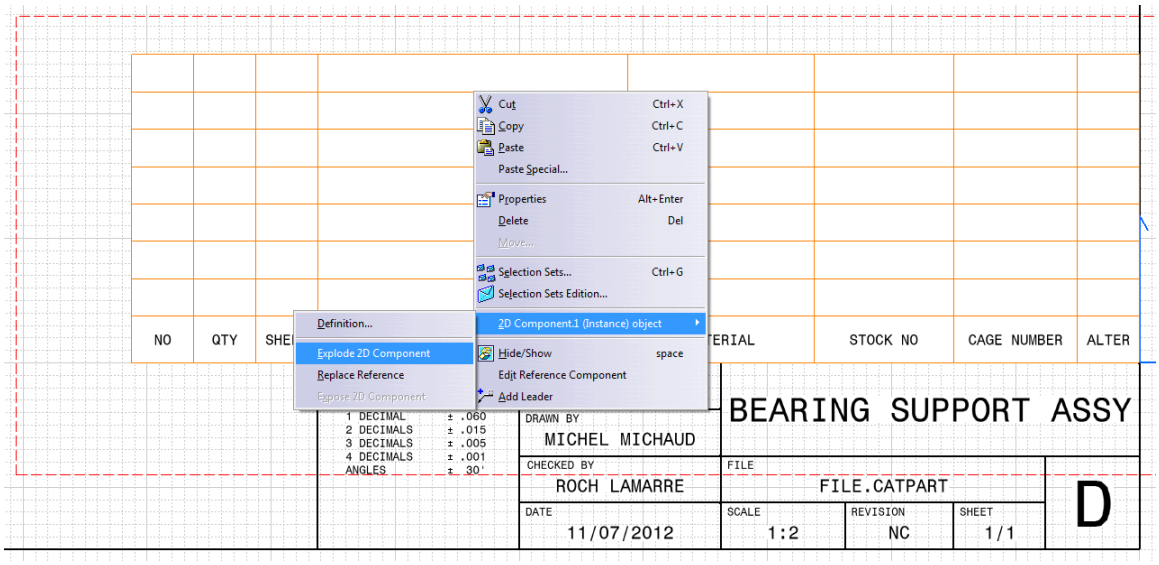
- A basic Parts List is now available as a detail.
- Save the drawing.
- Get back to the first sheet by selecting its tab in the screen's top left corner.
- Click the ***Instantiate 2D Component*** tool icon to insert the Parts List in the view previously created over the title block.
- Access the detail sheet by selecting its tab and select the table. It will appear in the first sheet's view. Locate the table's bottom right corner at the view's origin.



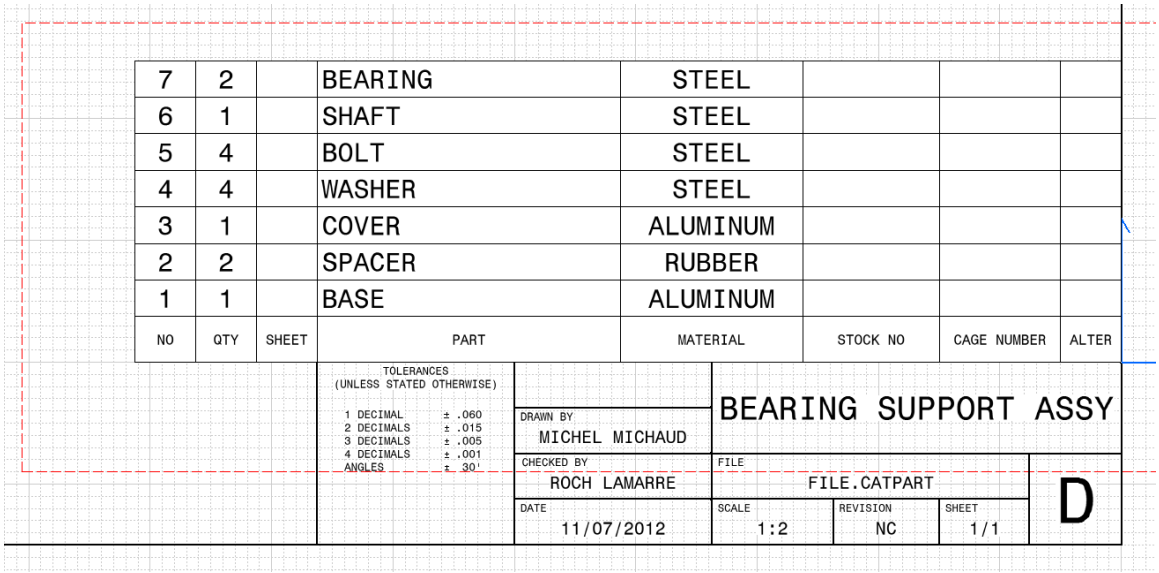
- Delete the view name and relocate the view in order to align the table's left side to the drawing border. Do not hesitate to zoom in to get a perfect fit.



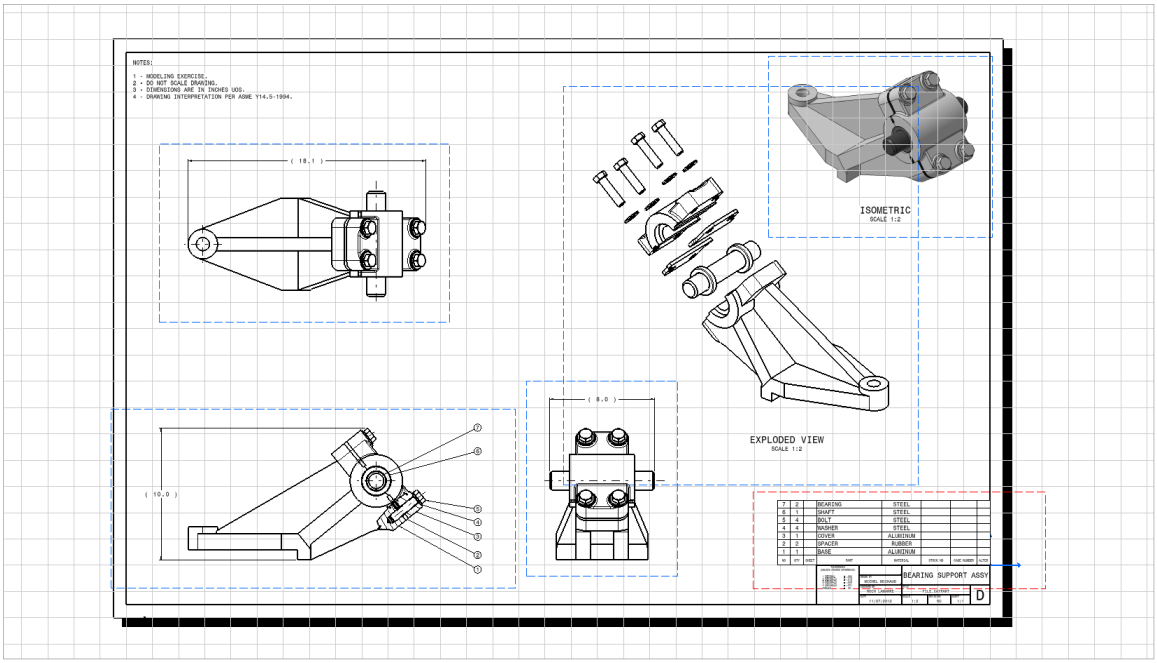
- The inserted table is still linked to its definition stored in the detail sheet. Should a new copy be necessary in a different sheet, for a sub-assembly for example, it would be impossible to edit one table without editing the other. In this case, it is preferable to break the link between the instance and the reference detail.
- Bring the mouse cursor anywhere over the table and right-click to access the context menu. Select the ***Explode 2D Component*** option found under ***2D Component.1 (Instance) object***. The table is now stand alone.



- Edit the table to complete the Parts List.

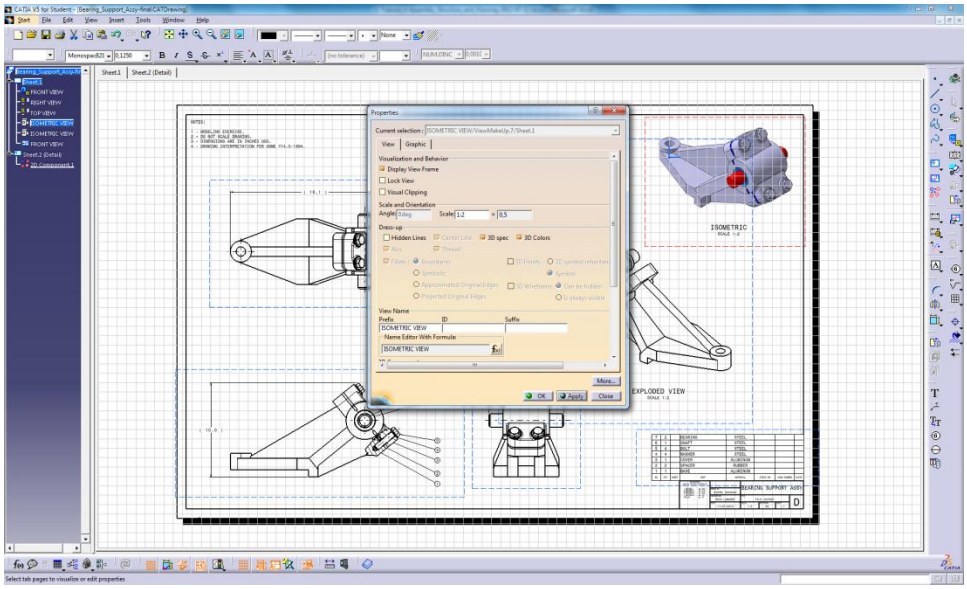


- The drawing is about to be completed.

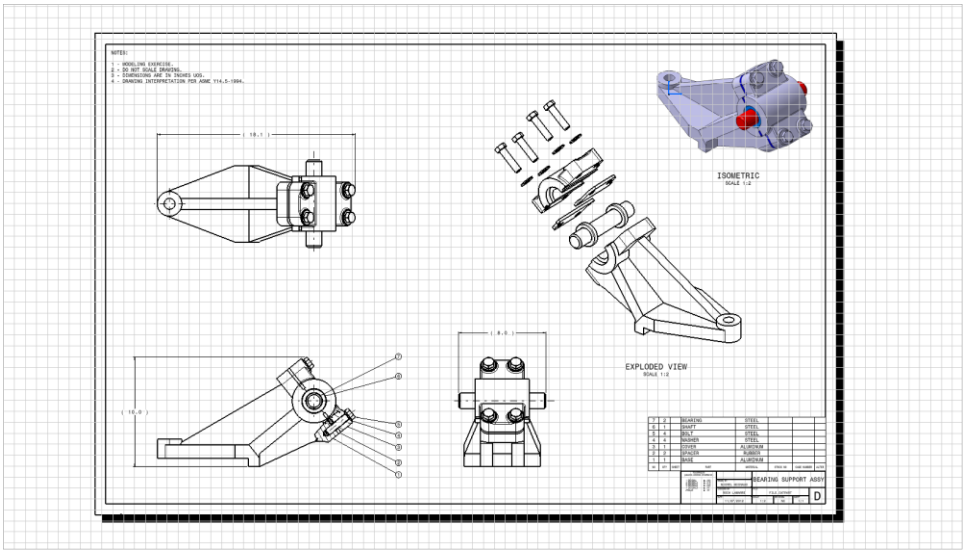


If desired, it is possible to add color to the drawing. This may be nice for a portfolio but is usually not an industry standard…

- Double-click the shaded isometric view border to make it active and right-click it to access the context menu. Select the ***Properties*** option.
- In the View tab, check the ***3D Colors*** checkbox and click the ***Apply*** button. Click the ***OK*** button to complete the operation.



- Click on the ***Display View Frame as Specified for Each View*** tool icon  to make all view frames disappear from the drawing.



- Save the drawing.

Thanks to Alice Michaud for revising this text.