Tutorial 1

Part Creation – Basic Modeling – Basic Drafting - Printing

Purpose:

To give the student the basic knowledge to create and model a solid part, to extract its views and to get a print.

Reference:

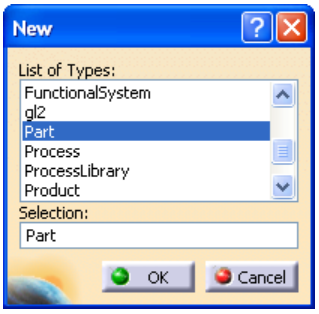
Use the ***01-Index_Plate.pdf*** and the ***R20-D_Size.CATDrawing*** *files.*

1 – Launch CATIA®

- If a product file is automatically created, close it.

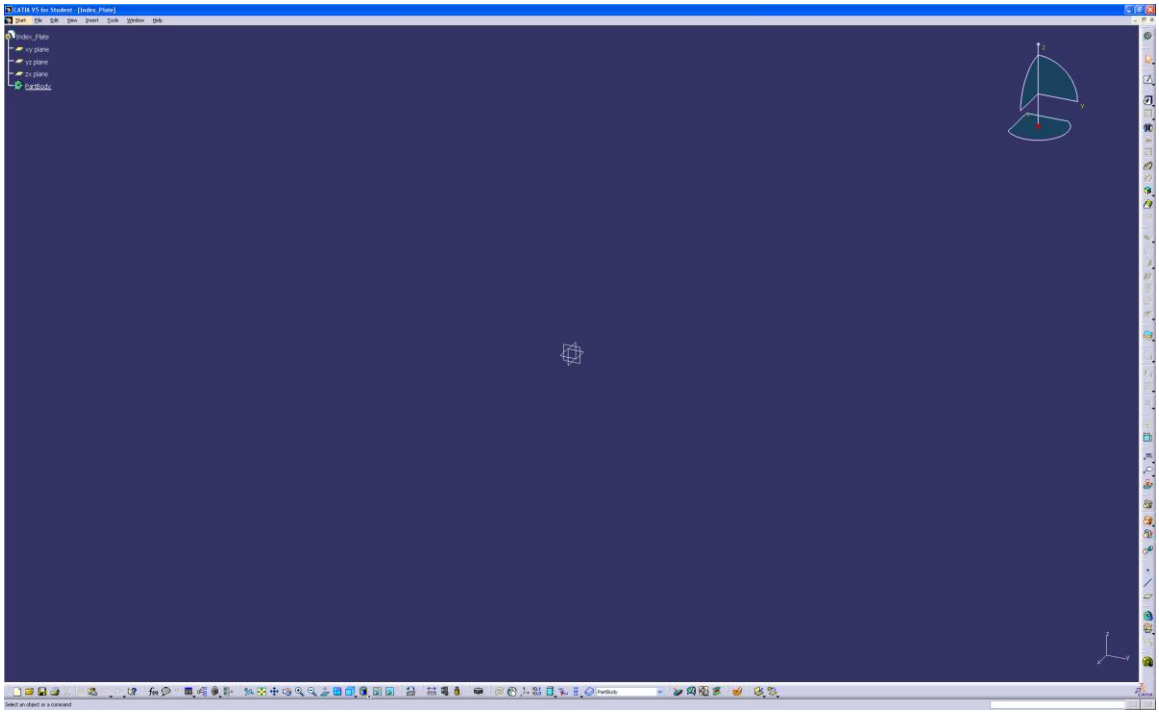
2 – Create a new part

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



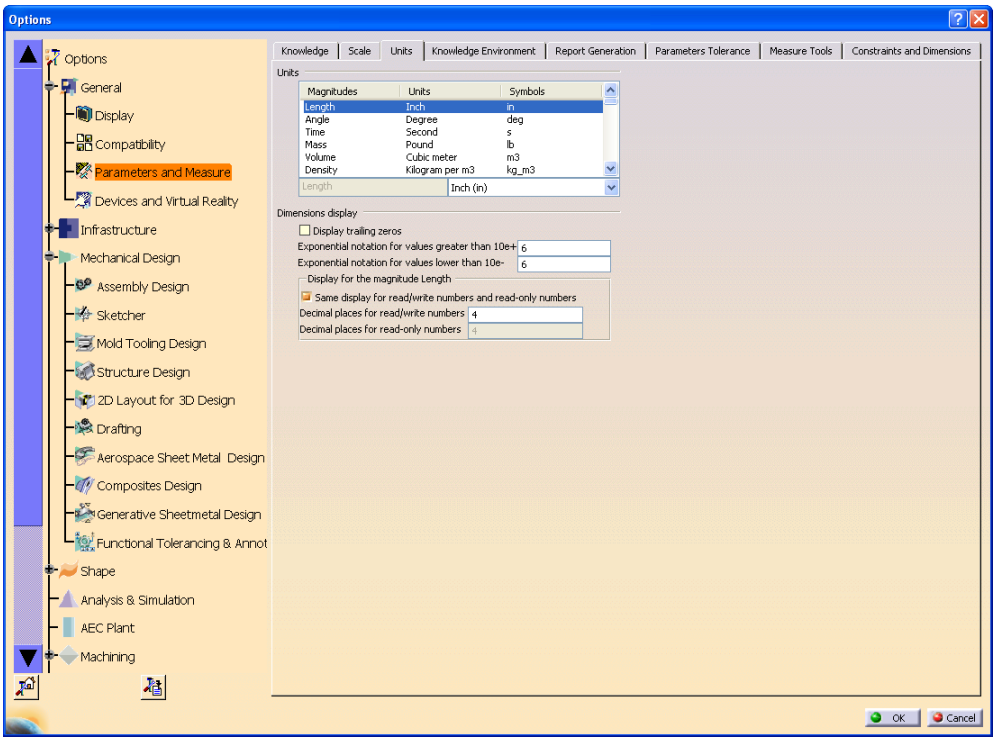
- In the ***New Part*** dialog box, replace *Part1* by *Index_Plate.*

- Make sure the ***Enable hybrid design*** check box is not selected.
- Click ***OK*** to close the dialog box.
- A new part file is open in CATIA®.

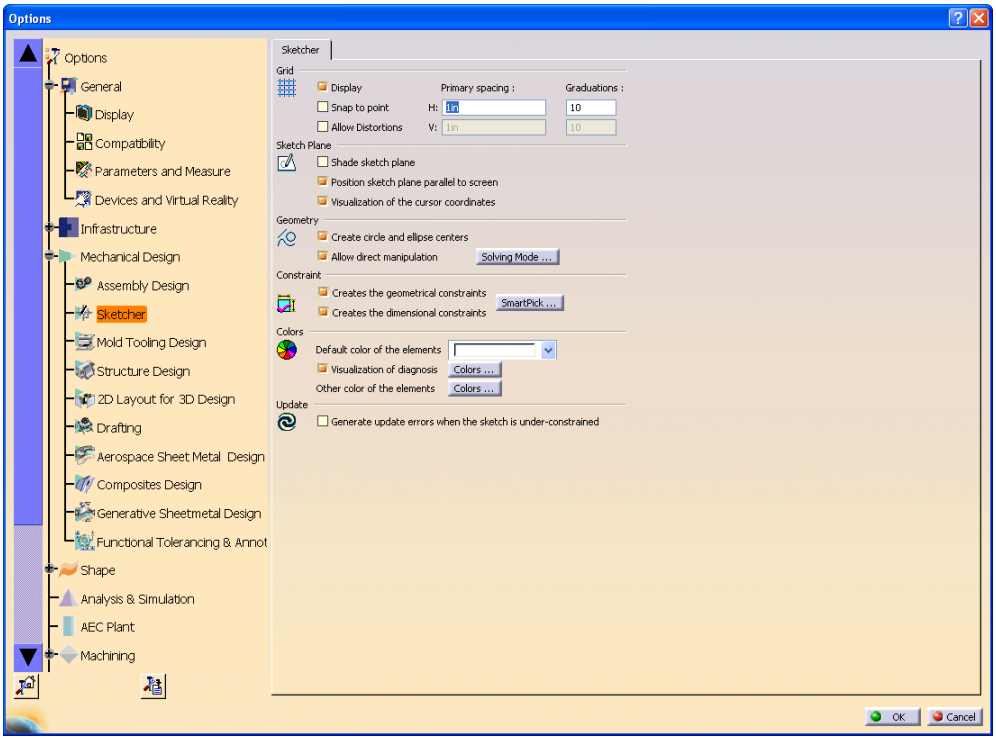


3 – Organize the environment

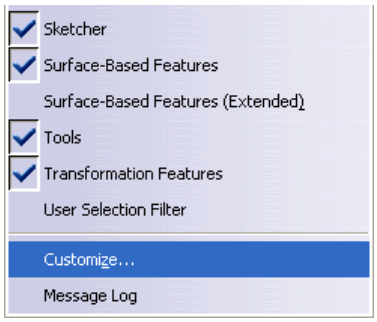
- Click on the ***XYZ*** tripod in the lower right corner to activate the ***Part Specification tree***. Note that the three planes shown in the middle of the screen become darker.
- Zoom in or out in order to get the tree to a correct size on the screen. Use the middle+left or middle+right button sequence to achieve it. Note that the name appearing at the top of tree is now *Index_Plate*.
- Click on the ***XYZ*** tripod in the lower right corner to de-activate the ***Part Specification tree***. Note that the three planes shown in the middle of the screen become lighter.
- Use ***Tools>Options*** to display the ***Options*** dialog box.
- In the ***General*** group, select ***Parameters and Measure***. Select the ***Units*** tab and make sure basic magnitudes (Length, area, volume etc.) are defined using *Imperial* units and the value is set to *4* for ***Decimal places for read/write numbers***.



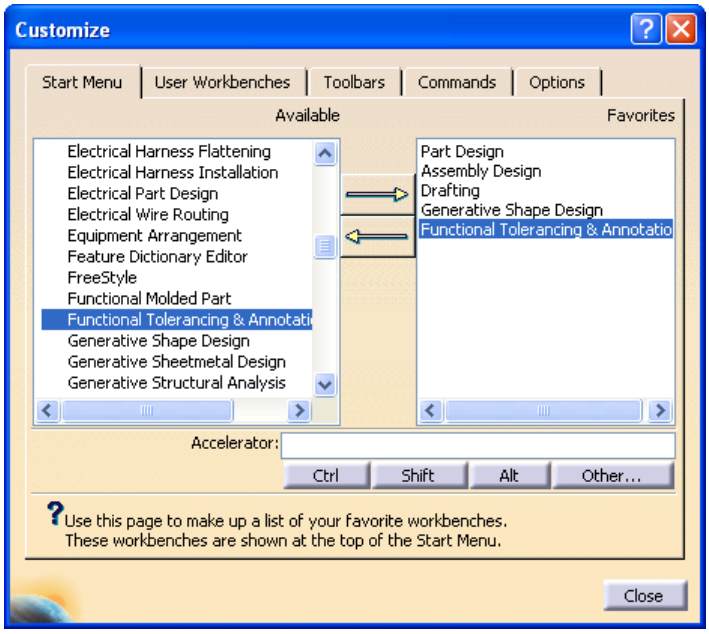
- In the ***Mechanical Design*** group, select ***Sketcher***. Make sure the ***Primary spacing*** value is set to *1* inch and that there are *8* or *10* ***Graduations***.



- Make the same dimensional settings in the ***General*** tab found under ***Drafting***.
- Click ***OK*** to close the dialog box.
- Bring the mouse cursor over any icon and right-click to access the context menu.
- Select the ***Customize*** option.



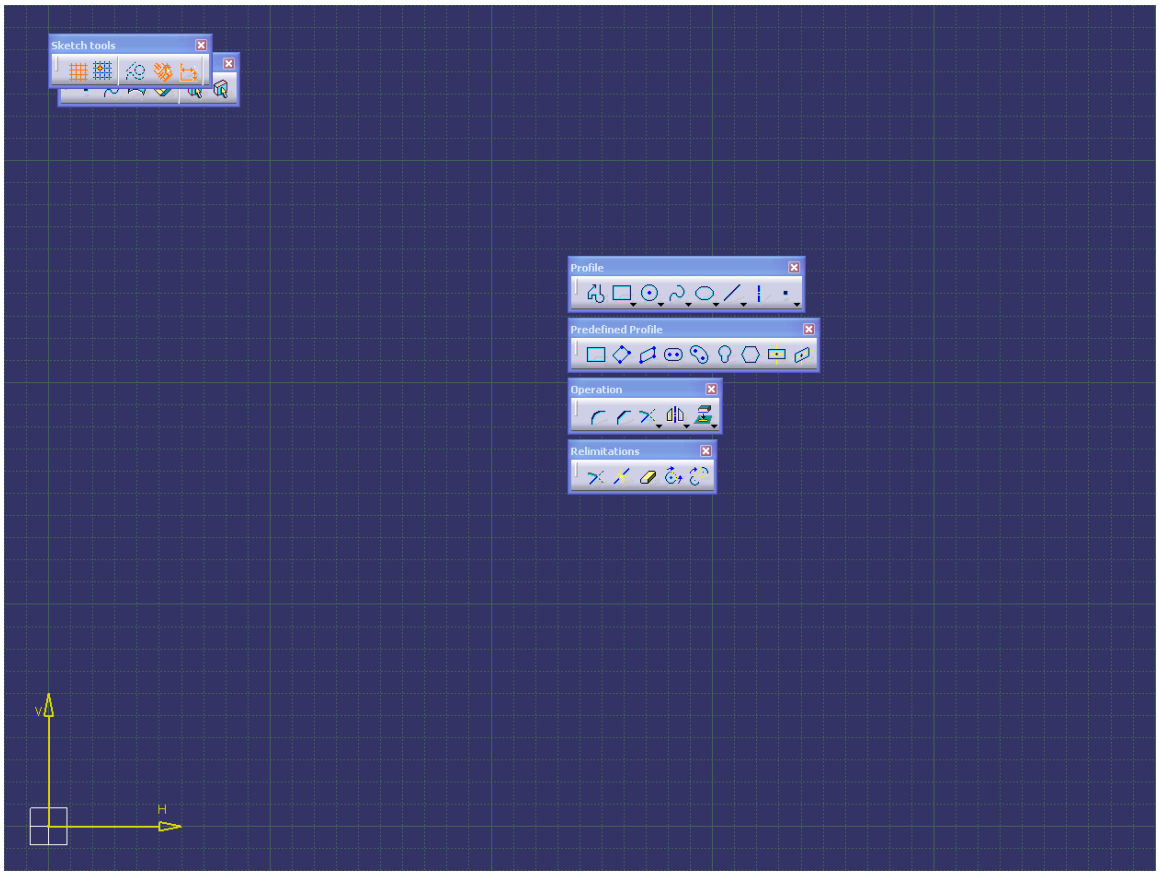
- In the ***Start Menu*** tab of the ***Customize*** dialog box, make sure to make the following workbenches favorite: ***Part Design***, ***Assembly Design***, ***Drafting***, ***Generative Shape Design*** and ***Functional Tolerancing and Annotation***. This will allows a quicker workbench change.



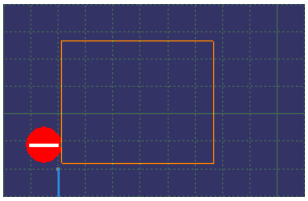
4 – Create a sketch

- Click the ***Sketcher*** tool icon  to create a new sketch on the ***XY*** plane.
- The ***Sketcher's*** environment becomes available.
- Use the mouse (or a similar tool) to pan and zoom in order to locate the origin in the bottom left corner and to have a grid about *8* inches wide by *5* inches high displayed on the screen.

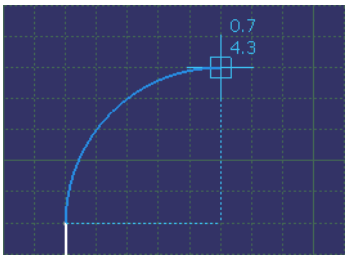
- Bring the ***Profile*** and ***Operation*** toolbars in the working environment. Access the ***Predefined Profile*** and ***Relimitations*** toolbars and make them readily available.



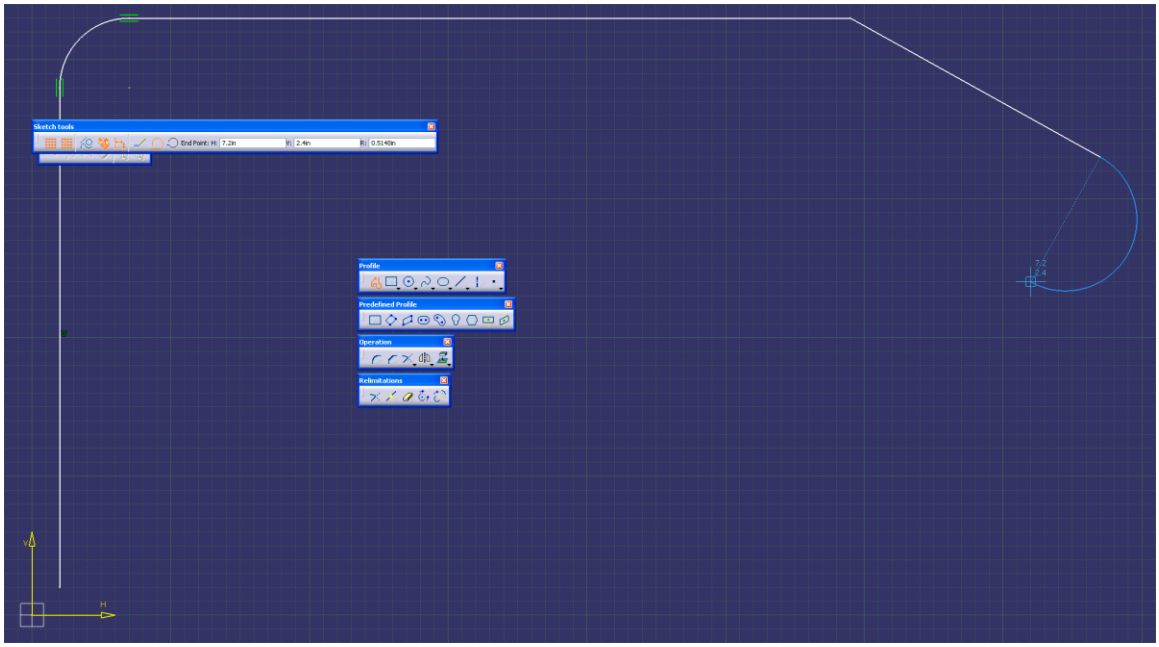
- Click the ***Profile*** tool icon  to create the part outline.
- Start by clicking the first point close, but not on the ***V*** and ***H*** axis origin.
- Create the first vertical segment by clicking a second point aligned with the first one. The line segment will turn ***blue*** to indicate that a vertical constraint will be created.
- Keep the left mouse button depressed.
- Move the mouse to the right in order to obtain a temporary ***red*** rectangle on the screen.



- Release the left mouse button and size the tangent arc in order to stop it ready to create a horizontal segment.

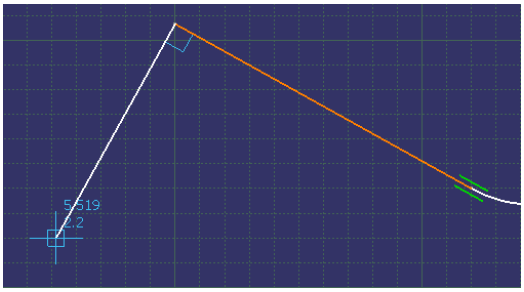


- Click to create the arc when it turns ***blue***.
- Create a horizontal segment by moving the mouse horizontally and click a point when a ***blue*** segment, approximately *5* inches long, is about to be created.
- Create a segment about *30°* from the horizontal, about *1.5* inches long. When clicking the second point, keep the left mouse button depressed to create a new tangent arc.
- Locate the end point of the tangent arc in order to be ready to create a new segment parallel to the previous one. The arc turns ***blue***.

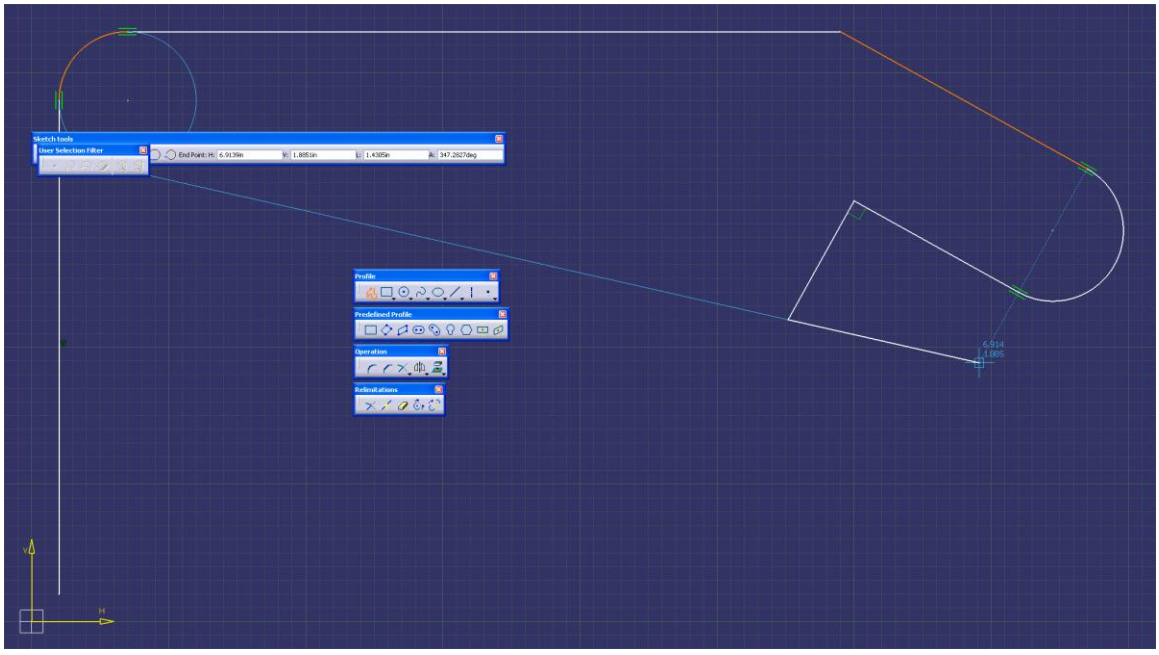


- Complete the arc creation and create a linear segment parallel to the inclined one.

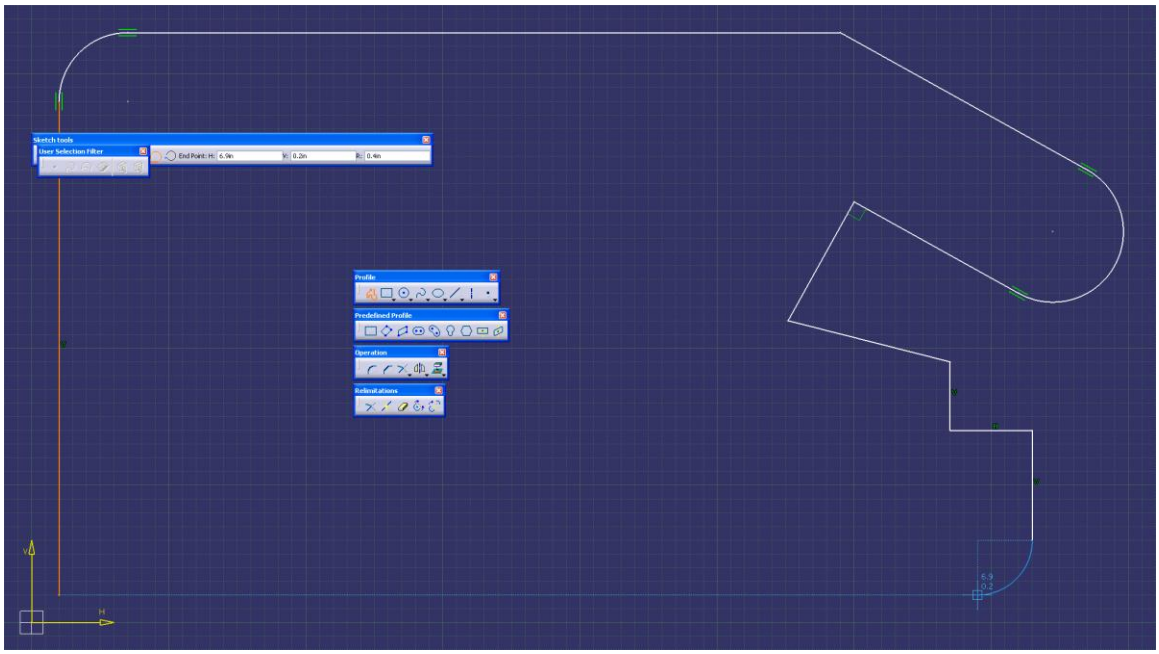
- Click a new point to create a segment orthogonal to the one just created. When locating the segment endpoint, the reference inclined segment turns orange and a ***blue*** orthogonal symbol appears at the junction of the two segments.



- Create the linear segments necessary to reach the contour's lower right corner. When clicking points to create the linear segments, be careful to connect to non-relevant entities since the ***Smart Picking*** mode is active. If a point must be selected but constantly falls into a ***Smart Picking*** selection, click on the SHIFT key simultaneously to momentarily deactivate the ***Smart Picking*** mode.

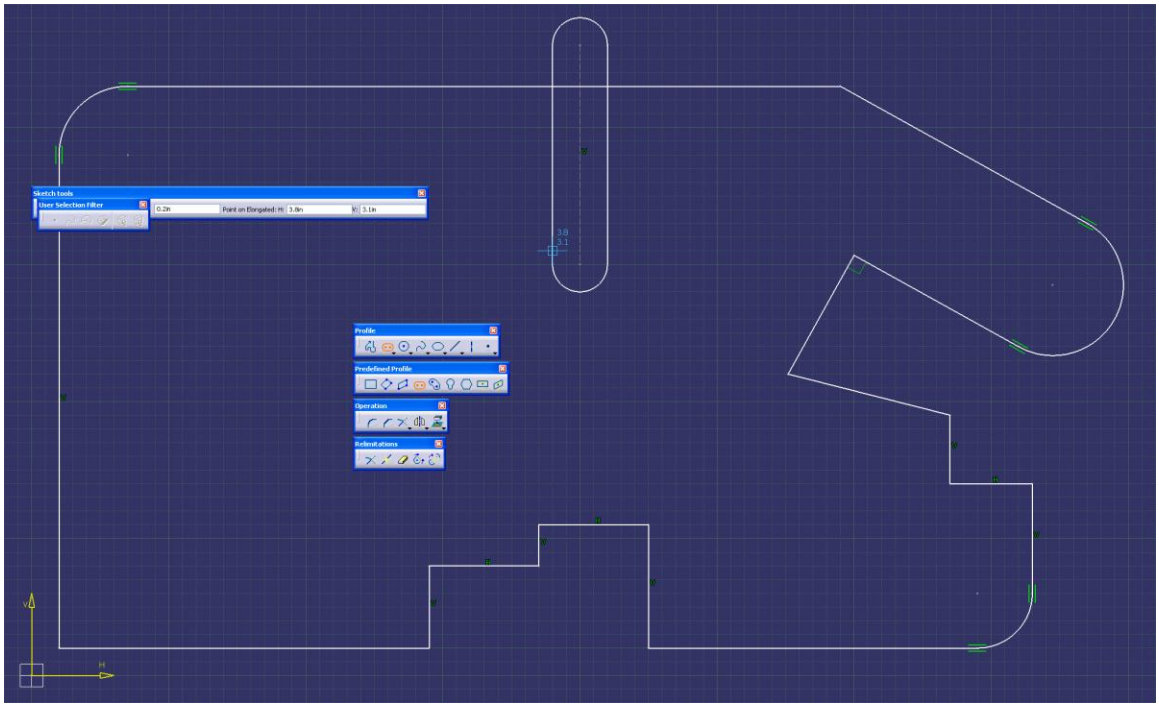


- When creating the last tangent arc, locate the arc's endpoint in order to align it to the profile starting point.

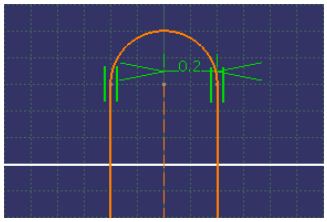


- Create all necessary linear segments to create the creation of the rough contour.

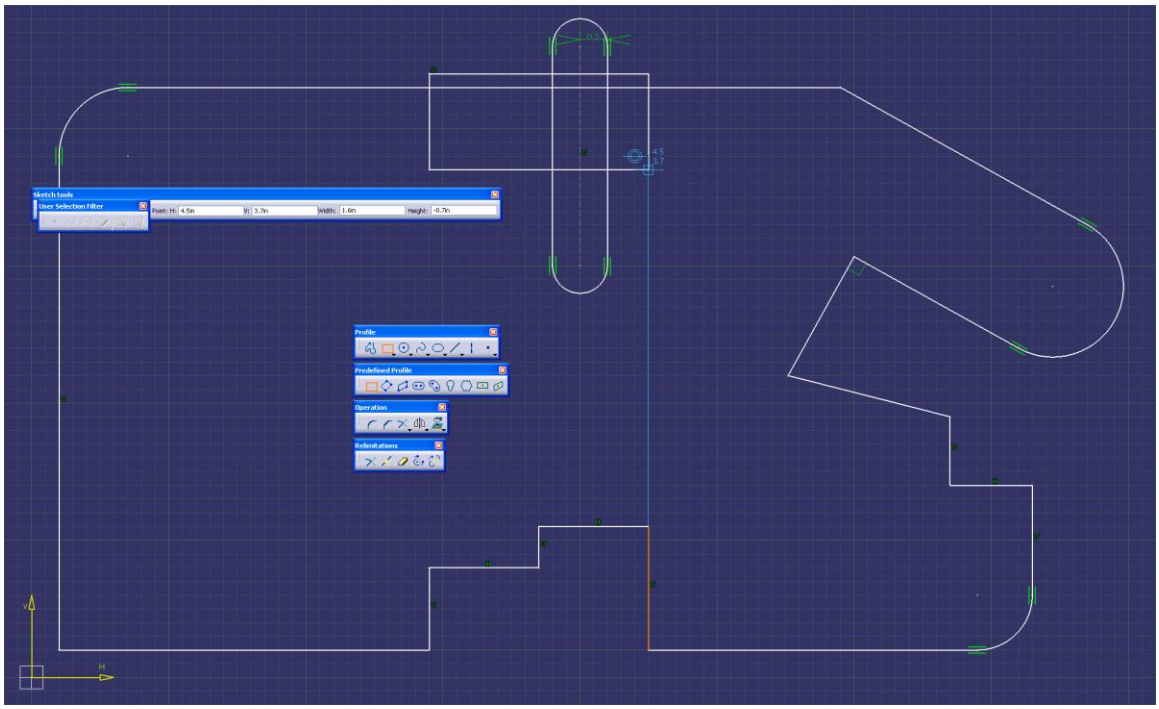
- Click the ***Elongated Hole*** tool icon  to create the rounded contour at the top of the profile. Try to use the ***Smart Picking*** behavior to align the side of the elongated hole contour on the bottom reference as illustrated on the drawing.



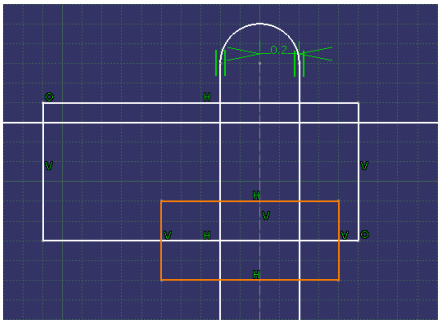
- Note the dimension that is automatically created in this case.



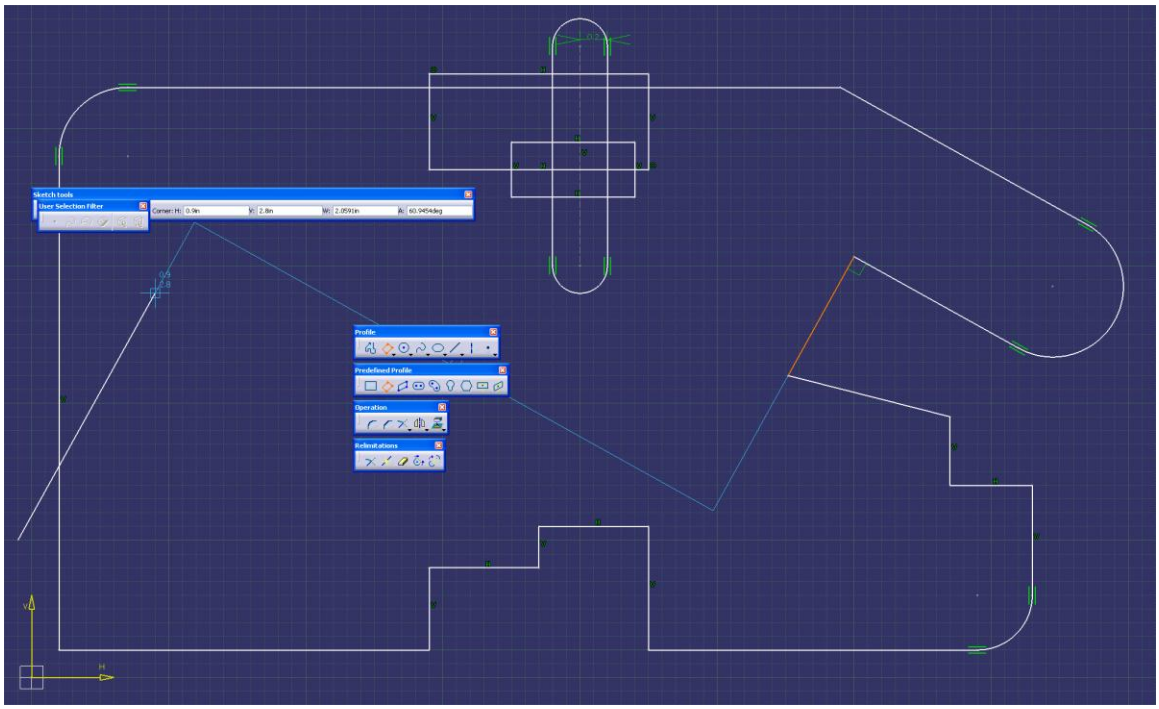
- Click the ***Rectangle*** tool icon  to create a rectangle aligning on the bottom references, as illustrated on the drawing. Try to use the ***Smart Picking*** behavior to connect on the geometry.



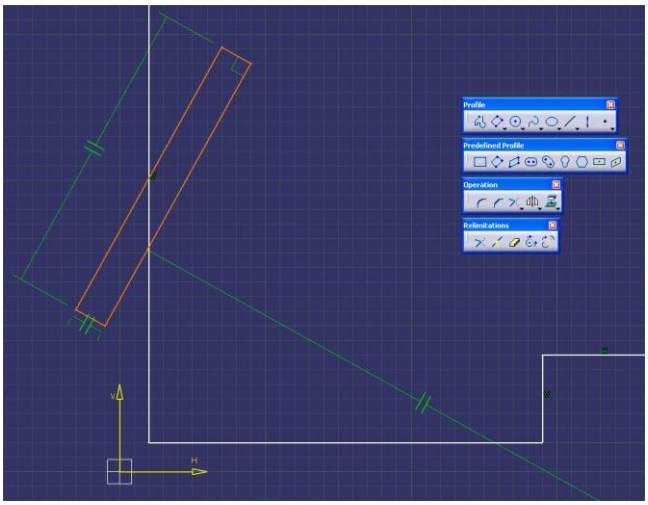
- After the creation of the rectangle, note the ***green*** circle that represents a coincidence constraint. By bringing the mouse cursor on that circle, a second circle appears near the bottom reference element and a temporary phantom line is displayed between the two elements.
- Click the ***Rectangle*** tool icon  to create a second rectangle.



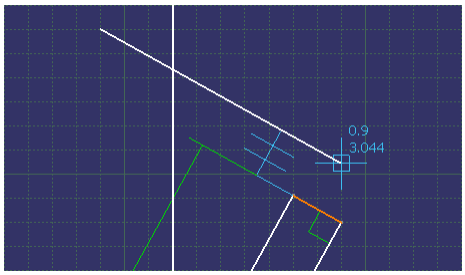
- Click the ***Oriented Rectangle*** tool icon  to create a rectangle from which the base is made parallel to the reference element found on the right side of the contour, as shown on the drawing.



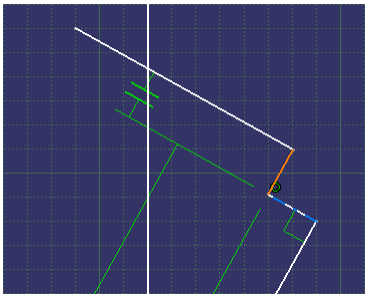
- Complete the creation of the rectangle.



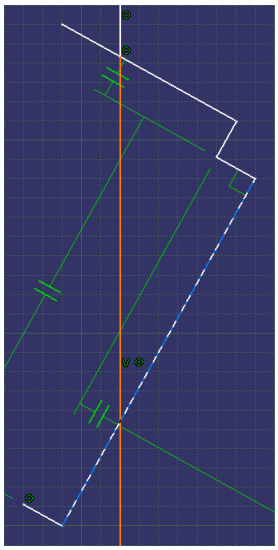
- Click the ***Line*** tool icon  to create the top element.



- Click the ***Trim*** tool icon to complete the internal corners.

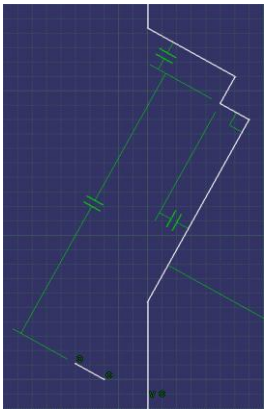


- Double-click the ***Break*** tool icon and use it to section the left side vertical line. Select the vertical line first and the top intersecting line. Since the tool is still active, select the bottom vertical segment and the bottom intersecting line.

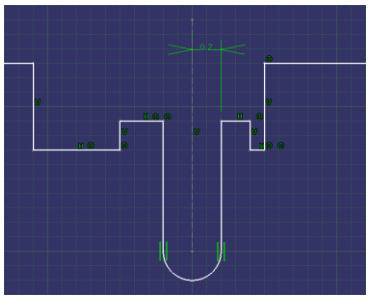


- Click the ***Break*** tool icon once again to deactivate the continuous mode. The vertical line is now divided in three independent segments.

- Select the middle vertical segment and click the ***Cut*** tool icon  to delete it.
- Click the ***Trim*** tool icon  to complete the external corners.

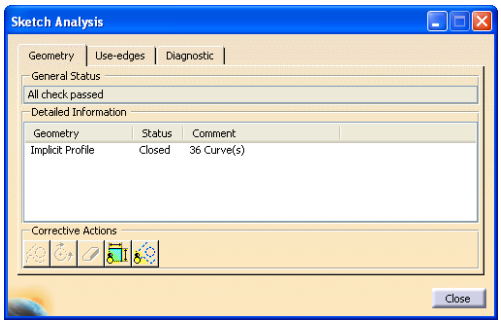


- Select the floating line segment and use the DELETE key to suppress it.
- Double-click the ***Quick Trim*** tool icon  to clean the top of the contour. Do not forget that the tool completely erase independent entities and delete portions of entities crossed by others.
- Click the tool icon again to deactivate the continuous use of the tool.



- In the ***Tools*** pull-down menu, select the ***Sketch Analysis*** tool to get a diagnostic of the contour.

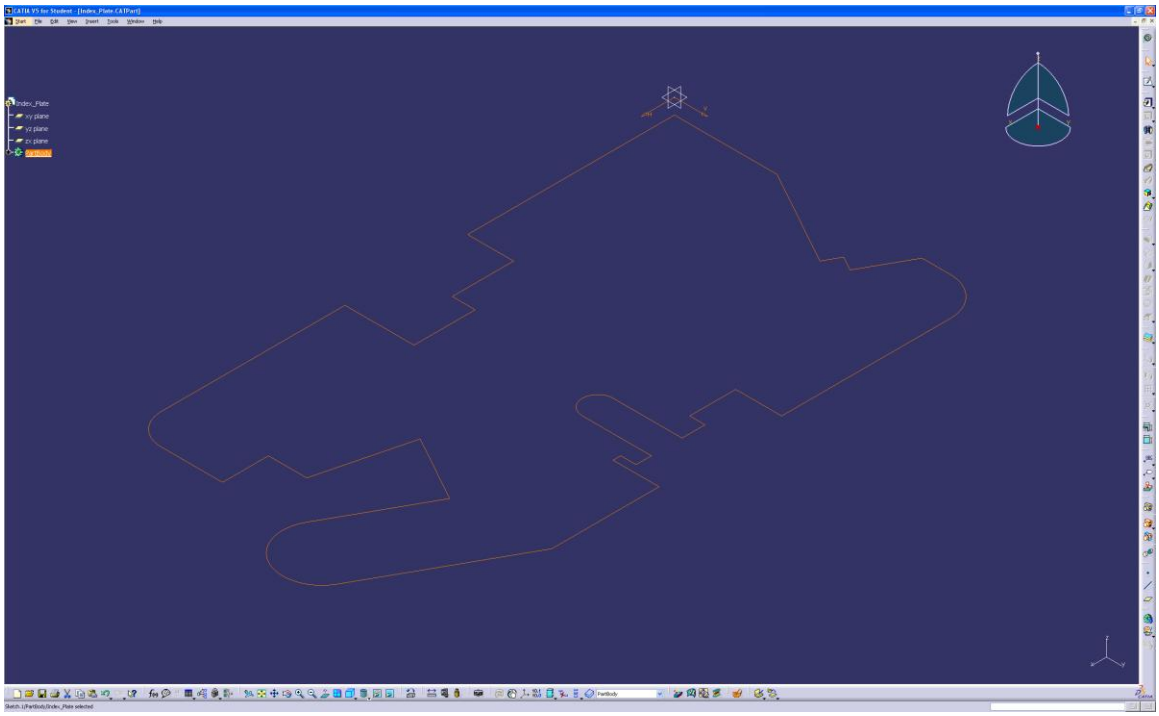
- If the geometry was properly modeled, a single contour should be present.



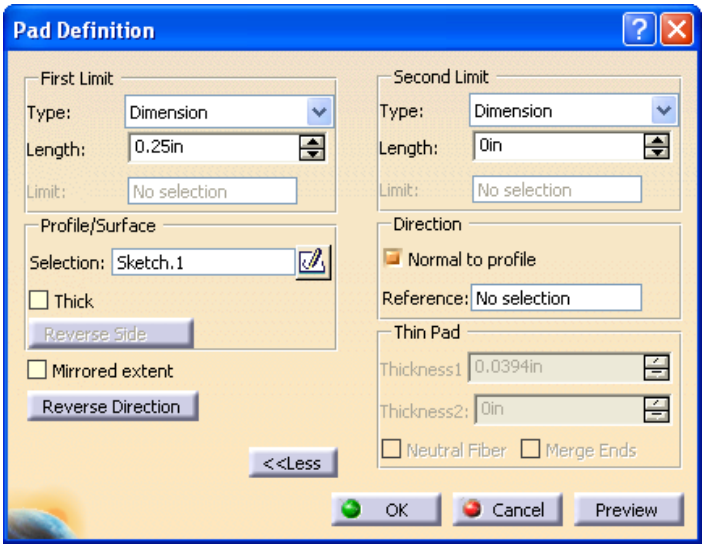
- Use the ***Exit workbench*** tool icon  to exit the ***Sketcher*** and get back to the 3-D environment.

5 – Create a solid and add a material

- Click the ***Isometric View*** tool icon  to better visualize the geometry.



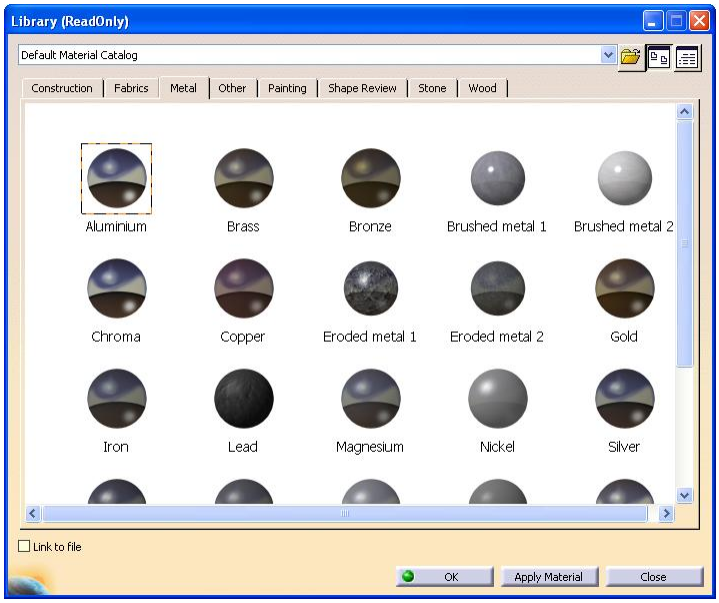
- Click the ***Pad*** tool icon  to start to create the solid model.
- In the ***Pad Definition*** dialog box, click on the ***More*** button to see all settings related to the command.
- Change the value of the ***Length*** parameter in order to make it *.25* inch.



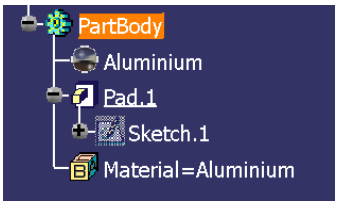
- The solid model is now created.

- Click on the ***XYZ*** tripod in the screen's lower right corner. The solid object darkens.
- Zoom in in order to increase the size of the ***Part Specification Tree*** and then click on the *Plus* sign located near the ***PartBody*** item to expand the tree. The *Plus* sign is now replaced by a *Minus* sign.

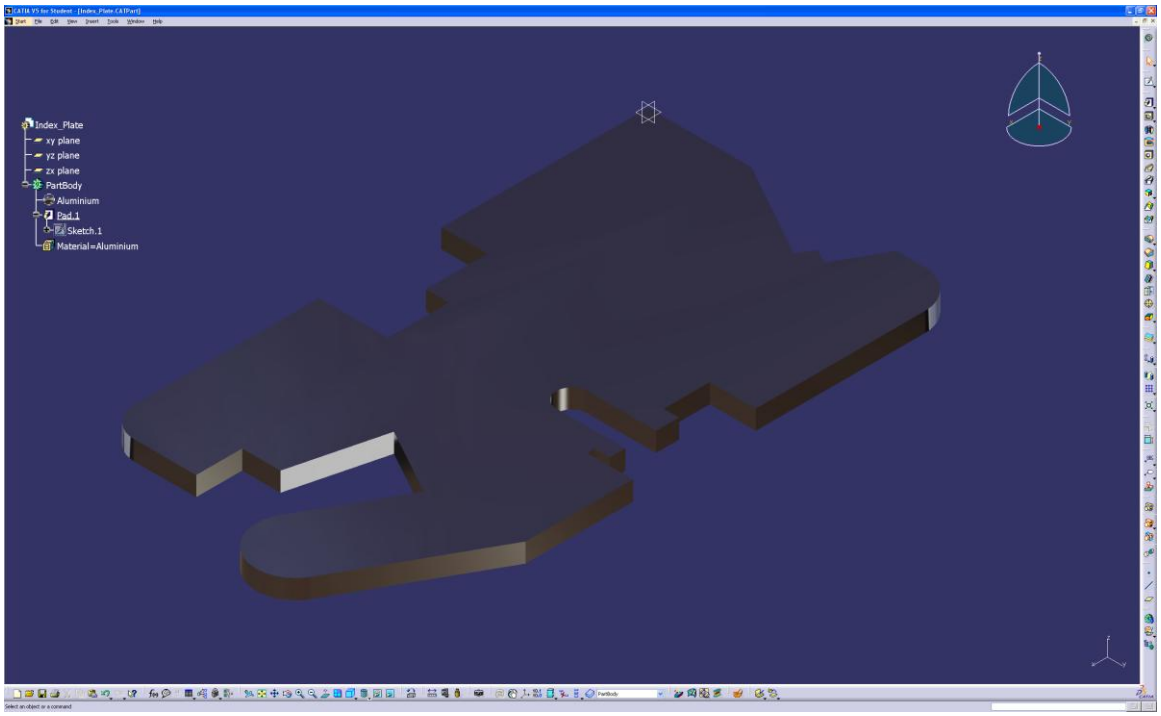
- Click on the ***XYZ*** tripod again to reactivate the part.
- Click the ***Apply Material*** tool icon  to assign the part a material.
- Select the ***Metal*** tab, select ***Aluminum*** and select the ***PartBody*** item in the ***Part Specification Tree*** prior to click the ***Apply Material*** button.



- The material is now present in the ***Part Specification Tree***.

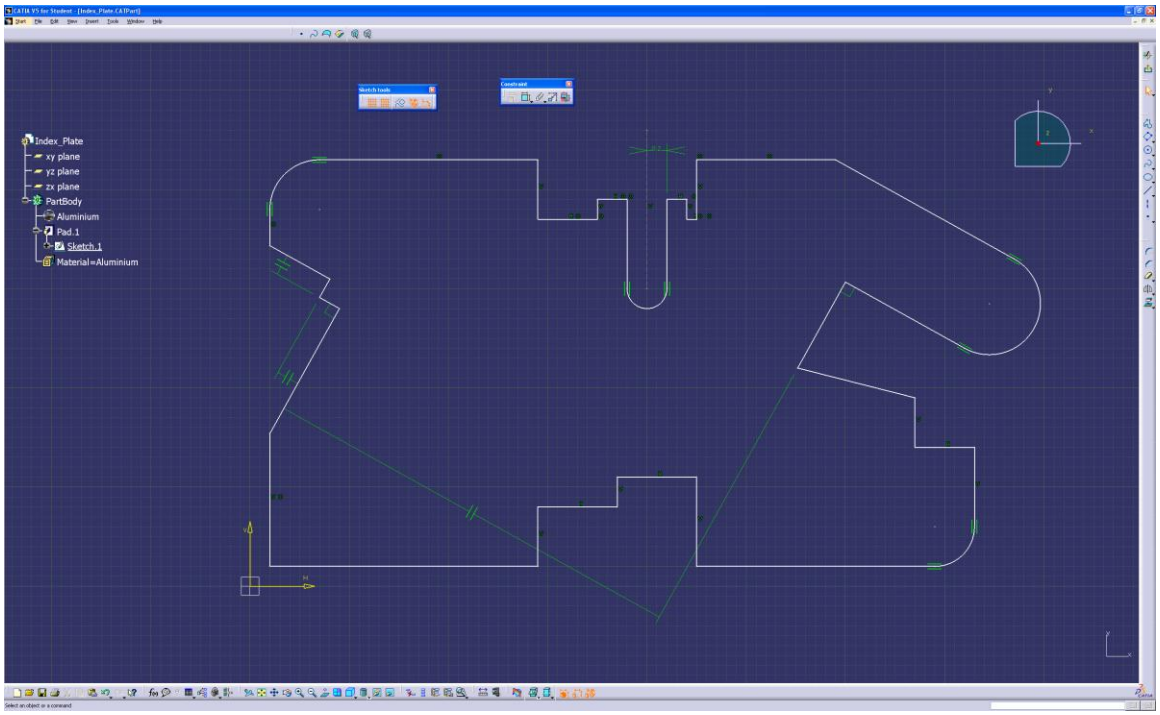


- By clicking the ***Shading with Material*** tool icon  , the part will be displayed with a different texture.

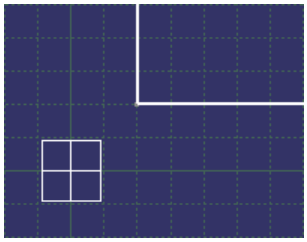


6 – Edit the sketch to get the proper shape

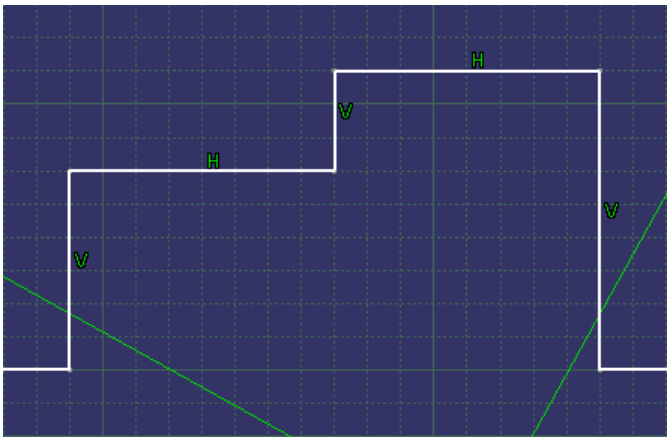
- Double-click on the ***Sketch*** item found under the ***Pad*** item, in the ***Part Specification Tree***. The part contour sketch will become available. Move the profile and operation related toolbars out of the working space and make the ***Constraint*** toolbar readily available.



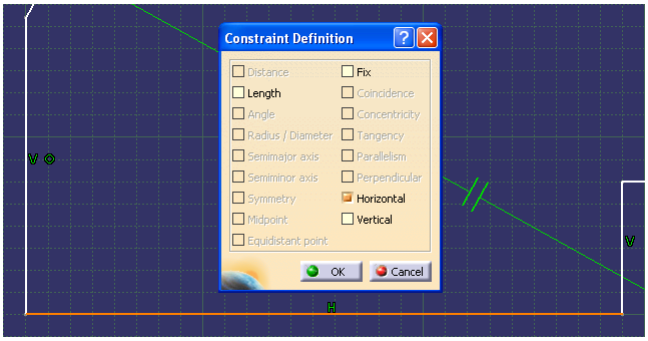
- Click the ***Hide/Show*** tool icon  to hide the ***V*** and ***H yellow*** axes as well as the point found at their intersection, keeping then the three default planes visible only.



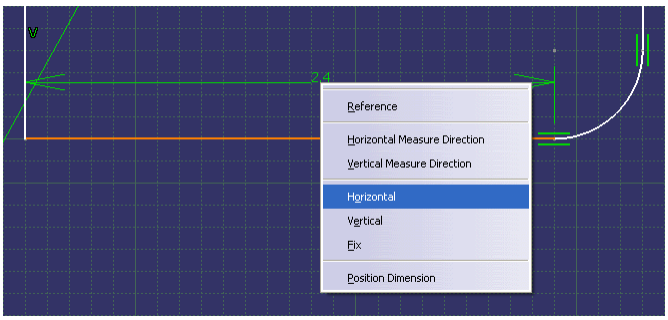
- Have a look at all the horizontal and vertical line segments of the contour. Make sure they all have a ***green V*** or ***H*** geometrical constraint next to them.



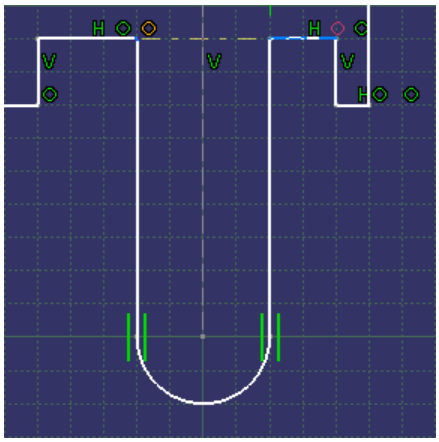
- If some elements do not have such a constraint, it is possible to select them individually and to click the ***Constraint Defined in Dialog Box*** tool icon  to access the related dialog box and select the appropriate constraining element.



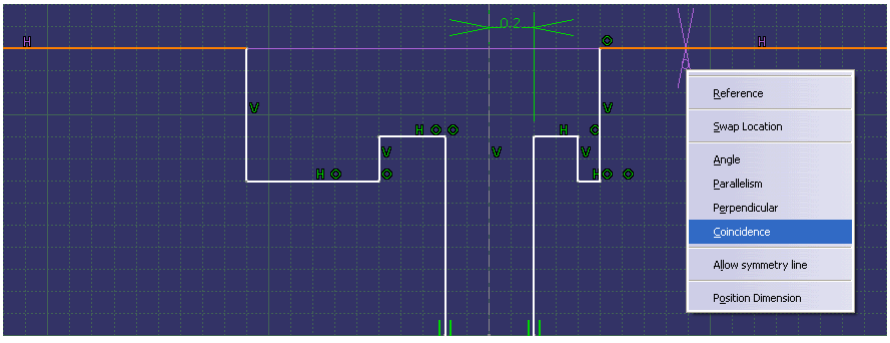
- It is also possible to click the ***Constraint*** tool icon  , to select and then right-click an element to constraint in order to access its context menu. All applicable geometrical constraints related to the selection will be available to the user.



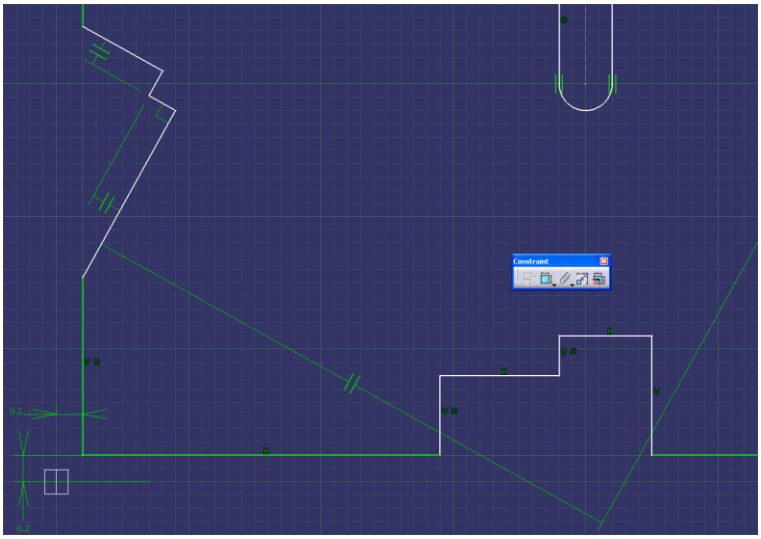
- Since the ***Constraint*** tool icon can be double-clicked to keep the selection mode continuous, this approach is usually more optimal in order to add both geometrical and then dimensional constraints, later.
- When all the horizontal and vertical constraints are applied to related elements make a verification of all coincidence (circle symbol) and tangency (equal sign symbol) constraints by bringing the mouse cursor over each of them and checking what elements are related by the constraint.



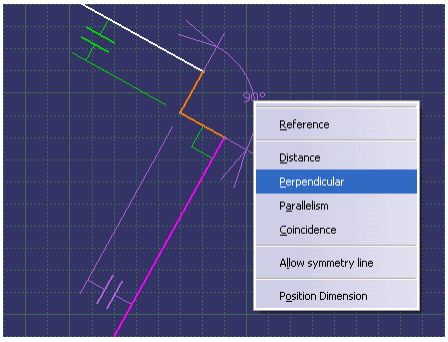
- Note that in the case of coincidences, a second circle symbol appears during the verification.
- When all existing constraints have been checked and none is creating problem by projecting on a distant element, add missing constraints. Again, the ***Constraint Defined in Dialog Box*** or ***Constraint*** tools can be used.



- Create dimensional constraints between the left and bottom line segments located close to the origin and their respective planes passing through the origin. Some elements turn ***green*** in the sketch. Do not edit the dimensions yet. They will be edited at the end of the process only.

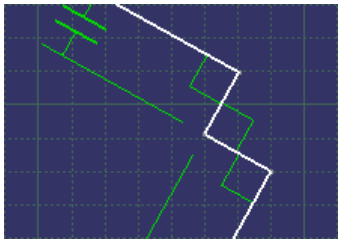


- Click the ***Constraint*** tool icon  and select the two bottom left side check segments. They will turn purple to show that an angular dimension would over-constrain them. Right-click to access the context menu and click the ***Perpendicular*** option.

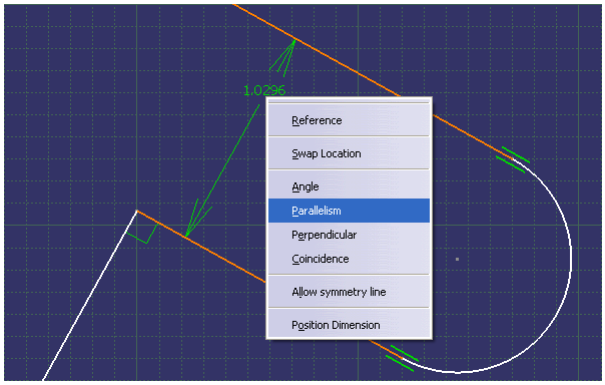


- Repeat the procedure for the next corner. Note that such perpendicular constraint does not have to be created between segments being vertically and horizontally constrained since they are implicitly orthogonal. As a basic drawing interpretation

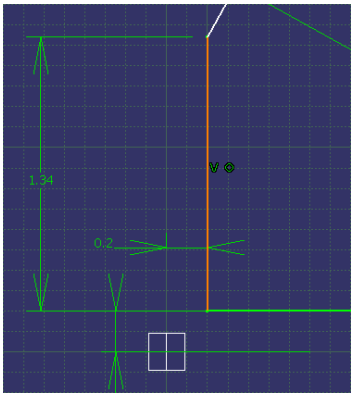
rule, if two segments look perpendicular in a drawing, they are. If they are not, a dimension indicating the angle (ex: 89° or 91°) should be present!



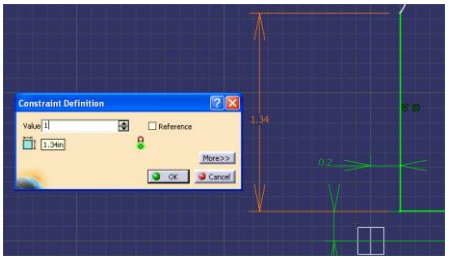
- Using the same approach, add a parallelism constraint to the two right hand side segments joined by a tangent arc.



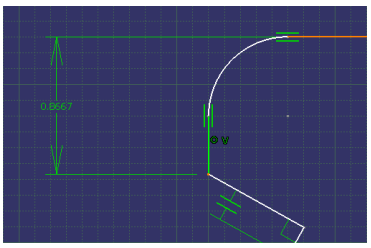
- Geometrical constraints should now all be present in the sketch.
- Click the ***Constraint*** tool icon  and select the bottom left vertical line. A dimensional constraint is created. Move the cursor and click to locate the dimension. Try to duplicate what is presented on the drawing.



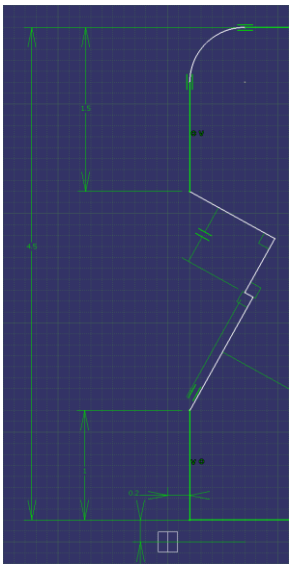
- Double-click the constraint to access its dialog box. Modify the value to make it *1.0* inch. Click the ***OK*** button to complete the edition.



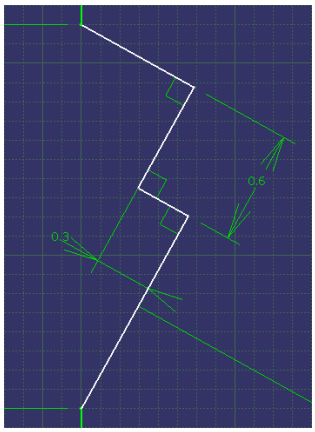
- Use the same constraining tool to select the top left horizontal line and the implicit endpoint that define the top left vertical segment. Create the dimensional constraint and edit it to make it *1.5* inches.



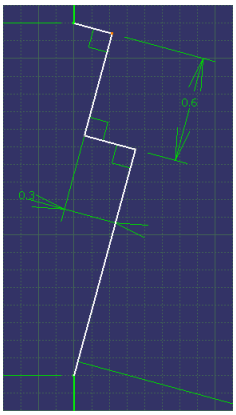
- Create the top and bottom outermost horizontal segments and create a *4.5* inches dimensional constraint between them. This will set the total height for the part.



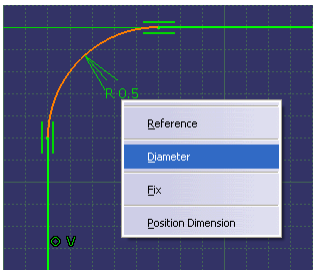
- Double-click the ***Constraint*** tool icon  to make its use continuous. Add the missing constraints to the lines defining the check found on the left side of the profile. Note that the geometry does not turn ***green***. Click the icon to stop the continuous use behavior.



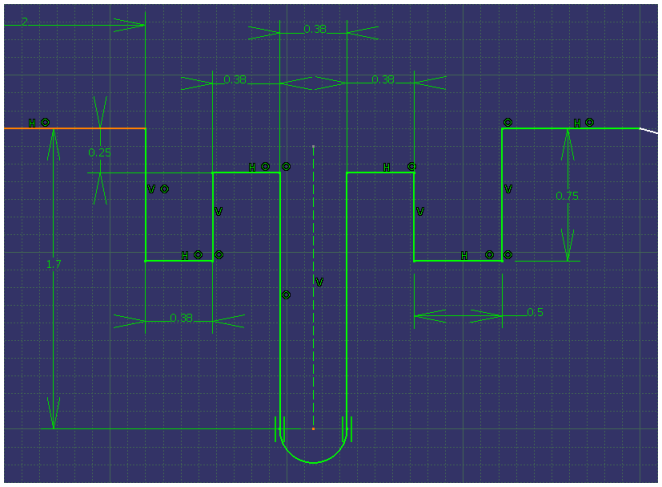
- Select one of the side check segments implicit endpoint and, keeping the left mouse button depressed, move the cursor slowly. The geometry must react and the right side segment constrained with a parallel constraint should react as well. In this case, an angular constraint is still missing. The left side geometry will only turn ***green*** when the right side segment will be fully constrained. When constraining a profile, this approach can be used at any time. If some geometry is still shown in ***white*** and it is not clear to the user to see what is missing to fully constrain it, just try to move the geometry to see what motion is still allowed. This will give a good lead on the missing constraints.



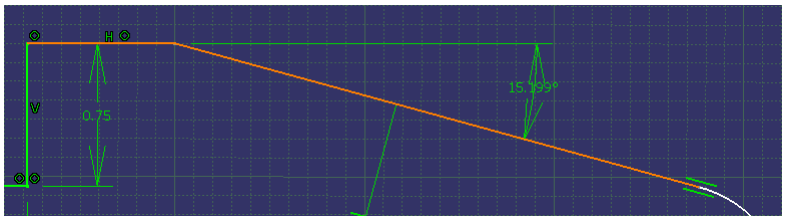
- Add a new dimensional constraint by adding the top left arc. If a diametral constraint ever appears, right-click to access the context menu and select the ***Radius*** option that would be available.



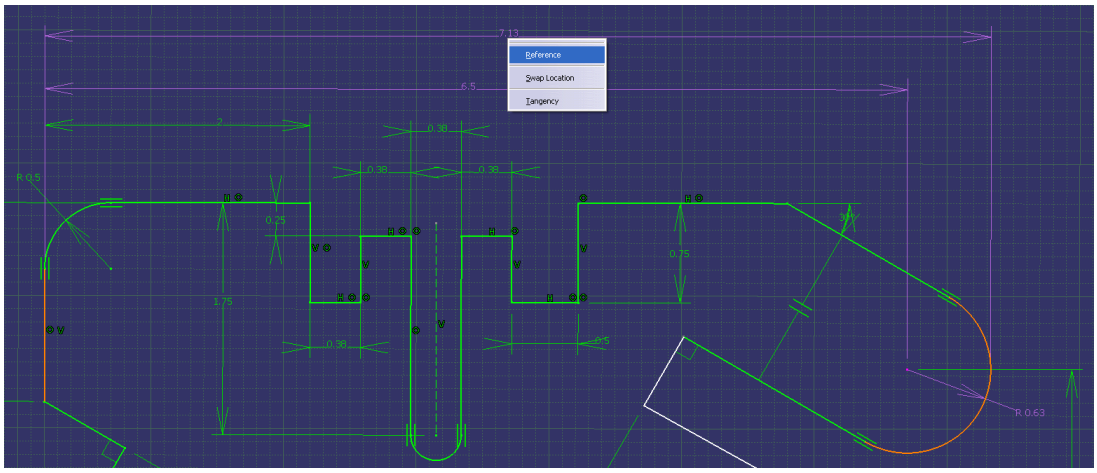
- Double-click the ***Constraint*** tool and add constraints to the top centered geometry. Click the icon once again to deactivate the continuous behavior. Note that the dimension that was existing between the middle slot side and the construction line was deleted and replaced by a dimension driving the slot width. The length of the slot is defined by selecting one of the top horizontal segments and the arc center point. Even if the drawing show a dimension saying *2 X 0.38*, two independent dimensions are created at this time.



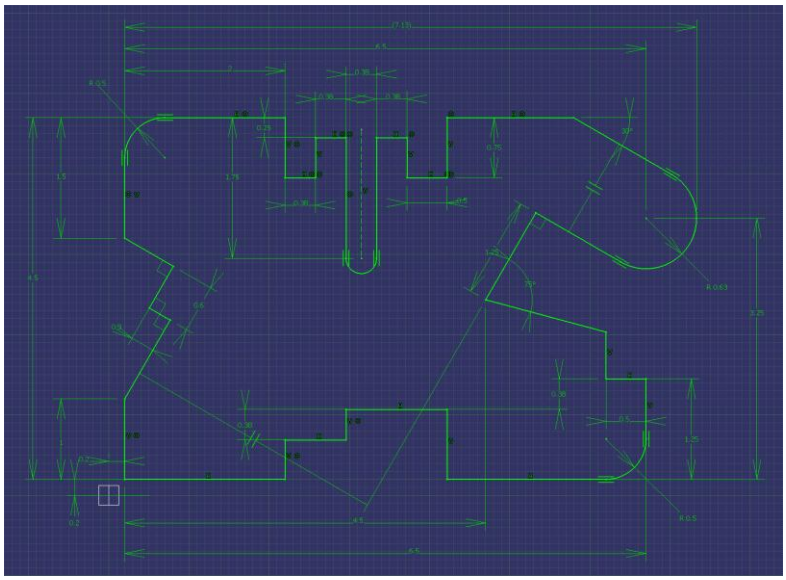
- Use the ***Constraint*** tool again to add an angular constraint by selecting the top right horizontal and inclined segments. Move the cursor in order to create the appropriate dimension. Not that the left hand side check geometry is now fully constrained and displayed in ***green***.



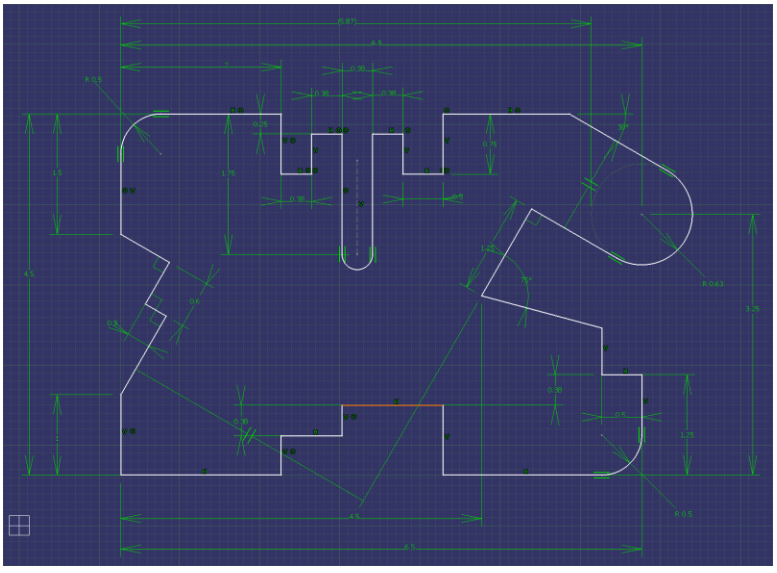
- Take the approach used previously to dimension the horizontal distance between the left hand side contour's vertical segment and the right hand side arc center point. Add a constraint to set the vertical distance between the contour's bottom line and the arc center point. Add a radial constraint to the arc. All elements should now be displayed in ***green***.
- Create a horizontal dimension between the left hand side vertical segment and the right hand side arc. This will temporarily over-constrain the sketch and some dimensions will turn purple. Without clicking to locate the dimension, right click to access the context menu and select the ***Reference*** option. The created dimension value will be surrounded by parenthesis and the whole dimension will be displayed in ***green***. Reference dimensions are not driving the sketch, but are driven by the sketch. Edit the 6.5 inches dimension to see how it does react.



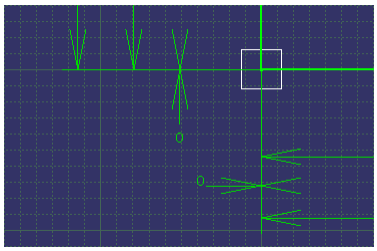
- Add the missing dimensions to the sketch. All contour elements should now be displayed in ***green***. Relocate the dimension in order to try to duplicate the drawing.



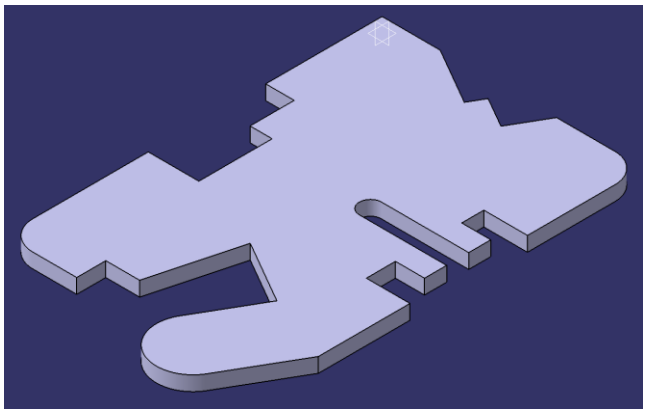
- Delete the two first dimensional constraints that were created between the contour's bottom left corner elements and the two reference planes. All contour elements will turn ***white*** again. Select any of the contour elements and while keeping the left mouse button depressed, move the cursor. The sketch should move without any distortion.



- Re-create the two constraints and make them equal to *0*. If it ever becomes necessary to move the sketch away from the origin, just edit or delete the two constraints.

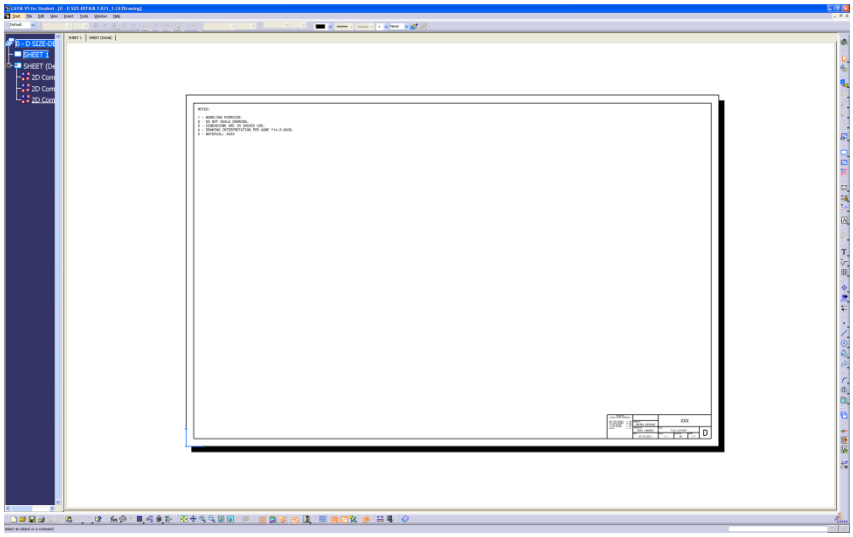


- Exit the sketch to obtain an updated version of the solid object.



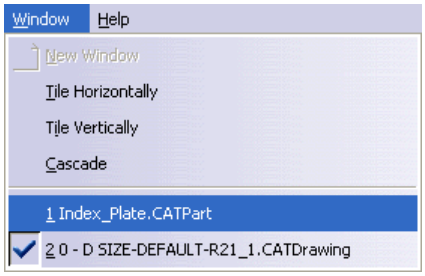
7 – Create a drawing and extract the part views

- Use ***File>New from*** to start a new drawing file from the ***R20-D_Size.CATDrawing*** prototype file. A drawing file should appear on the screen and the ***Drafting*** workbench interface should be available to the user.

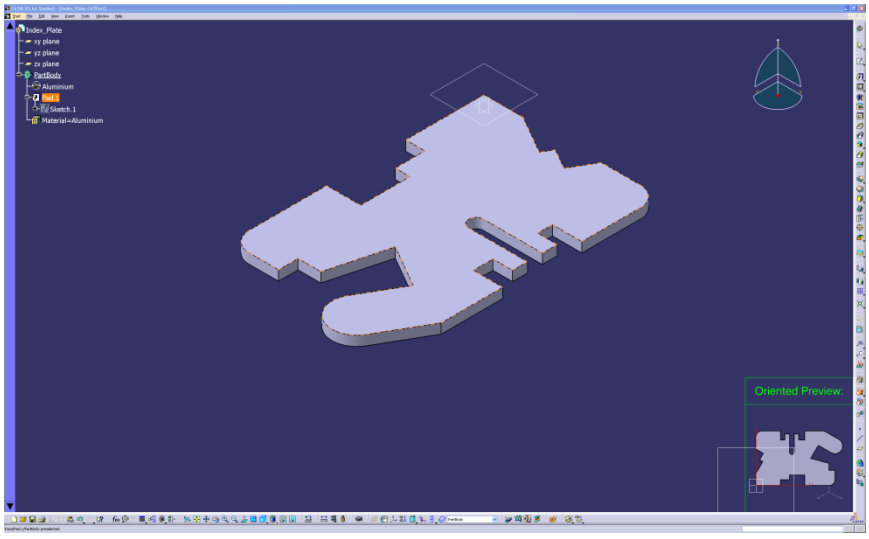


- Move the ***Views*** toolbar in the working area. Make the ***Projections*** toolbar readily available.

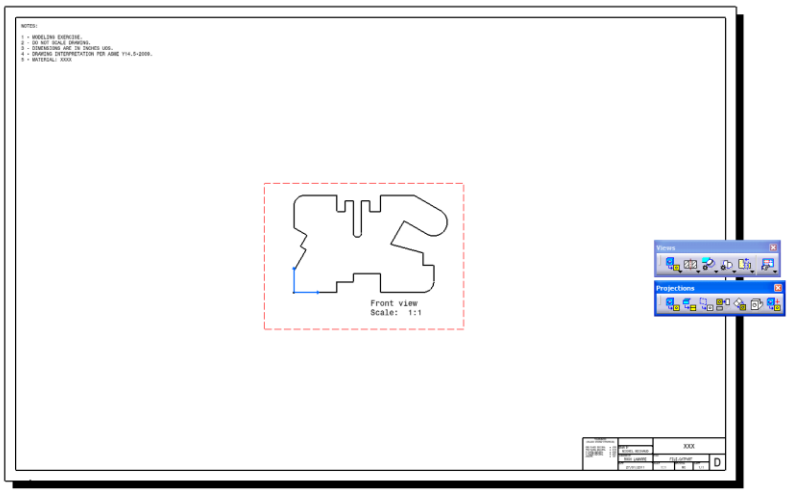
- Click the ***Front View*** tool icon  and access the 3-D part by selecting its option in the ***Window*** pull-down menu.



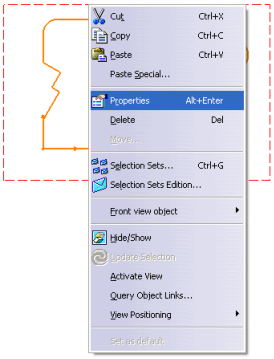
- Move the cursor over the solid part. Keep checking the ***Oriented Preview*** window found in the bottom right corner of the screen, to see what would be extract as a first view.



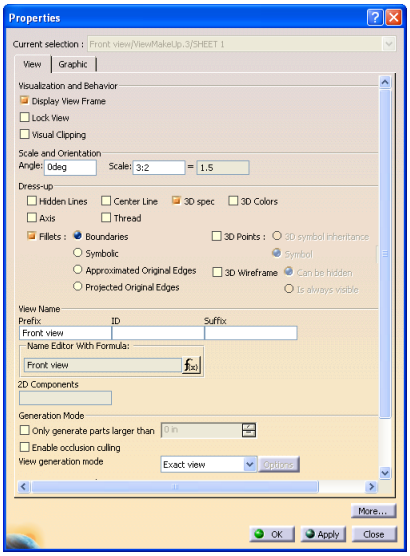
- By clicking a solid face, the user is automatically returned to the Drafting workbench interface. A first view is displayed in the drawing frame. It is possible to re-orient the view using the straight and curved ***blue*** arrows located in the top corner of the screen. When the desired view is obtained, just click the ***blue*** dot in the middle of the two ***blue*** curved arrows or click in the ***white*** working space. The solid view is extracted and a name and scale should appear under the part representation.



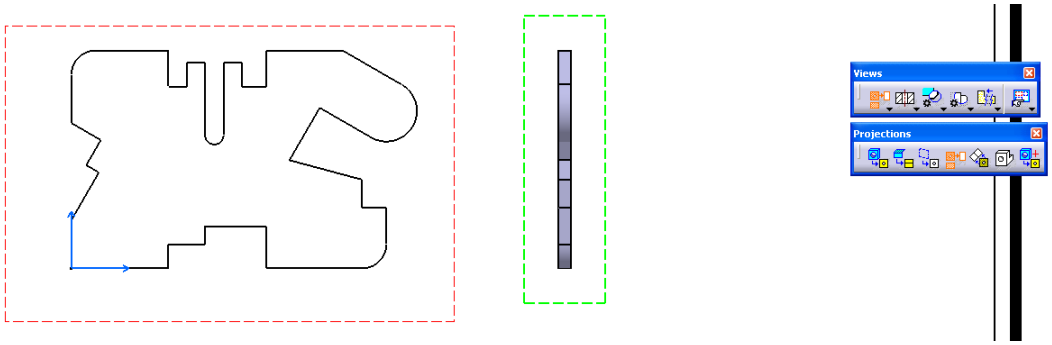
- Delete the view name. In standard mechanical drawings, no view name appears for orthogonal views. To do so, just click on the text and use the DELETE key.
- The part is small to be displayed in a D-Size sheet format. Bring the cursor over the ***red*** dashed border and right-click to access the context menu. Click on the ***Properties*** option.



- In the ***View*** tab, change the S***cale*** value to *3:2* in order to enlarge the representation of the part.

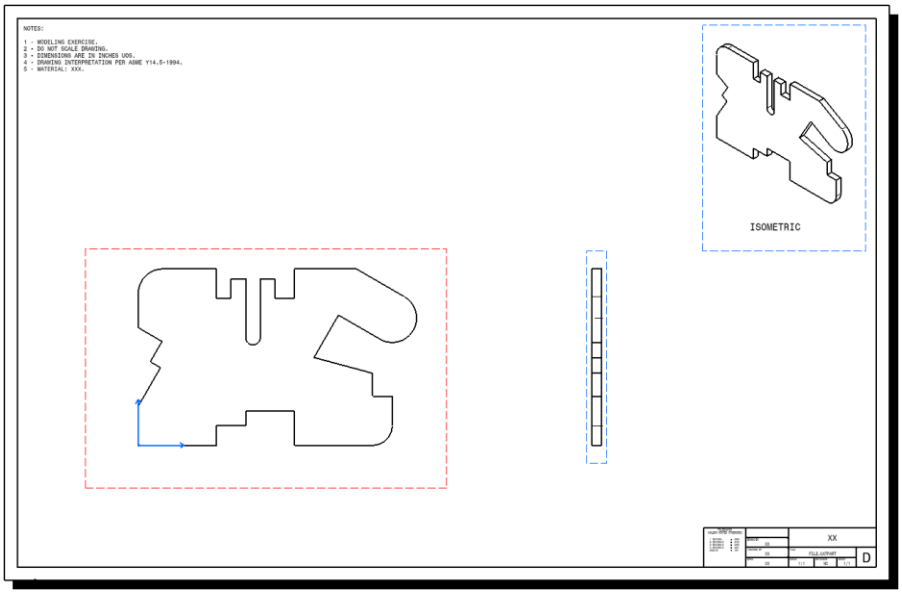


- Move the view to the bottom left section of the sheet by selecting it by its ***red*** border.
- Click the ***Projection View*** tool icon  and move the cursor to the right of the existing view to see the right hand side view of the part.

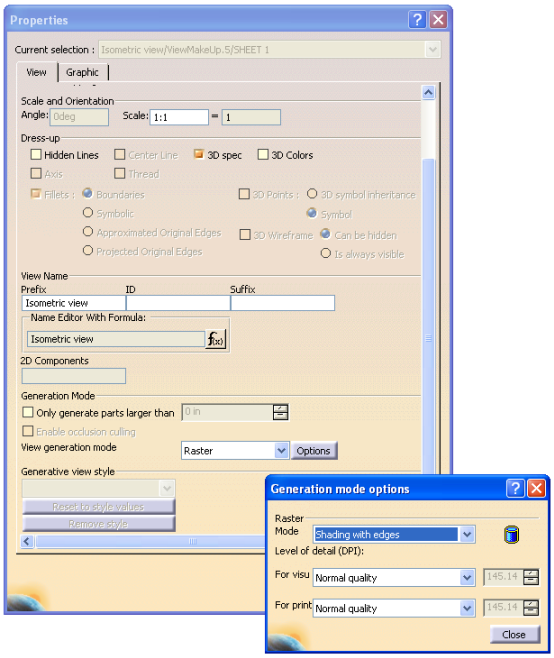


- For a simple plate like this, two orthogonal views are usually enough. Click in the ***white*** working space to complete the view extraction and delete the view name.
- Click the ***Isometric View*** tool icon  to obtain a "like 3-D" part representation on the drawing. Again, the user is asked to access the 3-D part environment. This can be done by using the ***Window*** pull-down menu.

- Once in the part environment, if the user wants to obtain a real isometric view, it is possible to click the ***Isometric View*** tool icon  found in the ***Part Design*** workbench interface in order to have the solid part properly represented. If this view does not satisfy the user, it is possible to manually re-orient the part first and then to click anywhere on the solid part. In this case, chances are very few that a real isometric view will be obtained, but a good axonometric view is correct too. Just take care to try to keep some sort of vertical representation of lines to respect drafting standards.
- An isometric (or axonometric) view is usually identified as such on a drawing to avoid any confusion with any auxiliary view that could look similar. Double-click the isometric view text to access the ***Text Editor***. Rename the view ISOMETRIC (even if it is axonometric) using capital letters only. Scale factor is not always added to isometric views since these views should be used for representation only and no measure should be taken on them.

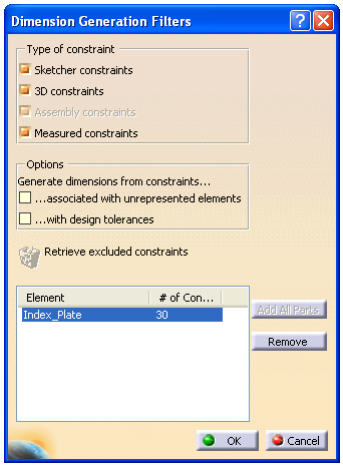


- This is not standard drafting, but for illustration purposes, if a shaded representation of a view is desired, one must modify the properties of a view to get it. Just access the ***View generation mode*** option in the ***View*** tab of the ***Properties*** dialog box and change it to ***Raster***. Then, click the ***Options*** button to make a choice through the different ***Raster Mode*** options.

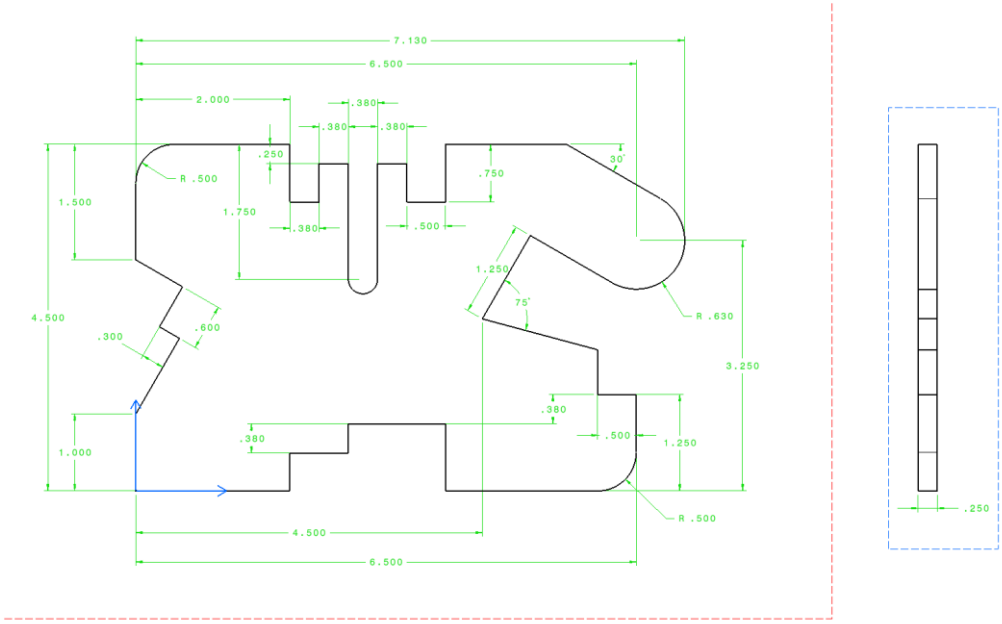


8 – Extracting dimensions from the sketch

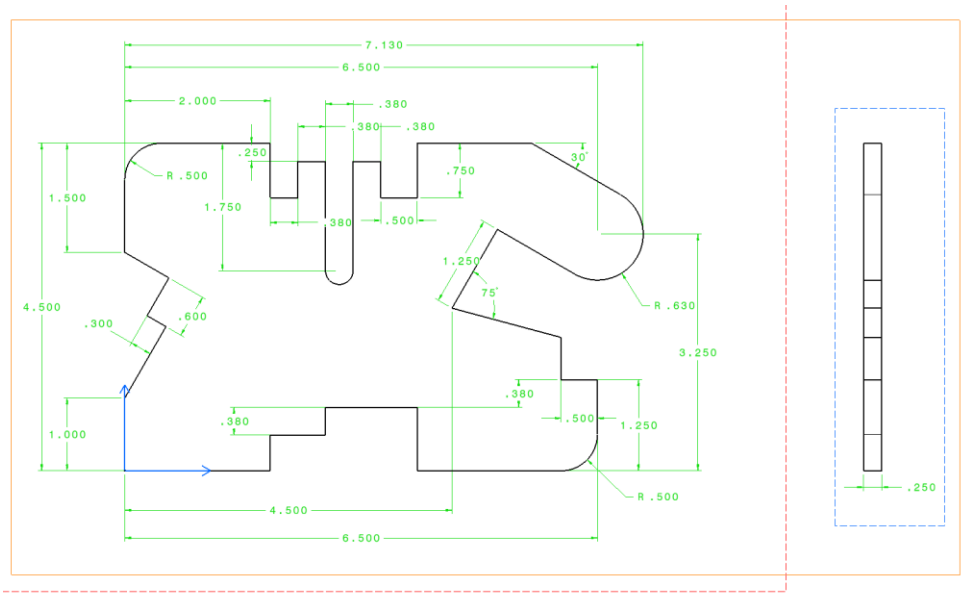
- Make sure the front view of the part is active. This is confirmed by the ***red*** dashed view frame. If it is not the case, double-click the ***blue*** view frame to activate the view and make the frame turn ***red***. Click the ***Generate Dimensions*** tool icon  to automatically generate the drawing dimensions from the dimensional constraints that were created in the sketch. Click the ***OK*** button found in the dialog box that becomes available to the user.



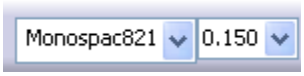
- Generated dimensions will be displayed in ***green*** as compared to the ones that will be created manually, displayed in black. Using this approach is not necessarily perfect, but if care is taken locating the different dimensional constraints in a sketch, a good amount of time can be saved redoing similar tasks in the ***Drafting*** workbench interface.



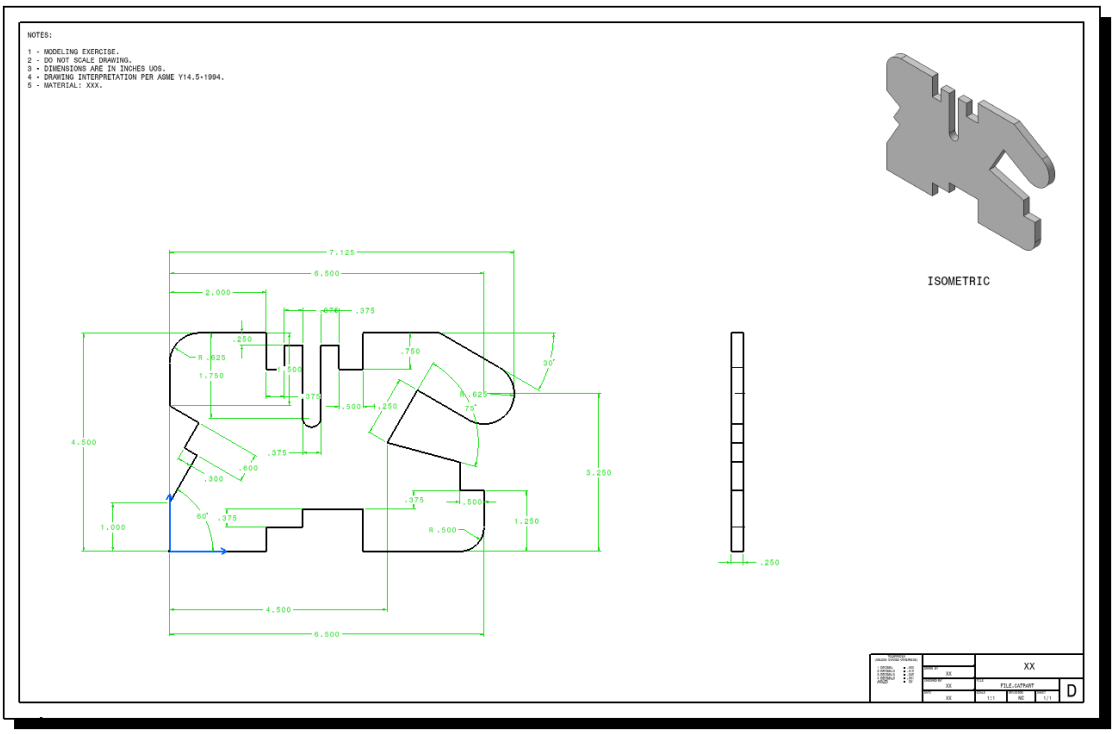
- Various standards can be used for dimension representation. In this tutorial, to make the dimensions readable when printed on an 11"x17" sheet, their height will be changed to *.150* inch, which is the size of the text notes in the top left corner of the sheet.
- Click in the lower left corner of the screen and while keeping the left mouse button depressed, move the cursor to create a large selection window that will contain all dimensions. Release the mouse button to complete the selection.



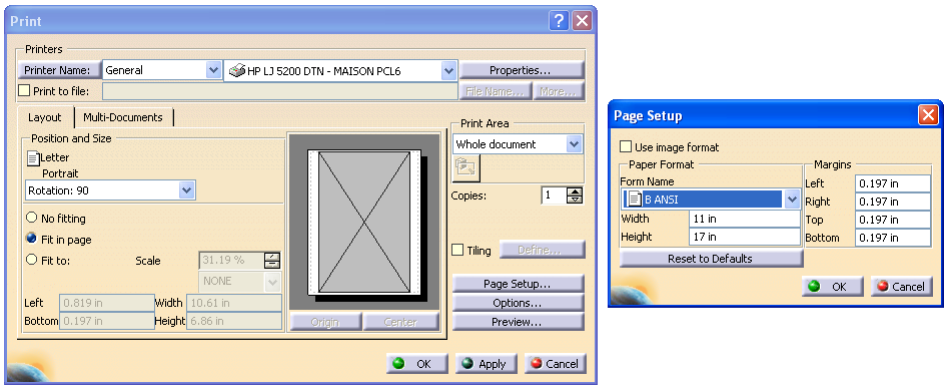
- Move the cursor in the pull-down menu area. Just after the selection, there are three drop-down menus showing nothing as content. Click on the down pointing arrows of the middle and the right one to select the font and the font size.



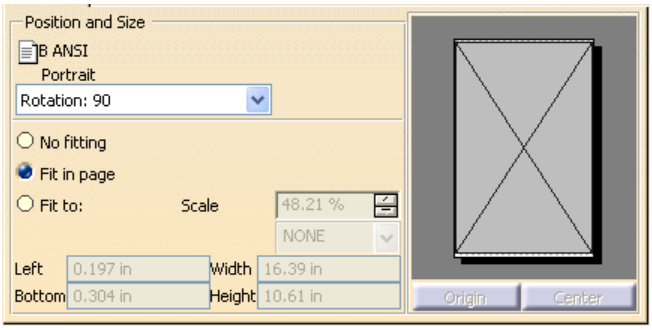
- Click in the ***white*** working area to de-select everything.
- Relocate the different views if required.
- If desired, click the ***Display View Frame as Specified for Each View*** icon  to deactivate the command. The icon will return to its original colors  and the view frames will be hidden.



- The drawing is considered complete for now. Some dimensions have to be reworked, but this can be done later, when dimensioning tools will be mastered.
- Use the ***Print*** option found in the ***File*** pull-down menu.
- In the ***Print*** dialog box, click the ***Page Setup*** button and make sure to select the proper sheet format. If a *B ANSI* format is available, use it. If not, use whatever is available.



- Use the ***Fit in page*** scaling option and use the proper rotation angle to optimize the use of the sheet to print the drawing. With a *B ANSI* sheet and the D-Size format, almost all of the sheet should be used.



- Click on the ***Preview*** button to see what the print will look like before to launch the printing process. Note that it is possible to zoom the preview using the standard mouse operation.
- Close the ***Print Preview*** dialog box by clicking the ***OK*** button.
- Click the ***OK*** button in the ***Print*** dialog box to launch the print and close the dialog box.
- Save the .CATDrawing file. Use the ***Window*** pull-down menu to access the part file. Save it as well.

Thanks to Alice Michaud for revising this text.