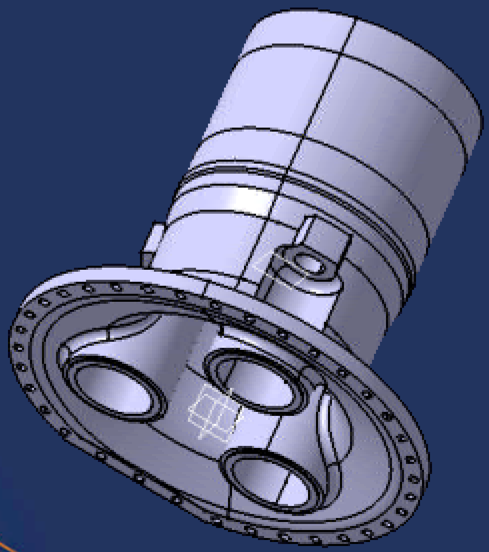
Part Design



Tutorial Objectives

Description

- This tutorial is an introduction to Part Design.

Message

- This tutorial illustrates how CATIA can
    - Design precise 3D mechanical parts with an intuitive and flexible user interface
    - Accommodate design requirements for parts of various complexities- from simple to advanced
    - Apply the combined power of feature-based design with the flexibility of a Boolean approach

Duration

- 45 minutes

Product Coverage

- Part Design

Scenario Major Steps

Here are the major steps of the scenario:

Step 1

- ✍ Add a new Part body

Step 2

- ✍ Create a Shaft feature

Step 3

- ✍ Create a Tap

Step 4

- ✍ Create a Circular Pattern

Step 5

- ✍ Create a Union Trim

Step 6

- ✍ Create an Edge Fillet feature

Step 7

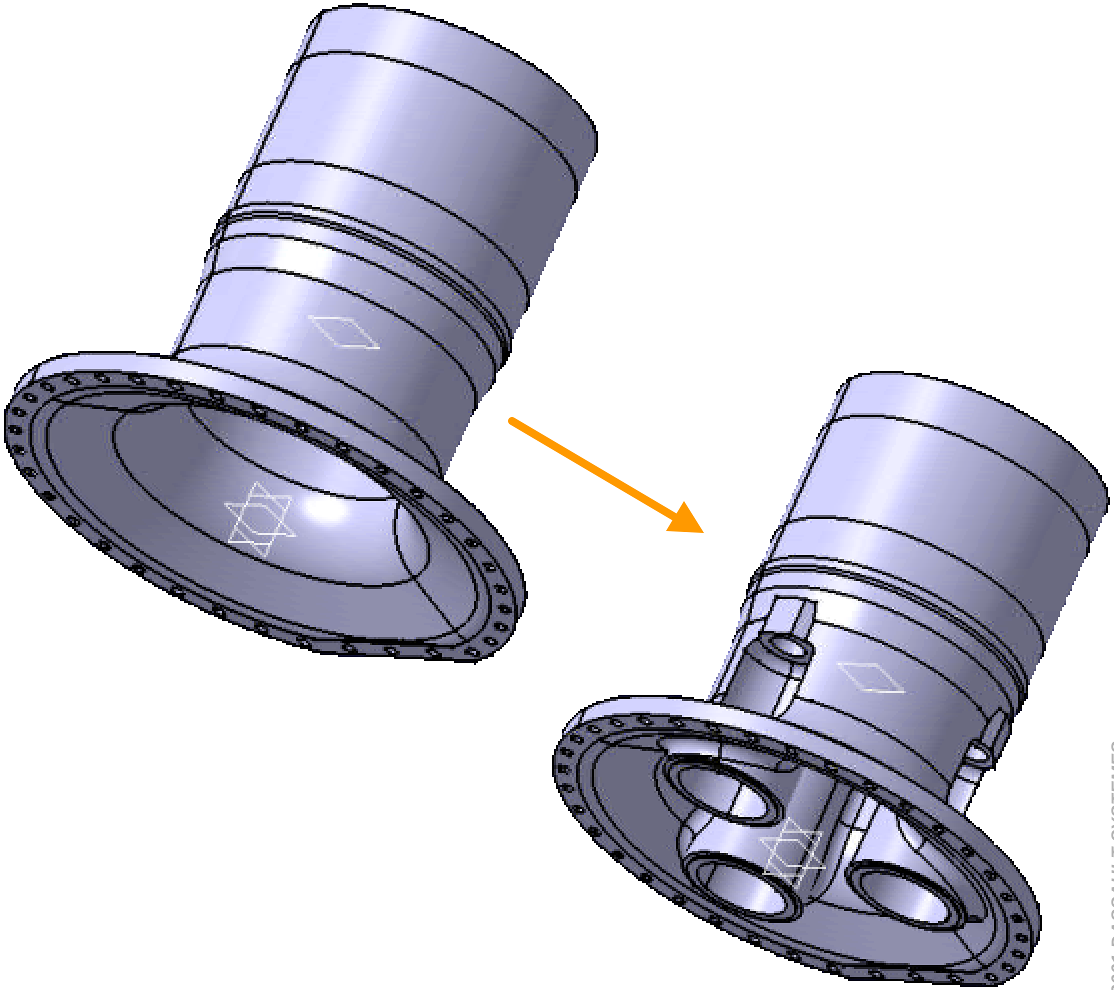
- ✍ Create a Pad feature

Step 8

- ✍ Create a Hole feature

Step 9

- ✍ Thread / Tap Analysis



Settings 1/2

Depending on your needs, you may have to modify the CATIA V5 settings (*units, default directory, visualisation parameters, etc*…)

In order to use the appropriate settings for this tutorial, you have two possibilities:

1. Do the following operations (simplest one):
   - **BEFORE STARTING YOUR CATIA V5 SESSION:**
     - *Copy or replace the directory ..\Part Design\Data**CATSettings** in:*

For NT users ➡ C:\Winnt\Profiles**XXXXX**\Application Data\DassaultSystemes

For Windows 2000 or XP users ➡ C:\Documents and settings\Profiles**XXXXX**\Application Data\DassaultSystemes

For Windows 98 users ➡ C:\Windows\Profiles**XXXXX**\Application Data\DassaultSystemes

XXXX is the name used to log on to your computer

- ***Do not forget*** *to put this folder (CATSettings) in* ***read mode****:*
  - *Select* **the folder** *(CATSettings)*
  - *Click mouse button 3 then click on* Properties *and uncheck the* Read-only Attribute
  - *Select* **all the files** *in the folder*
  - *Click mouse button 3 then click on* Properties *and uncheck the* Read-only Attribute

2. Set them manually:
   - *Launch your CATIA V5 session and do the operations from page 40 onwards*

Settings 2/2

For this tutorial you also need to install a material catalogue:

Do not do this step if you have already done it in getting started or in a previous tutorial

✍ *Copy the ..\Getting Started**Catalog.CATMaterial** file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\French directory*

✍ *Copy the ..\Getting Started**Catalog.CATMaterial** file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\German directory*

✍ *Copy the ..\Getting Started**Catalog.CATMaterial** file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials\Japanese directory*

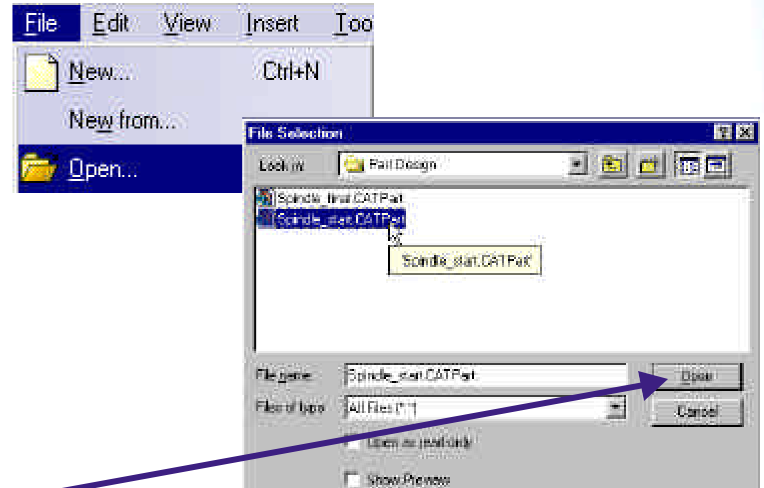
✍ *Copy the ..\Getting Started**Catalog.CATMaterial** file under ..\Program Files\Dassault Systemes\M07\intel_a\startup\materials directory*

✍ *Answer Yes in order to replace the old catalogue*

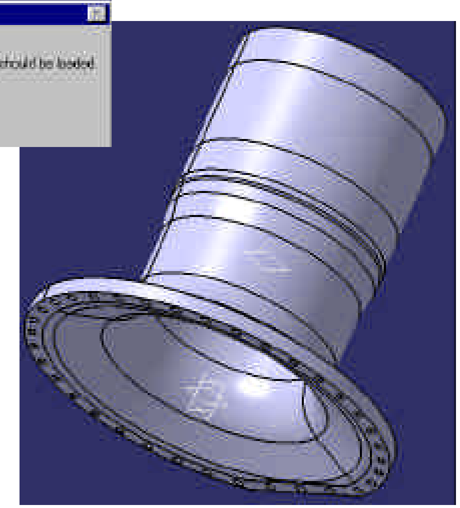
You are now ready to launch your CATIA V5 session

Step 1: Read the CATPart

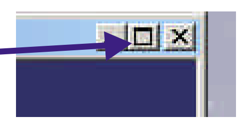
- You are going to read a CATIA V5 document
  - *Click file on the menu bar*
  - *Select open*
    - The file selection definition box should appear
  - *Select Spindle_start.CATPart in the …\Part Design\Data\ directory where you installed the scenarios data*



  - *Click Open to confirm the selection*



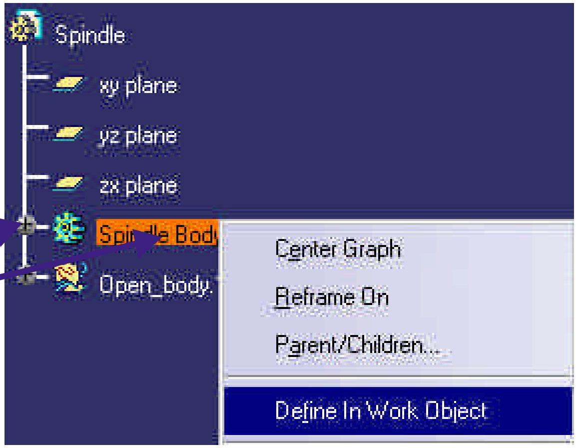
  - *Answer Yes to the question because some libraries are not installed in this demonstration mode*
  - *Maximise the window*

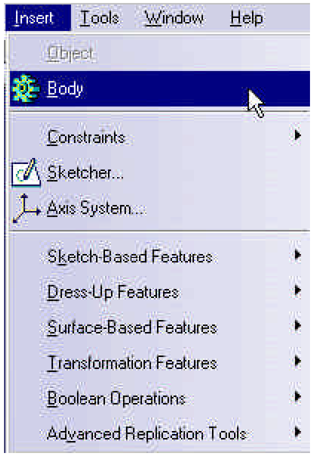


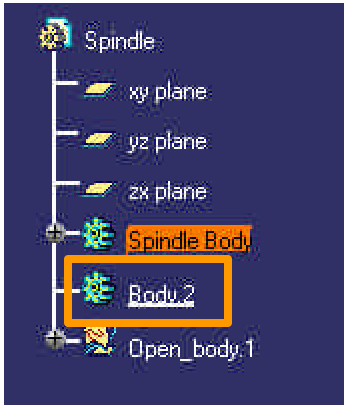
Step 1: Add a new Part Body

✍ You are going to add a new Part Body to allow Boolean operations

- ✍ *Reduce the Spindle Body tree by clicking on the (+) sign*
- ✍ *Right click on **Spindle Body***
- ✍ *Select **Define In Work Object***
  - ✍The Spindle Body should now be underlined in the specification tree to indicate that it is the "In Work Object"
- ✍ *Click on **Insert** on the menu bar*
- ✍ *Select **Body***
  - ✍**Body.2** appears in the specification tree and is now the "in work" body



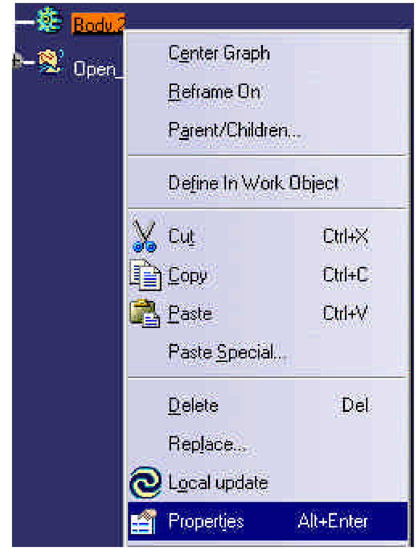


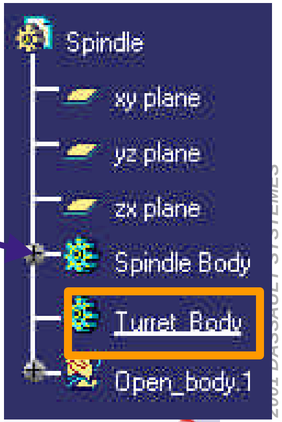


Step 1: Rename the new Part Body

- You will learn how to rename a Part Body

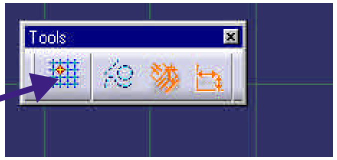
  - *Right click on **Body.2** to display the contextual menu*
  - *Select **Properties***
    - The Properties dialog box will appear
  - *Select **Feature Properties** tab*
    - Change the name from **Body.2** to **Turret Body**
  - *Click **OK** to confirm the change*
    - Turret Body will now be visible in the specification tree
    - You can compress the tree by clicking on the minus symbol if you've not already done so

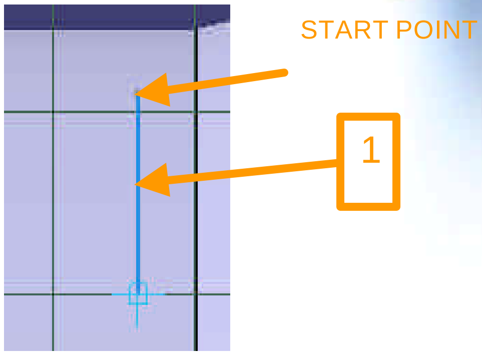




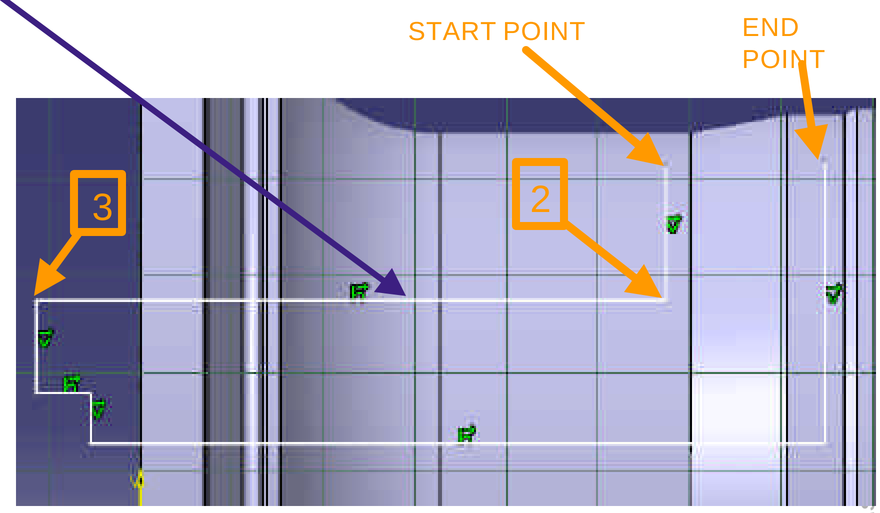
Step 2: Create a Shaft feature

- You are going to draw a sketch and use it to create a Shaft
  - *Click on **yz plane** in the specification tree*
  - *Click on the sketcher icon*
  - *Deactivate the **Snap to Point** capability by clicking on the icon (it must not be red)*





  - *Click on the **profile** icon to create the white profile as shown on the right*
  - *Single-click at the indicated START point and move cursor down creating a vertical line (1)*
    - The sketcher assistant tells you when the line is vertical with the blue colour line
  - *Single click to stop your first line (2)*
  - *Move the cursor left to continue your profile with a horizontal line*
    - The sketcher assistant tells you when the line is horizontal with the blue colour line
  - *Single click to stop your second line (3)*



If you fail to create the sketch go to page 14

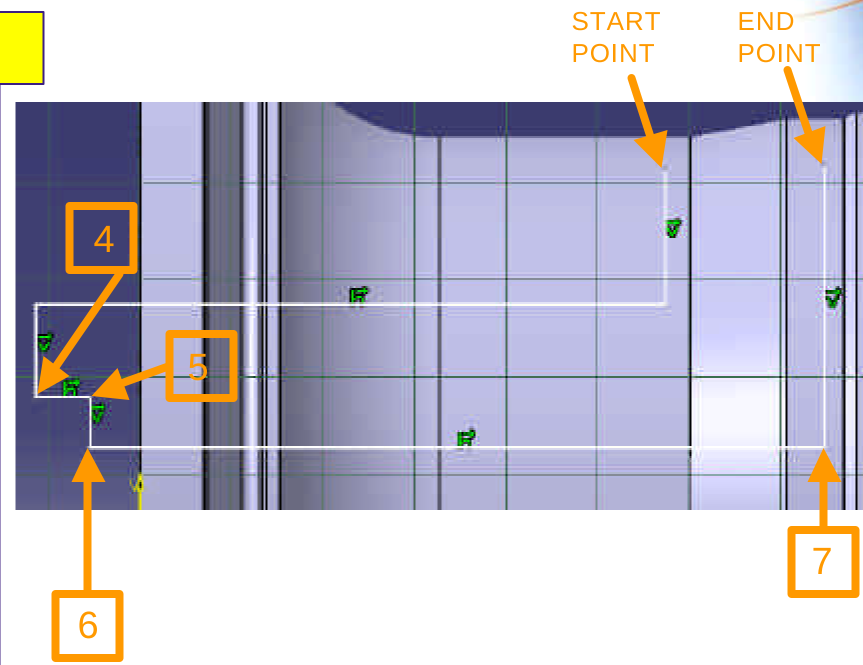
Step 2: Create a Shaft feature

- *Move the cursor down to continue your profile with a vertical line*
  - The sketcher assistant tells you when the line is vertical with the blue colour line
- *Single click to stop your third line (4)*

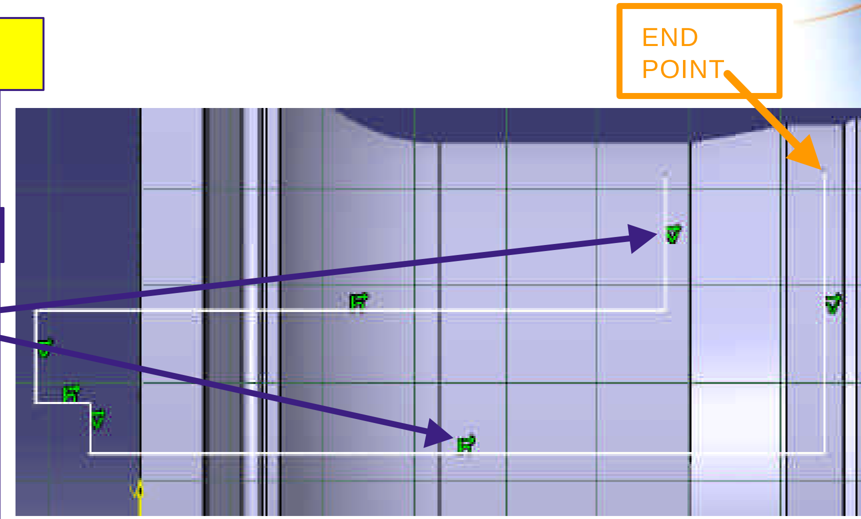
- *Move the cursor right to continue your profile with a horizontal line*
  - The sketcher assistant tells you when the line is horizontal with the blue colour line
- *Single click to stop your fourth line (5)*

- *Move the cursor down to continue your profile with a vertical line*
  - The sketcher assistant tells you when the line is vertical with the blue colour line
- *Single click to stop your fifth line (6)*

- *Continue the profile until the END point checking the horizontal and vertical constraints (blue lines)*



Step 2: Create a Shaft feature



✍ *Single click at the "END POINT" to finish the profile and click on the **Profile** icon again to exit the function*

✍ Be sure that all geometrical constraints shown on the right are visible on the profile

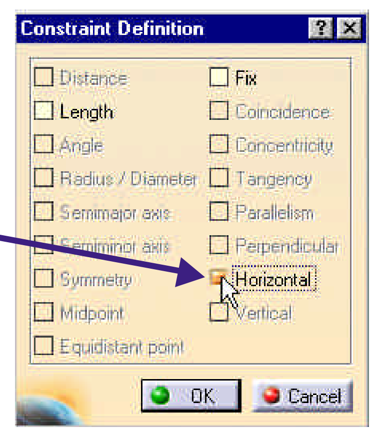
✍ If one or more geometrical constraints are missing, follow the following steps to add the constraint(s)

✍ *Select the line*

✍ *Click on **Constraints Defined in a Dialog Box** icon*

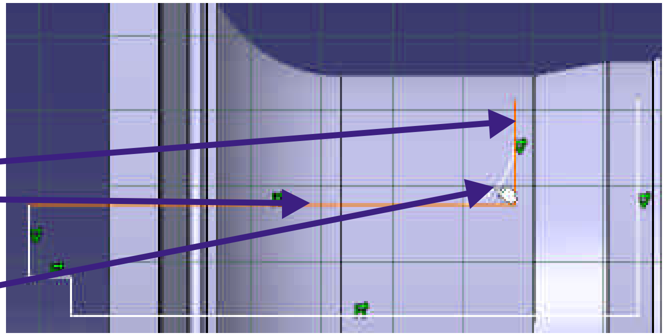
✍ *In the **Constraint Definition** window, click on Vertical or Horizontal box option*

✍ *Click **OK** to confirm selection*

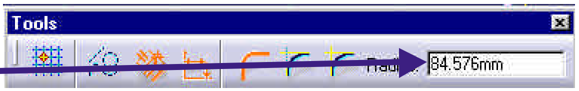


✍ Repeat the same procedure for any other missing geometrical constraint(s)

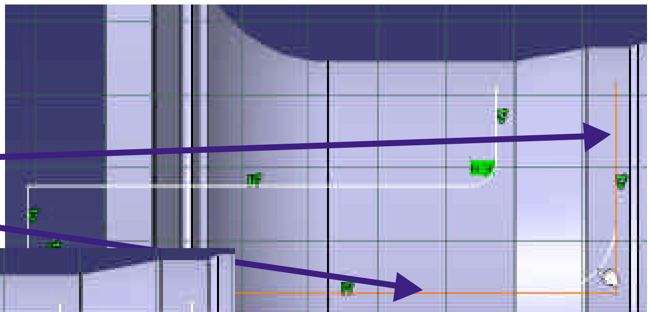
Step 2: Create a Shaft feature



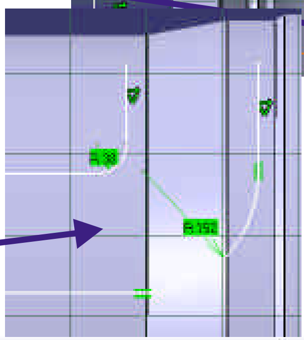
- Click on the **Corner** icon
- Select the two indicated lines
  - An arc and and its radius are displayed
    - Take care to have the pre-visualisation arc at the correct location (as shown) before pressing "Enter"



- Enter "38" and the value will be automatically placed in the Radius field - then press "Enter"



- Click on the **Corner** icon
- Select the two indicated lines
  - An arc and and its radius are displayed
- Enter 152 and the value will be automatically placed in the Radius field - then press "Enter"
    - Take care to have the pre-visualisation arc on the right location (as shown) before pressing "Enter"



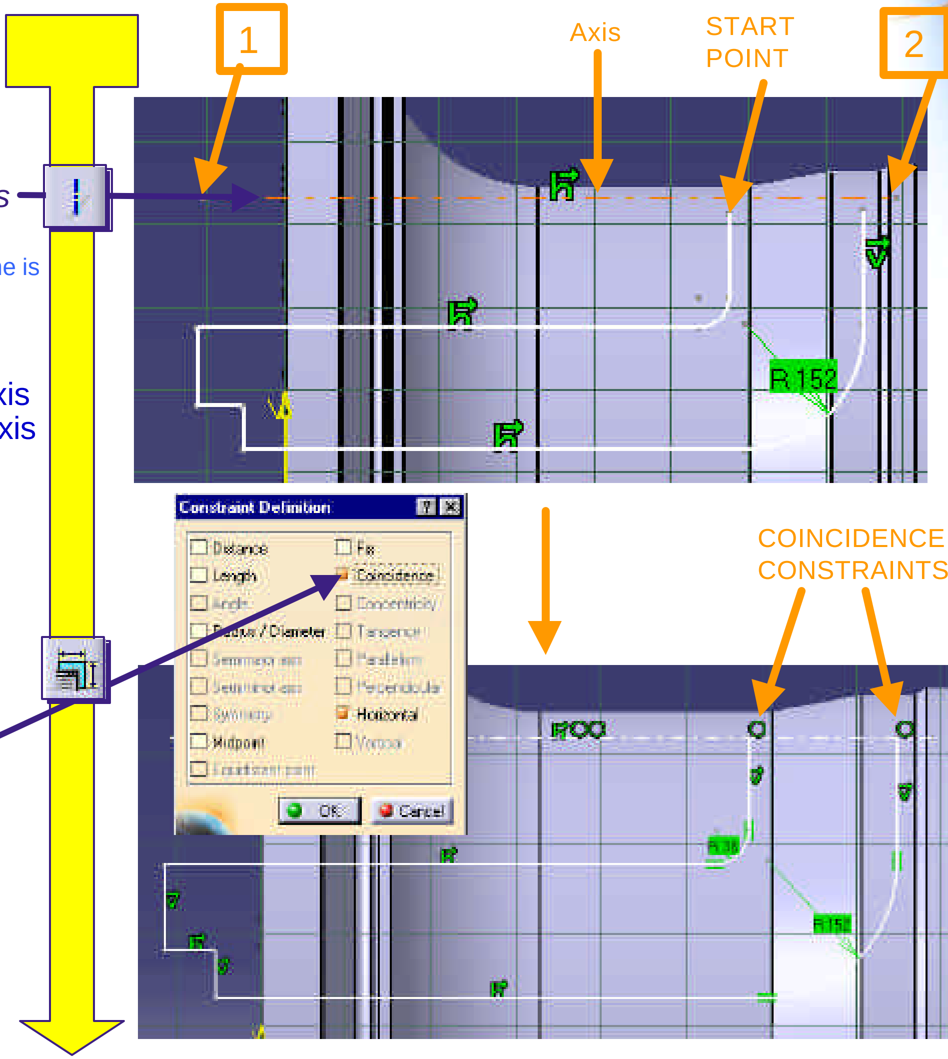
  - You should have this result

Step 2: Create a Shaft feature

- ✍ Create an axis
  - ✍ *Click on the **axis** icon to add an axis as shown on the right*
    - ✍ Click in the “1” area. Make sure the line is blue
    - ✍ Click in the “2” area

- ✍ Add a coincidence constraint between the axis and START and END points of the sketched axis line
  - ✍ *Click on the axis*
    - ✍ Press <CTRL> key to multi-select elements
  - ✍ *Click on the START point*
  - ✍ *Click on **Constraints Defined in a Dialog Box** icon*
    - ✍ The constraint definition box should appear
  - ✍ *Check **Coincidence** box*
  - ✍ *Click **OK** to confirm selection*

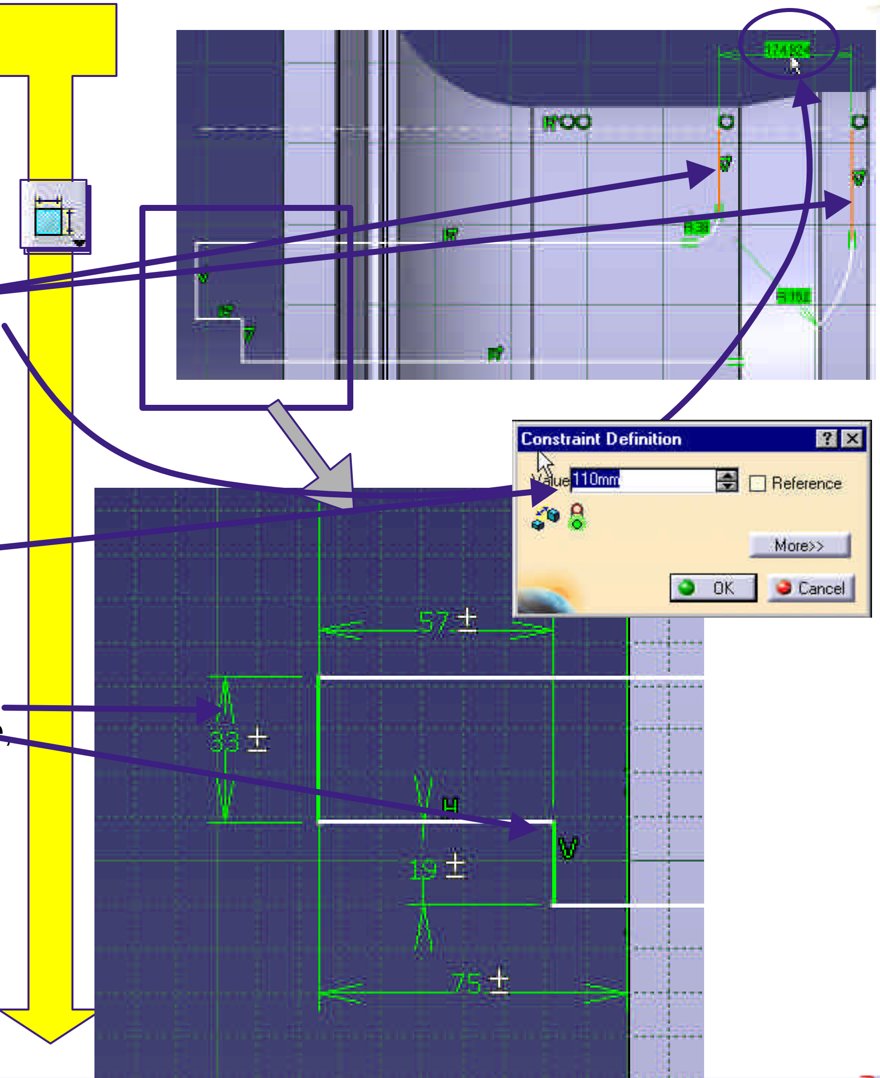
- ✍ Repeat the same procedure for the axis and the END point of the sketched axis line



Step 2: Create a Shaft feature

- You will add other dimensional constraints to the sketched profile as shown on the right
  - *Click on the **Constraint** icon*
  - *Select the two indicated lines*
  - *Click on the area where you want to place the dimension*
  - *Double click on this dimension to edit it*
  - *Enter "110" in the **Value** field and press <Enter> key*

  - *Repeat these operations to create the result as shown in the picture*
    - Remember: to keep an icon active, you can double-click on it instead of clicking…
    - To create the "75" dimension you need to select an edge from the 3D geometry

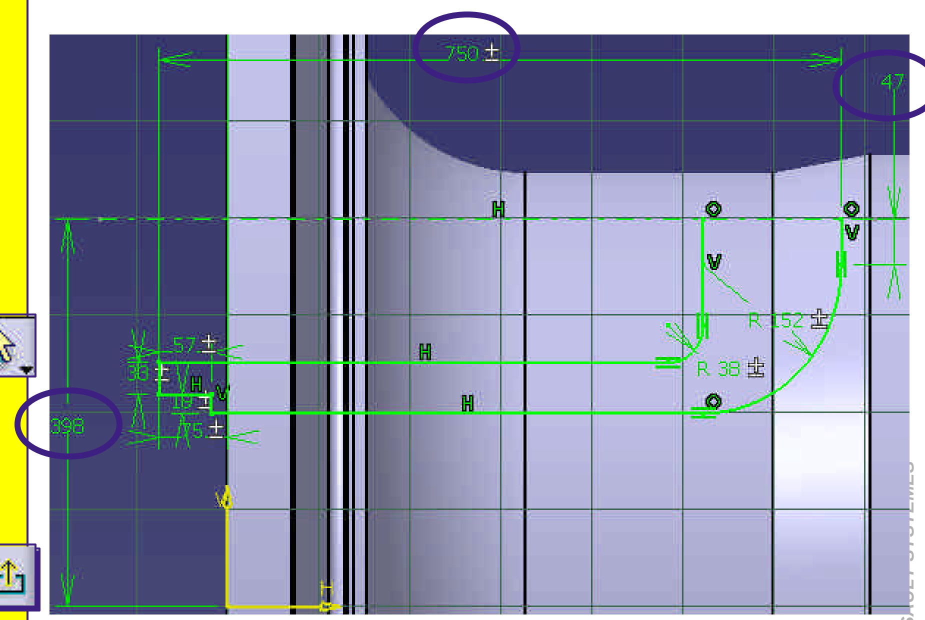


Step 2: Create a Shaft feature

✍ Modify dimensional constraints to match the illustration as shown on the right

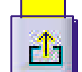
✍ *Repeat these operations to create the result as shown in the picture*

✍ When you create the "47" dimension select the vertical line. Click on MB3 and select the Position Dimension option to place the dimension – this will prevent you from accidentally selecting the geometry and creating a wrong dimension.



✍ If you accidentally create a wrong dimension, you can restart by clicking on the **Select** icon

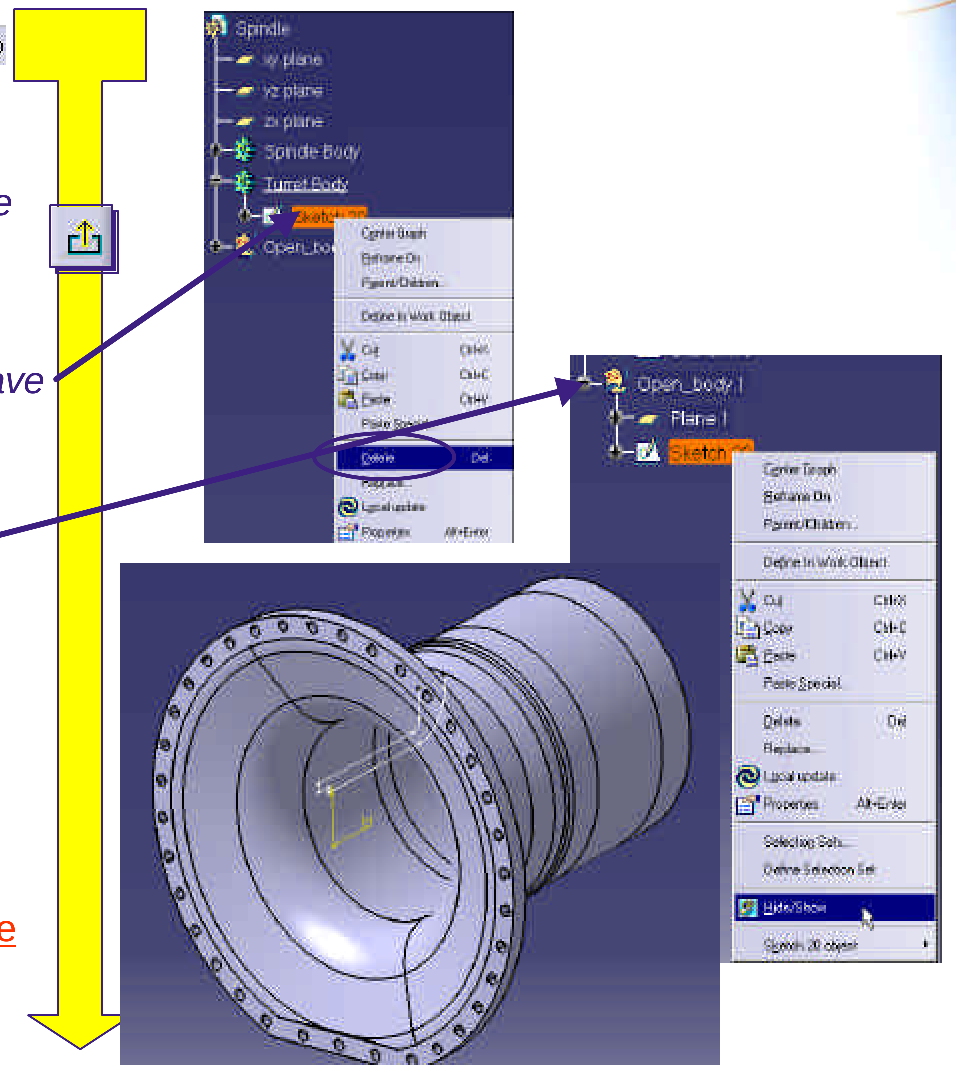
✍ *Click the **Exit Workbench** icon*



Step 2: Create a Shaft feature

✍ If you fail to create the sketch:

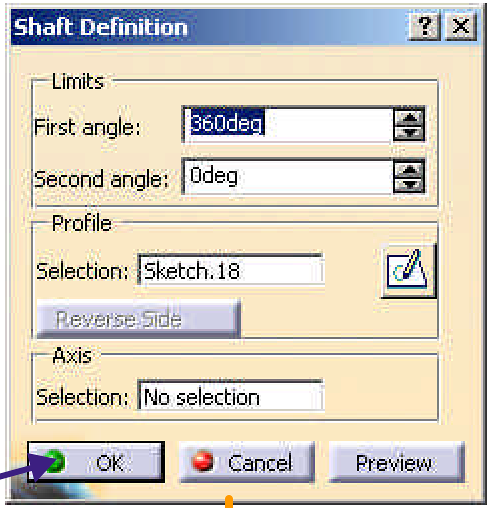
- ✍ *If you have not already exited the Sketcher, click on the **Exit workbench** icon*
- ✍ *Delete the **Sketch.20** you just have created with MB3 on **Sketch.20 + Delete***
- ✍ *Expand the Open_Body.1 by clicking on the "+" symbol*
- ✍ *With MB3 on **Sketch.20** select **Hide/Show***

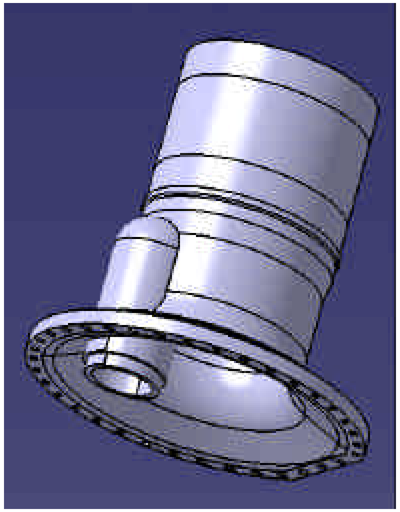


✍ If you have succeeded in creating the sketch, continue with the scenario on the next page

Step 2: Create a shaft feature

- You can now create the shaft feature
  - *Click on the shaft icon*
    - The shaft definition box should appear
    - Be sure the parameters in the definition box match the parameters shown on the right. Select the sketch you have just designed if necessary.
  - *Click on **Preview** to preview shaft feature*
  - *Click **OK** to confirm shaft feature*
    - The shaft feature is added to the specification tree

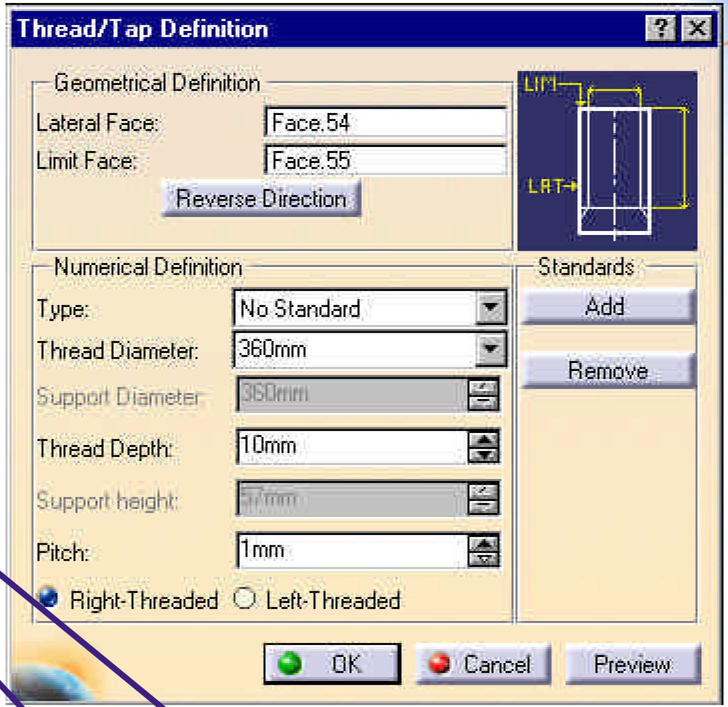


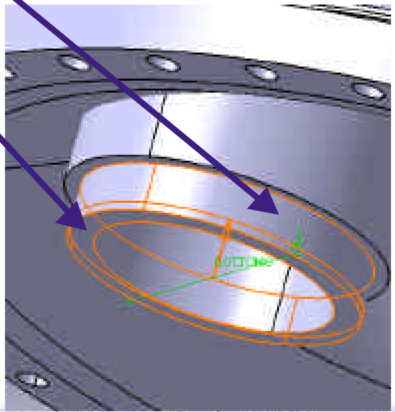


Step 3: Create a Tap

- We will add a tap at the end of the shaft
  - *Click on the thread / tap icon*

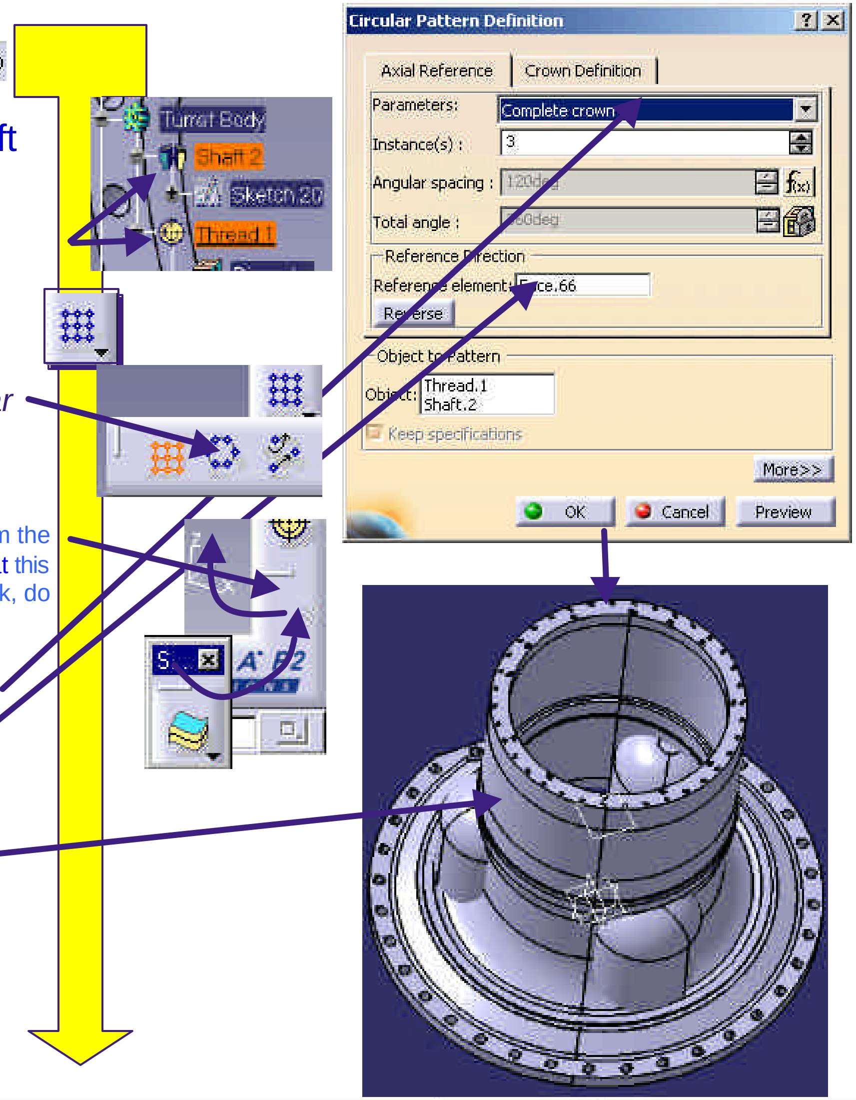
    - The thread / tap definition box should appear
  - *Click on the cylinder surface*
  - *Click on the limit face*
    - Be sure the parameters in the definition box match the parameters shown on the right. Select the sketch you have just designed if necessary.
  - *Click on **Preview***
    - The result is highlighted
  - *Click **OK** to confirm*





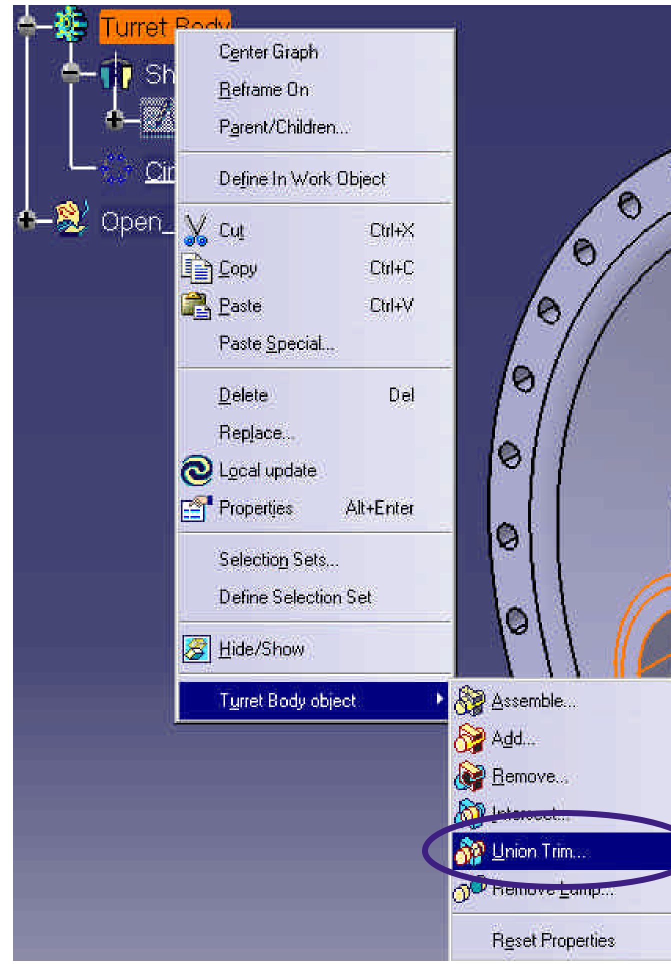
Step 4: Create a circular pattern

- You can create a multi instantiation of the shaft and of the tap using the circular pattern icon
  - *Holding the < Crtl > key, Click on Shat.2 and Thread.1 in the tree.*
  - *Click and keep MB1 on the black arrow at the bottom right of the Rectangular Pattern icon*
  - *Drag the mouse then release MB1 on the circular pattern icon*
    - If you don't see the icon, it means that the corresponding toolbar is hidden due to your display settings. To find it, drag and drop the empty area from the bottom right side to the centre of the 3D view. Repeat this operation until you find the right toolbar. To put it back, do the reverse operation
    - The circular pattern definition box should appear
  - *Select Complete crown in the **Parameters** field*
  - *Enter 3 for the **Instance(s)** field*
    - If not already done so.
  - *Click on the **Reference element** field*
  - *Select the external cylinder in the 3D view*
  - *Click **OK** to confirm circular pattern of shaft*



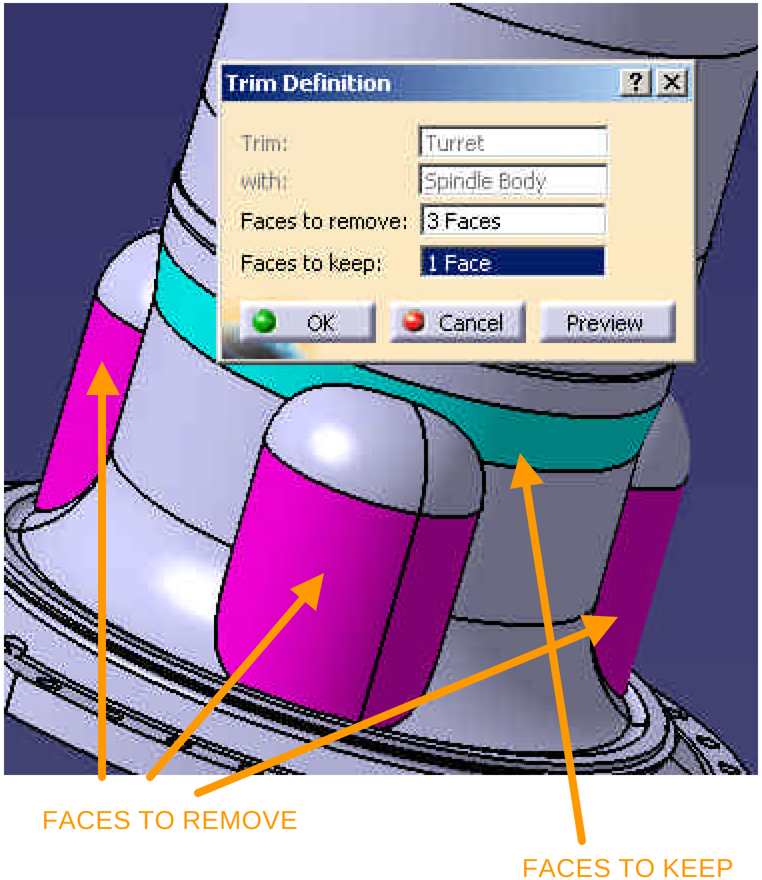
Step 5: Create a union trim

- You can use a Boolean operation to assemble and trim the two bodies
  - *Right click on Turret Body in the specification tree*
  - *Select Turret Body object*
  - *Select Union Trim…*
    - The trim definition box should appear



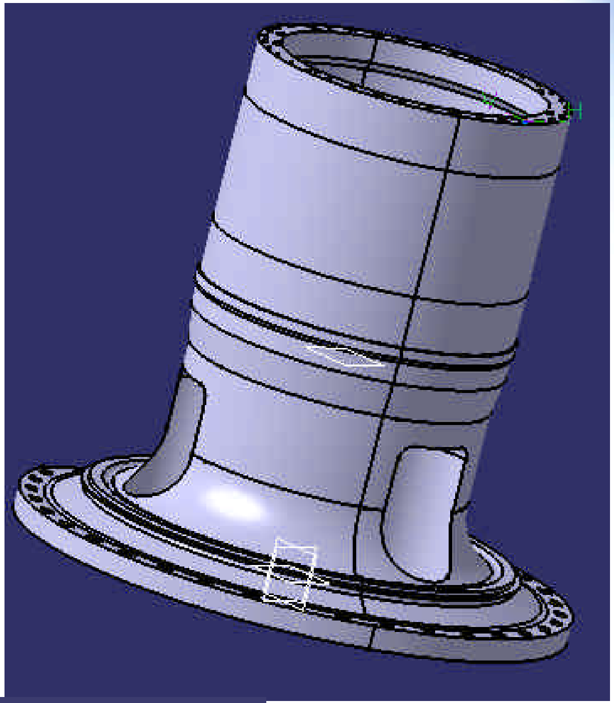
Step 5: Create a union trim

- *Click on the **Faces to remove** field*
- *Using the mouse, select the 3 faces of the shaft on the geometry as shown on the right*
  - The faces should turn pink
- *Click on the **Faces to keep** field*
- *Using the mouse, select the lower band of the Spindle Body as shown on the right*
  - The face should turn blue



Step 5: Create a union trim

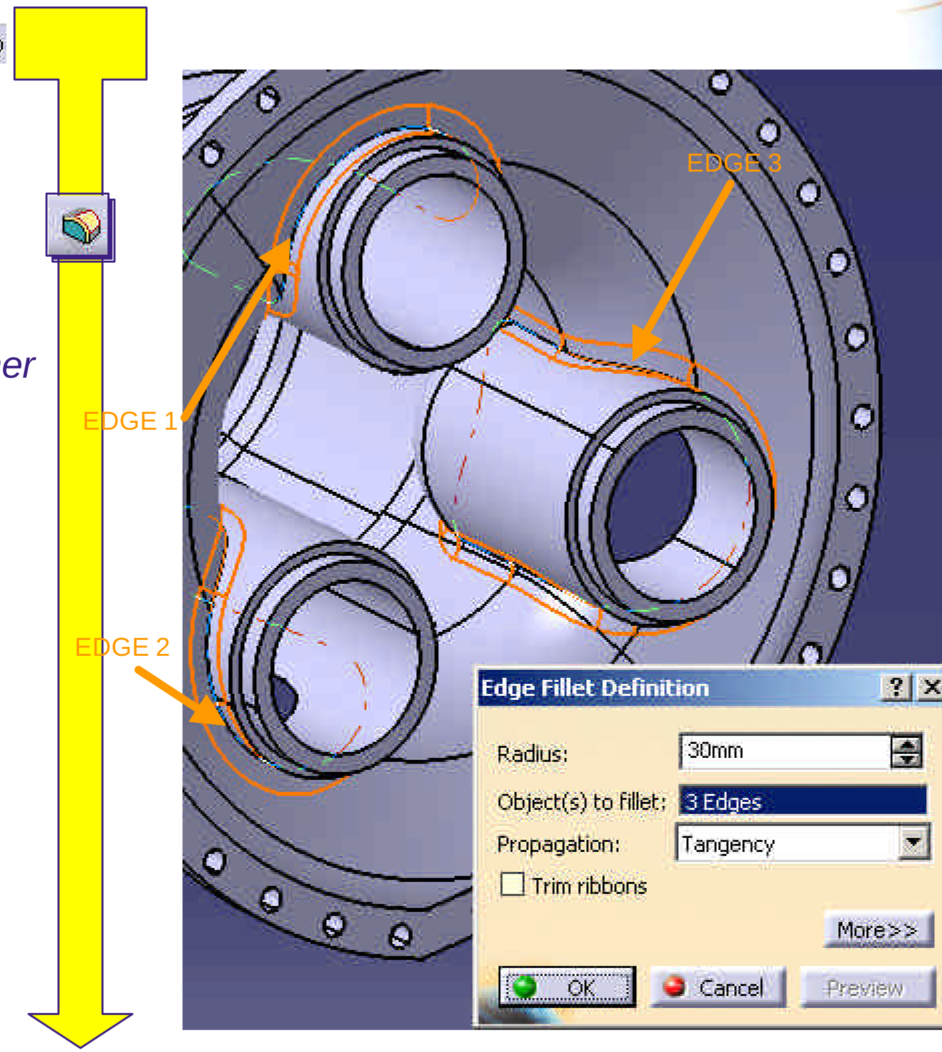
- *Click* ***Preview*** *to preview union trim*

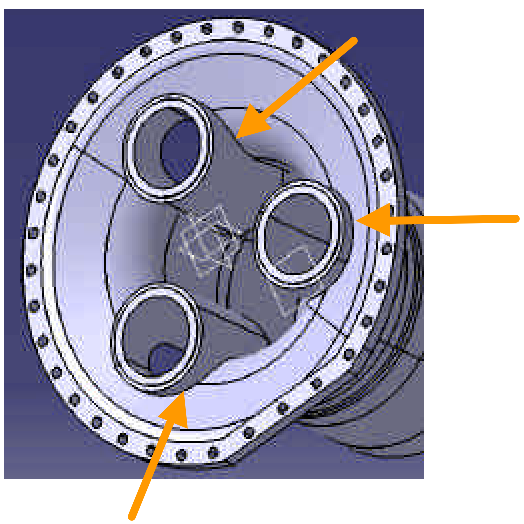


- *Click* ***OK*** *to confirm union trim*
  - Trim.1 is added to the specification tree

Step 6: Create an edge fillet

- You can now create an edge fillet
  - *Click on the edge fillet icon*
    - Key in 38mm for the radius field
  - *Click on **Object(s) to fillet** field*
  - *Using the mouse,select the 3 inner edges of the shaft as shown below*
  - *Click **OK** to confirm edge fillet feature*



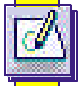


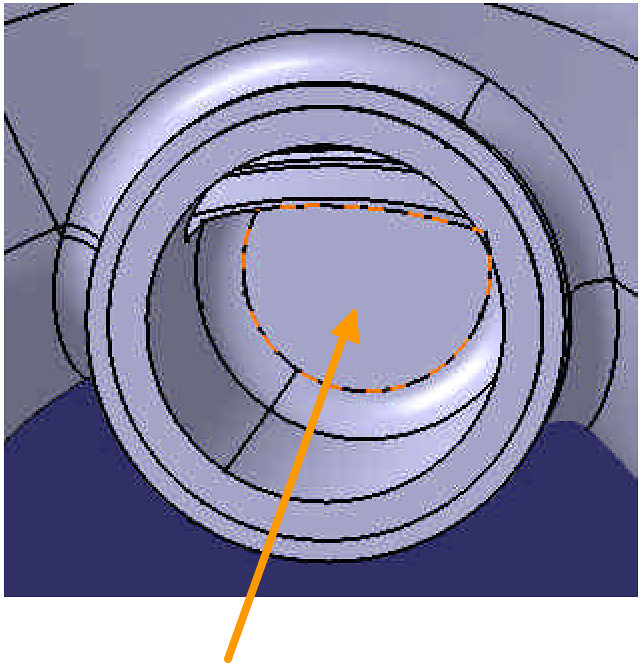
Step 7: Create a pad feature

✍ You will create a pad feature, using the profile sketched on the surface, as shown on the right

✍ *Click on one of the 3 sketch surface as shown*

✍ *Click on the **Sketcher** icon*

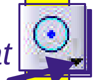


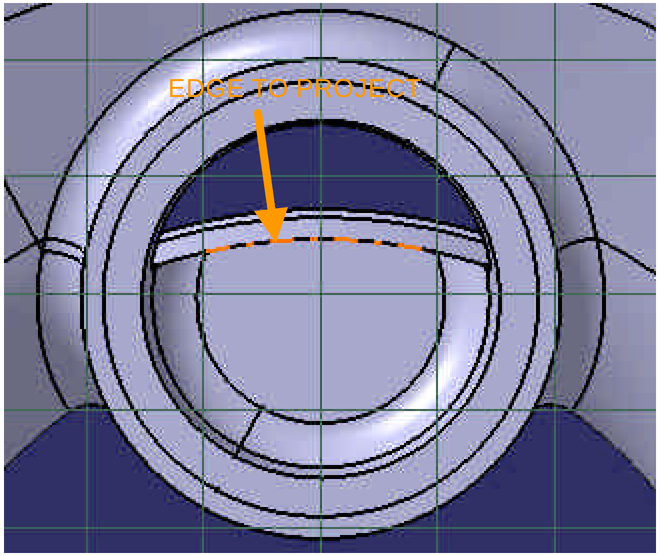


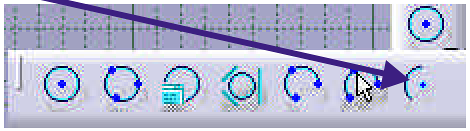
SKETCH SURFACE

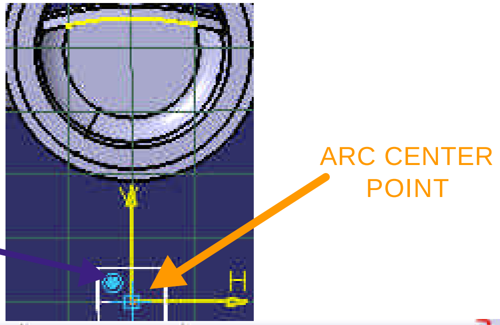
Step 7: Create a pad feature

- Sketching the pad profile
  - Click on the ***project 3D elements*** *icon*
  - *Select the edge you want to project, as shown on the right*
  - *Click anywhere off the geometry to confirm selection of edge*
    - The colour of the edge should now be yellow
  - *Click and hold MB1 on the black arrow at the bottom right of the* ***Circle*** *icon*
  - *Drag the mouse then release MB1 on the* ***Arc*** *icon*
  - *Select the origin as the centre point of the arc*
    - Make sure the Double circle appears. This means there will be a coincidence constraint between this point and the origin

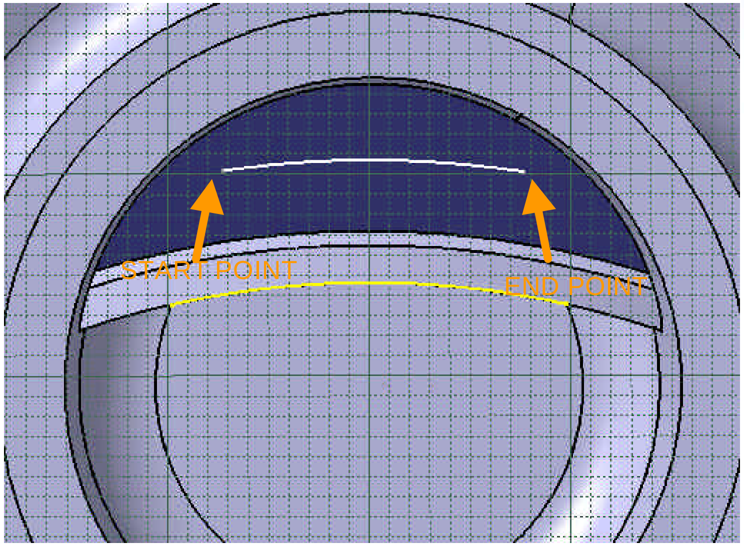









Step 7: Create a pad feature



✍ *Click on the start point and end point area of the arc as shown on the right*

✍ Connecting the start point of the arc with the endpoint of the projected edge as shown on the right

✍ *Click on the line icon*

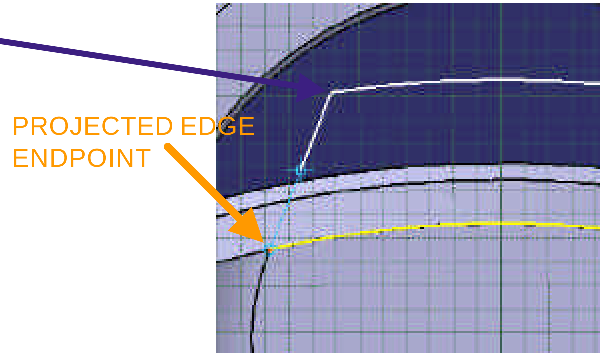
✍ *Select the start point of the arc*

✍Here again, make sure a Double circle appears

✍ *Select the endpoint of the projected edge*

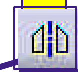
✍Here again, make sure a Double circle appears

✍ *Click anywhere off the geometry to confirm creation of the line*

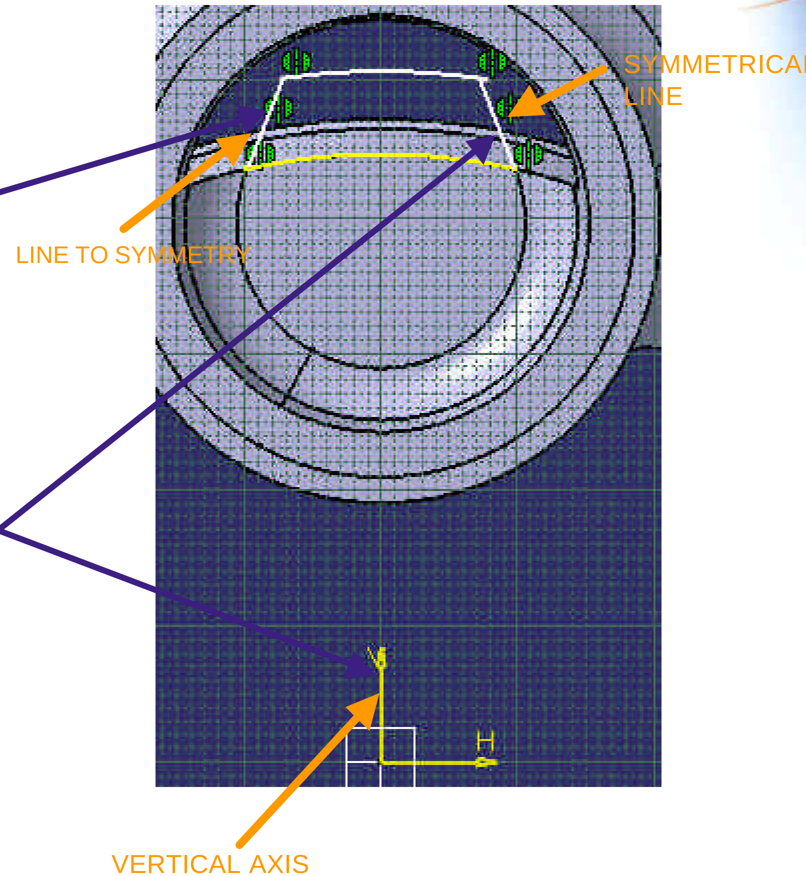


Step 7: Create a pad feature

- You will now duplicate an element using symmetry
  - *Click on the* ***Symmetry*** *icon*

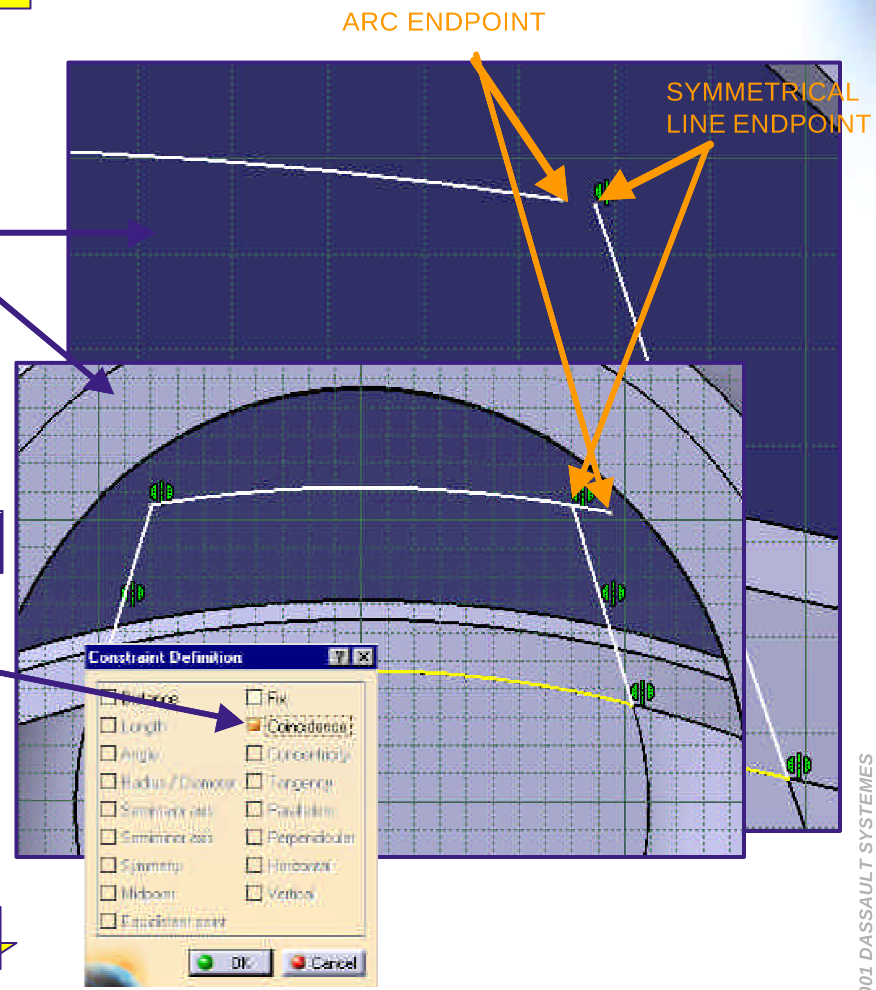


  - *Click on the line between the start point of the arc and the endpoint of the projected edge.*
  - *Click on the vertical axis. This is the line from which the element will remain equidistant*
    - A line should appear between the endpoint of the arc and the endpoint of the projected edge as shown on the right
    - The symmetry symbol should appear as shown on the right



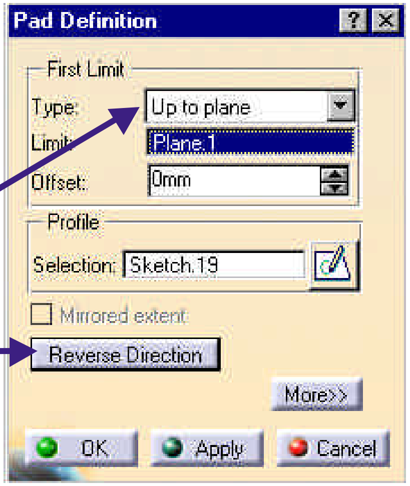
Step 7: Create a pad feature

- Adding a coincidence constraint between the endpoint of the arc and the endpoint of the symmetrical line as shown on the right. This will close the contour
  - One of the scenarios on the right should match what you have on your screen
- *Click on the endpoint of the arc*
  - Hold control key to multi-select
- *Click on endpoint of the projected edge*
- *Click on **Constraints Defined in a Dialog Box** icon*
- *Check Coincidence box*
- *Click **OK** to confirm selection*
  - Of course, you could have created the second line with the same operations used for the first one: the above operation was just to demonstrate another functionality
- *Click on the **Exit Workbench** icon*

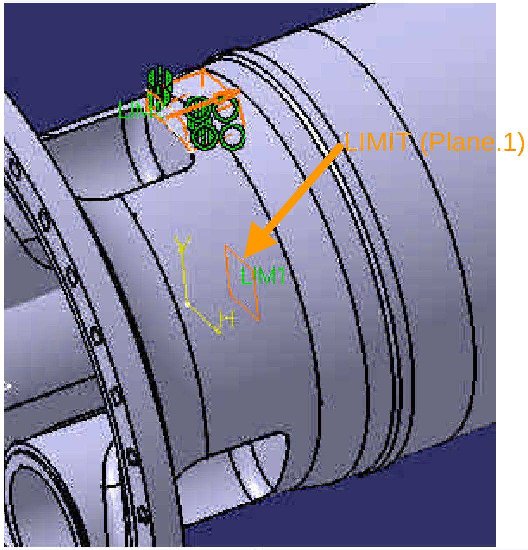


Step 7: Create a pad feature

- *Click on* ***Pad*** *icon*
  - *The pad definition box should appear*
- *Select the profile you have just created if needed*
- *Click on the* ***Reverse Direction*** *button*
- *Click on the* ***Type*** *field*
- *Select Up to plane*
- *Click on the* ***Limit*** *field*



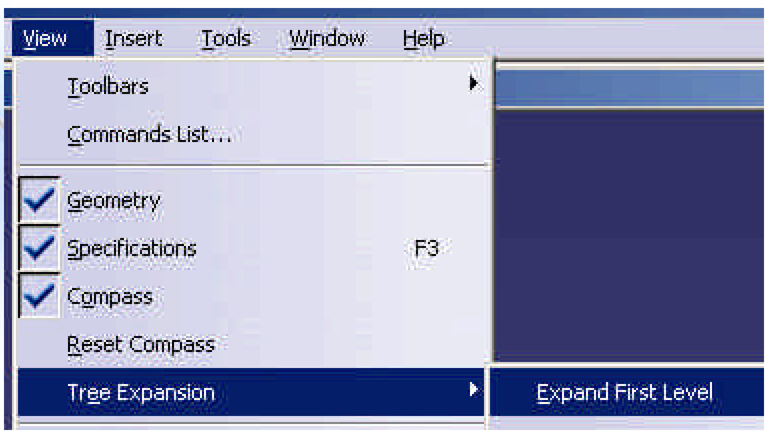
- *Using the mouse select Plane.1 in the 3D window as shown on the right*

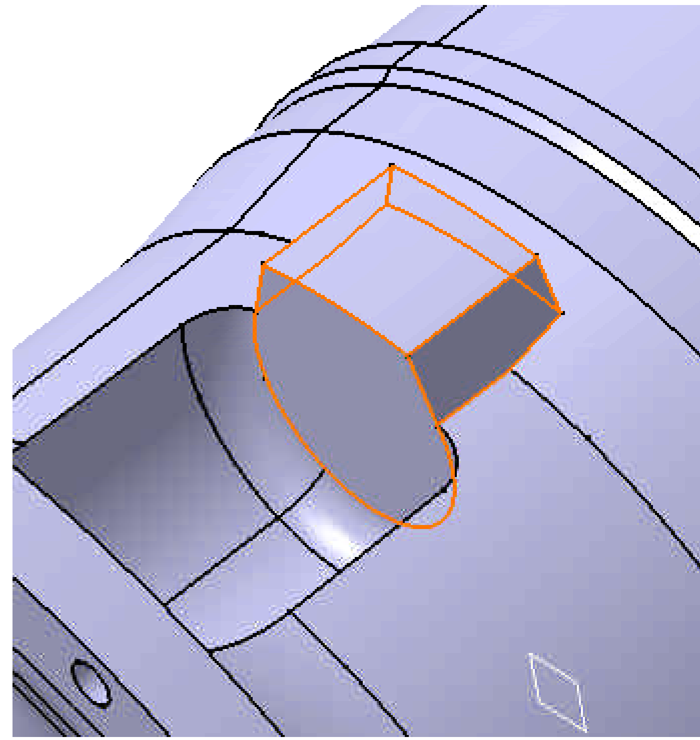


Step 7: Create a pad feature

✍ *Click **OK** to confirm pad feature*

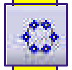
✍ *Select **View / Tree Expansion / Expand First Level***

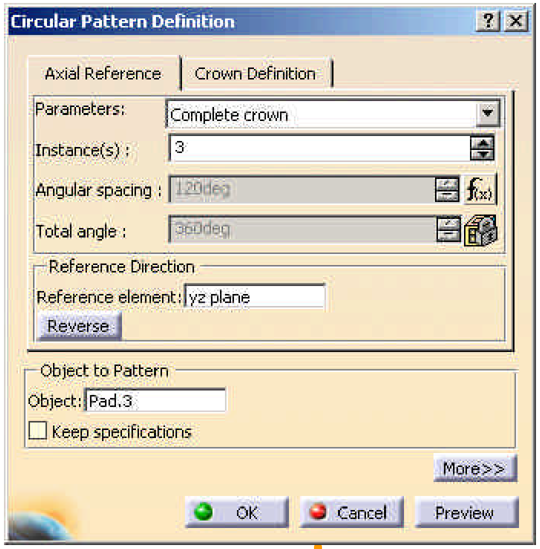


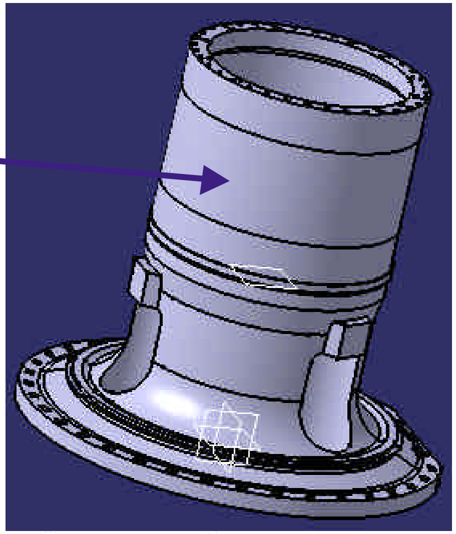


Step 7: Create a circular pattern

- Creating a multi instantiation of the pad using the circular pattern icon
  - *Click on Pad.3 in the specification tree*
  - *Click on the **Circular Pattern** icon*
    - The circular pattern definition box should appear
  - *Select Complete crown in the **Parameters** field*
  - *Enter 3 in the **Instance(s)** field*
  - *Click on the **Reference element** field*
  - *Click on the external cylinder in the 3D view*
  - *Click on **Preview** to preview circular pattern of pad*
  - *Click **OK** to confirm circular pattern of pad*







Step 8: Create a hole feature

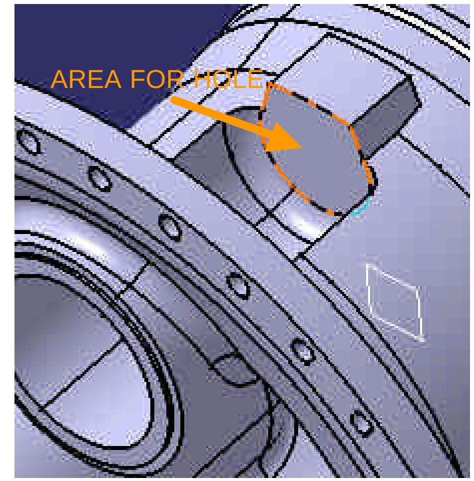
- You will use the drag and drop feature of V5 to create a hole feature

  - *Drag and drop the **Hole** icon*

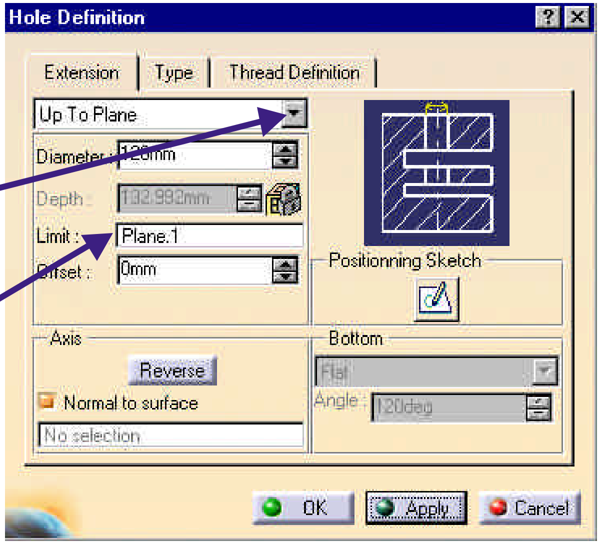
    

    - Make sure you keep MB1 pressed when using drag and drop feature

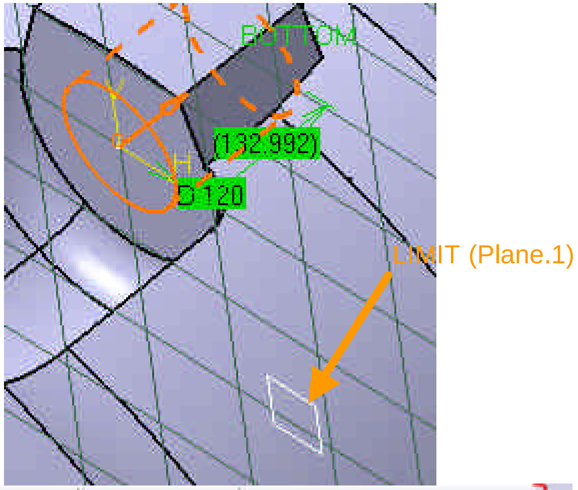
  - *Release the icon on the surface where you want to create the hole, as shown on the right*



Step 8: Create a hole feature

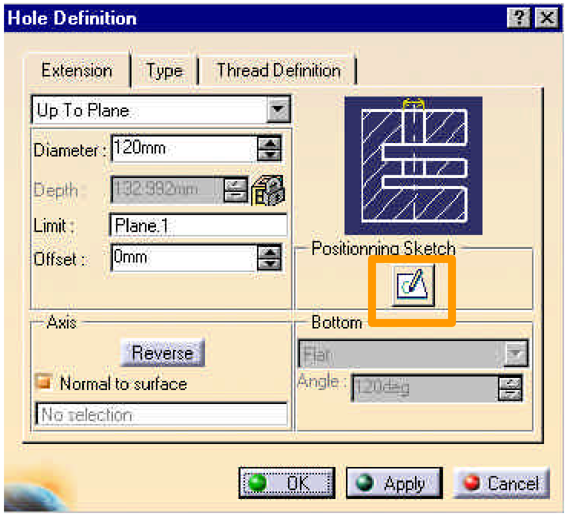


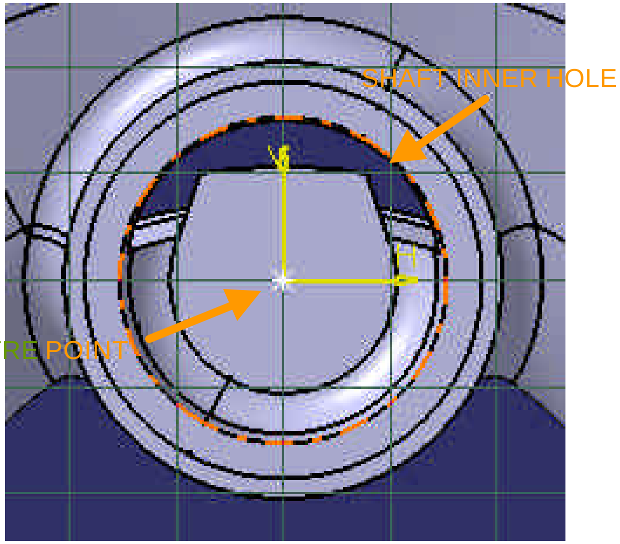
- Changing the extension type of the hole from Blind to Up To Plane
  - *Click on pull down arrow and change setting from Blind to Up To Plane*
  - *Click on the **Diameter** field*
    - Key in 120 mm
  - *Click on **Limit** field*
  - *Select Plane.1 in the 3D window as shown on the right*



Step 8: Create a hole feature

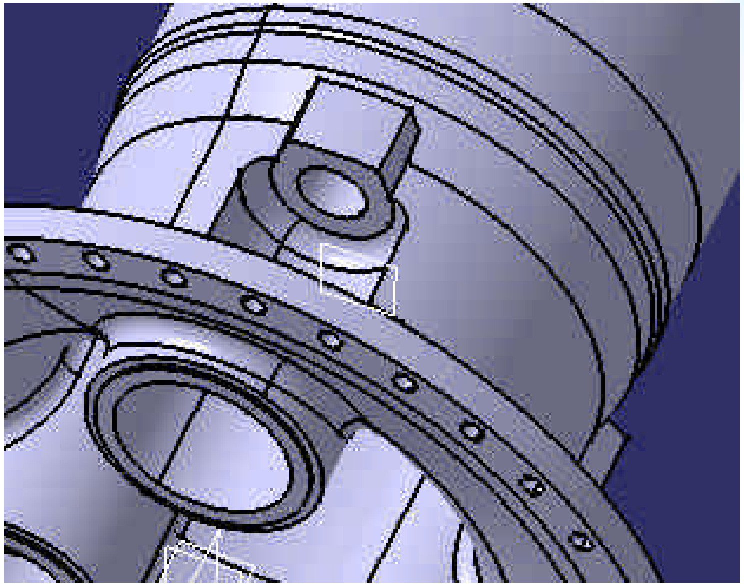
- You will now add a concentricity constraint between the hole and the inner hole of the shaft
  - *Click on the **Sketcher** icon in the **Hole Definition** box*
    - This automatically takes you into the sketcher workbench
  - *Click on centre point of the hole as shown on the right*
    - Hold control key to multi-select
  - *Click on inner hole of the shaft as shown on the right*
  - *Click on **Constraints Defined in a Dialog Box** icon*
  - *Check Concentricity box*
  - *Click **OK** to confirm selection*





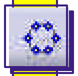
Step 8: Create a hole feature

- *Click on the **Exit** icon*
- *Click on **Preview** to preview hole feature*
- *Click **OK** to confirm hole feature*

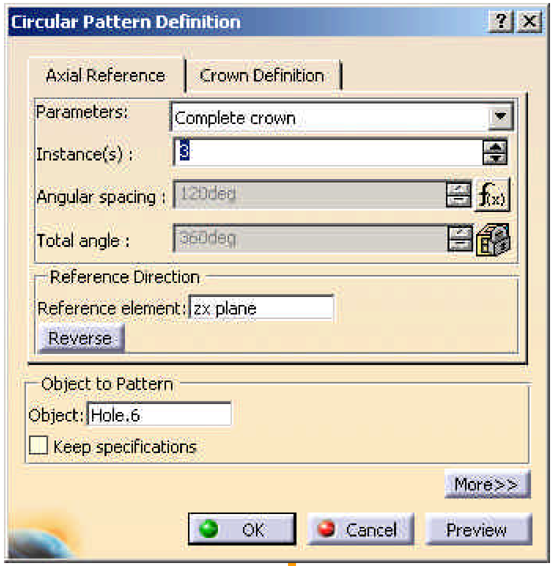


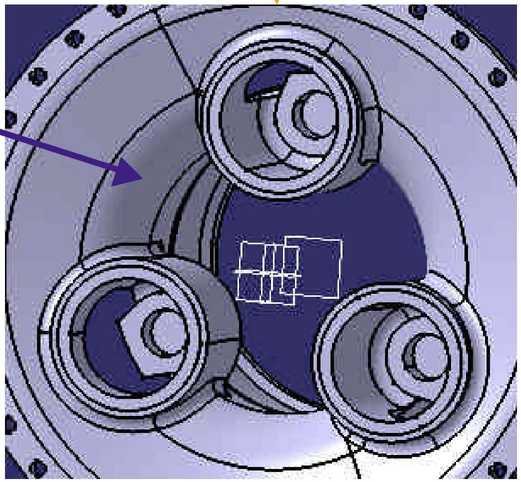
Step 8: Create a circular pattern

- Creating a circular pattern of the hole using the circular pattern icon
  - *Click on the **Circular Pattern** icon*



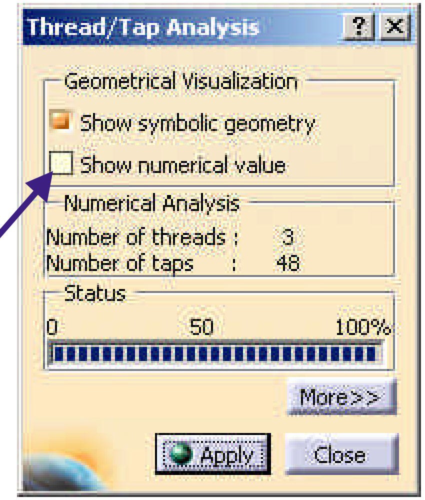
  - *Click on Hole.6 in the specification tree*
    - The circular pattern definition box should appear
  - *Select Complete crown in the **Parameters** field*
  - *Enter 3 in the **Instance(s)** field*
  - *Click on the **Reference element** field*
  - *Click on the external cylinder in the 3D view*
  - *Click on **Preview** to preview circular pattern of hole*
  - *Click **OK** to confirm circular pattern of hole*

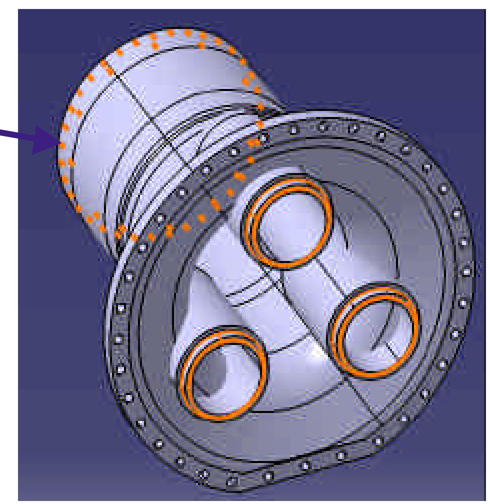




Step 9 : Thread / Tap Analysis

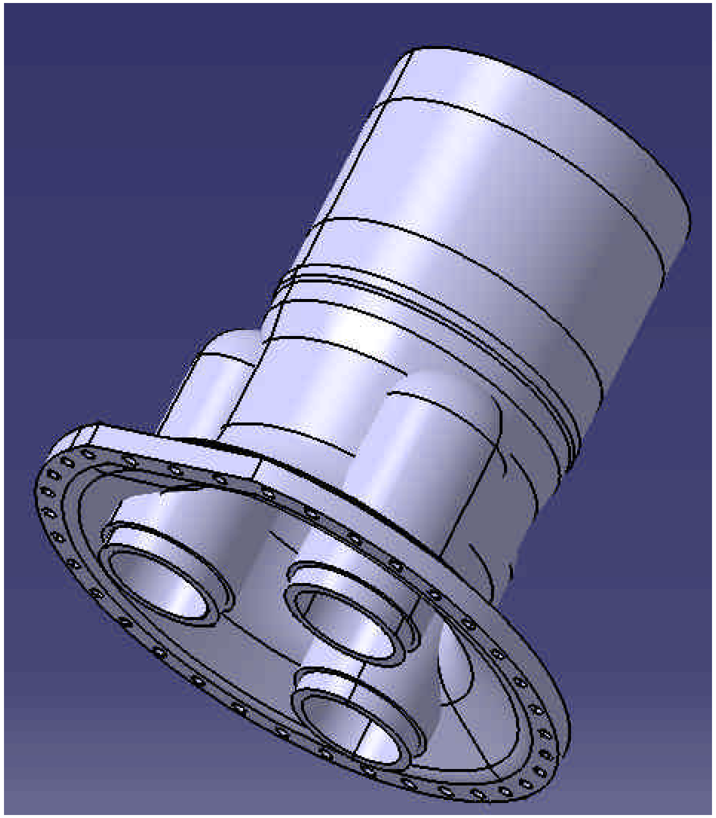
- Check every tap / thread realised in your part
  - *Click on the Tap – Thread Analysis icon*
    - As previously, if the icon is not visible, drag and drop the **bottom right** toolbars…
  - *Switch off Show numerical value*
    - All manufacturing is highlighted.
  - *Click on Apply to show the threads and taps.*
  - *Click on **Close** to finish*





End of scenario

- You now have the final part as shown on the right
- CONGRATULATIONS

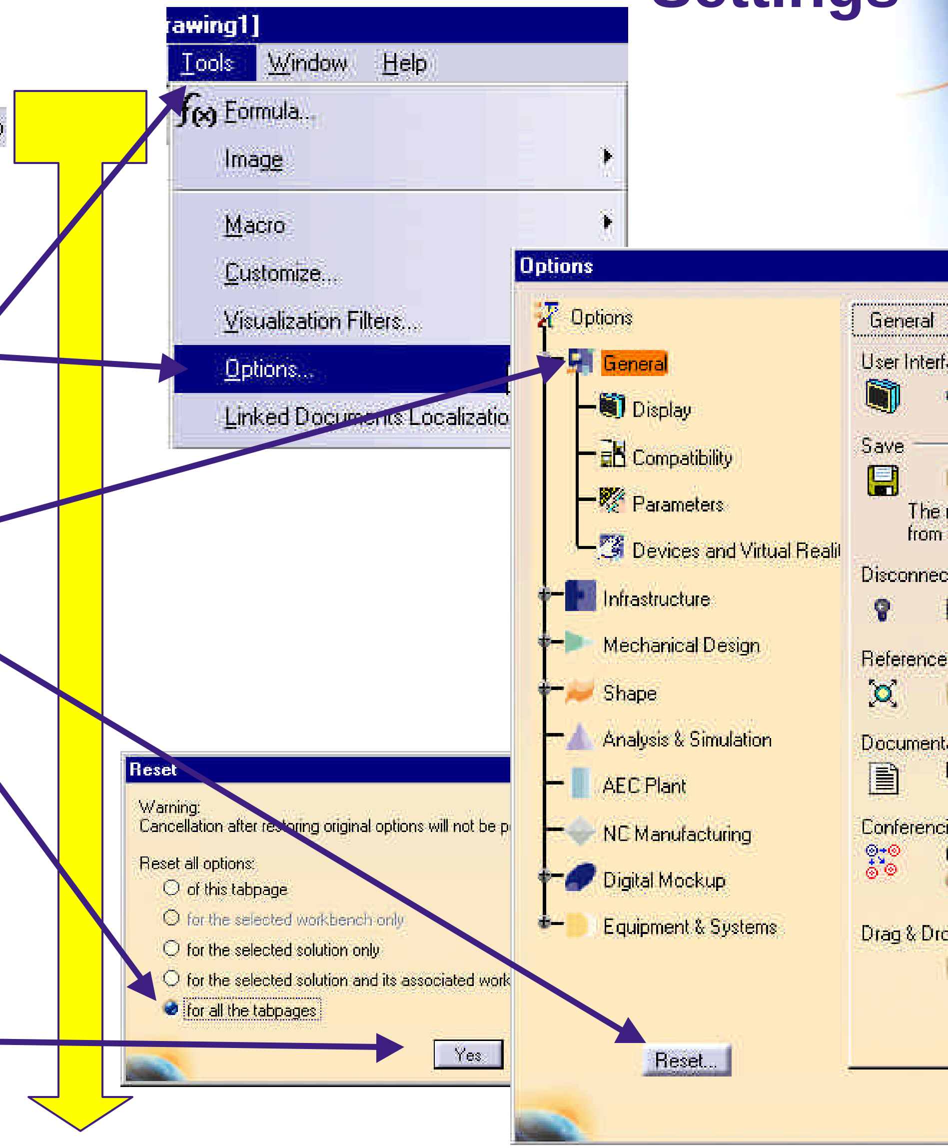


Manual Settings

Settings

Tools/Options

- We will configure the environment of CATIA
  - *Select Tools + Options menu*
- We will erase any previous configurations.
  - *Click on General*
  - *Click on the Reset Bottom*
  - *Select for all the tabpages*
  - *Click YES*

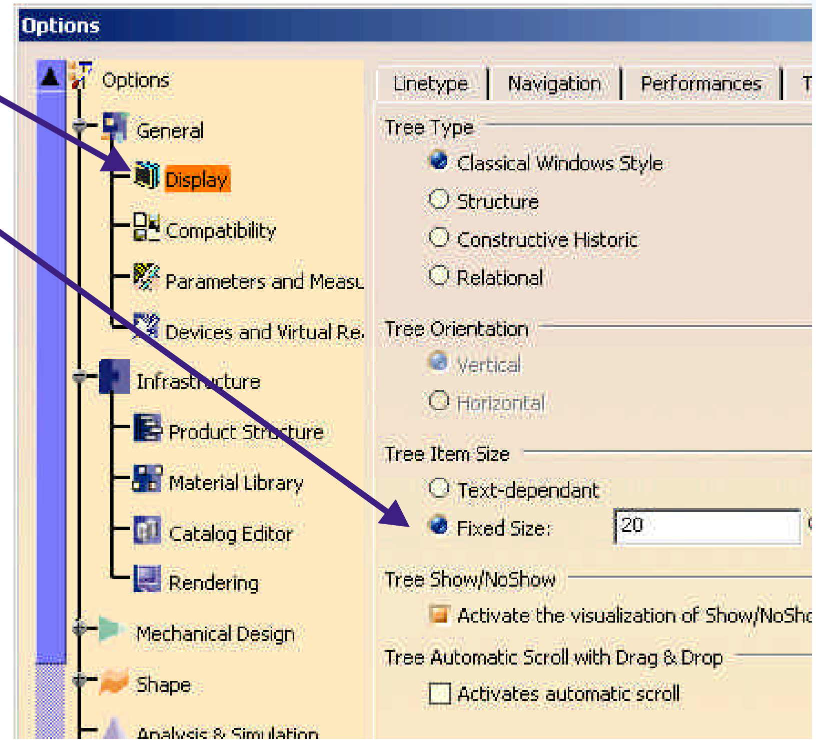


Settings

Tools/Options

- Under **General** select **Display** on the tree

  - *Check **Fixed Size** and enter 20 as value in the field.*



Congratulations