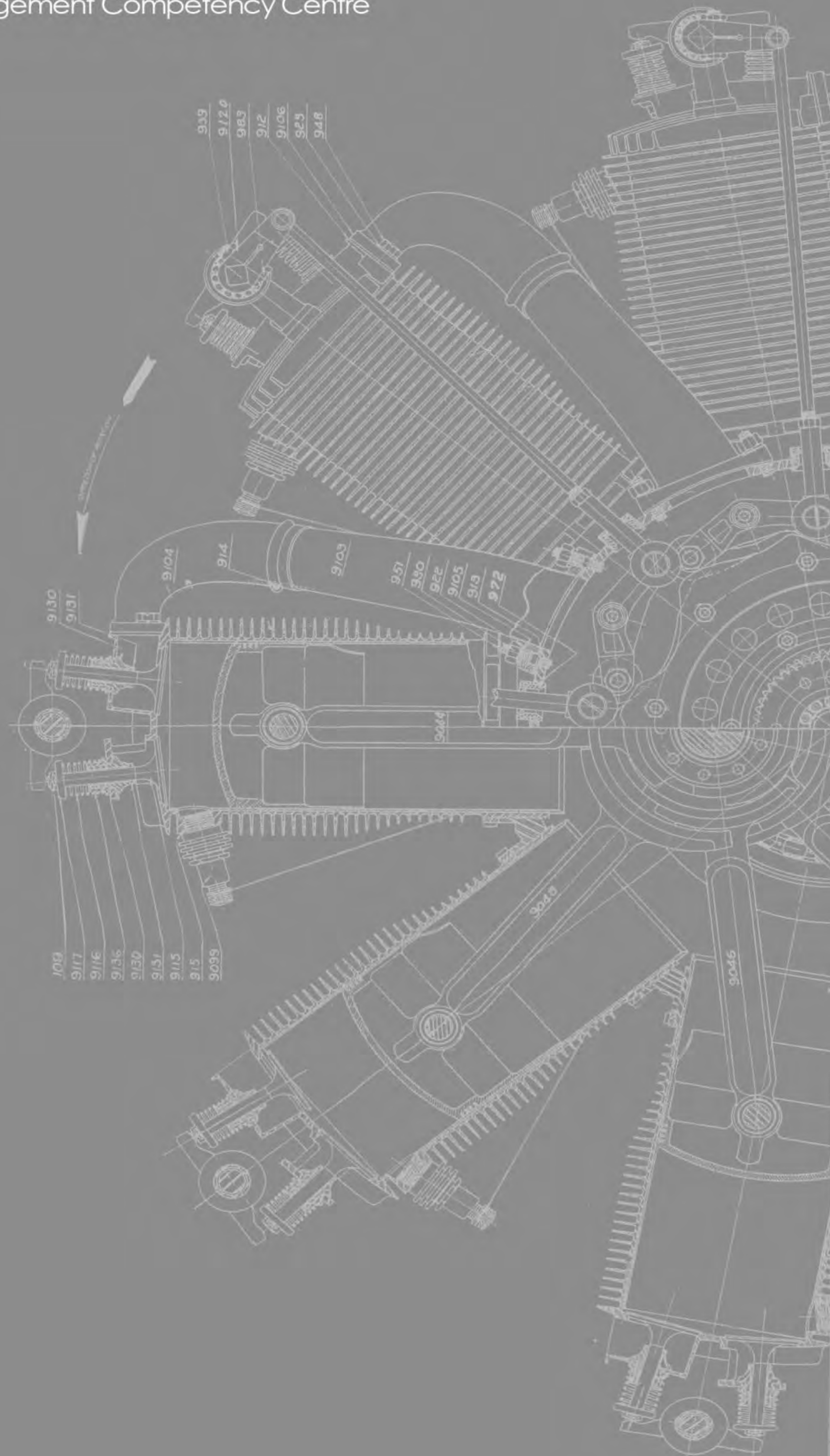
PLMCC

Product Lifecycle Management Competency Centre



Catia V6 Introduction

Buggy Case Study

DASSAULT SYSTEMES

Cape Peninsula University of Technology

Liberté • Égalité • Fraternité
RÉPUBLIQUE FRANÇAISE

French ministry of education and research

Course Objectives

This course will teach you how to design parts using feature-based and functional modeling techniques. You will also learn how to create a simple assembly, simulate a mechanism, generate a simple detail drawing and define a manufacturing process.

Upon completion of this course you will be able to:

- Connect to V6 database
- Manage PLM data
- Design parts
- Create a mechanism
- Create kinematics simulation
- Define a Manufacturing process

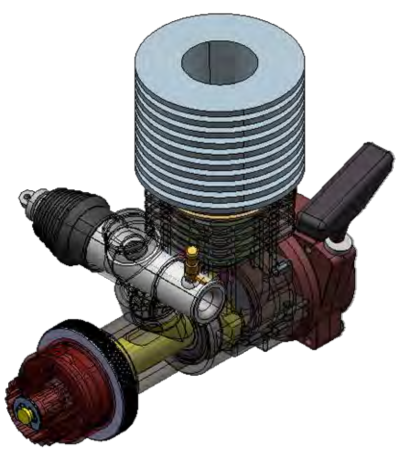
This course is designed on a process-based approach. Rather than focusing on individual features and functions, this course emphasizes both design and manufacturing processes of the product.

▷ With PLM 2.0 solutions such as V6, virtual products and systems behave as they would in the physical world, allowing all actors to have immersive, lifelike experiences in 3D and to encourage innovation while ensuring respect of the client's expectations.

▷ The V6 allows you to design with CATIA V6, simulate with SIMULIA V6, manufacture with DELMIA V6, experiment with 3DVIA V6 and also collaborate at all levels of the extended enterprise with ENOVIA V6.

▷ In the V6 architecture, all your product data are managed and contained in the database. This database contains the physical data, logical structure and requirements of your products and parts.

▷ You will be able to work on them without having to copy or store them on your local computer. This simplifies the process of data management and sharing of information.



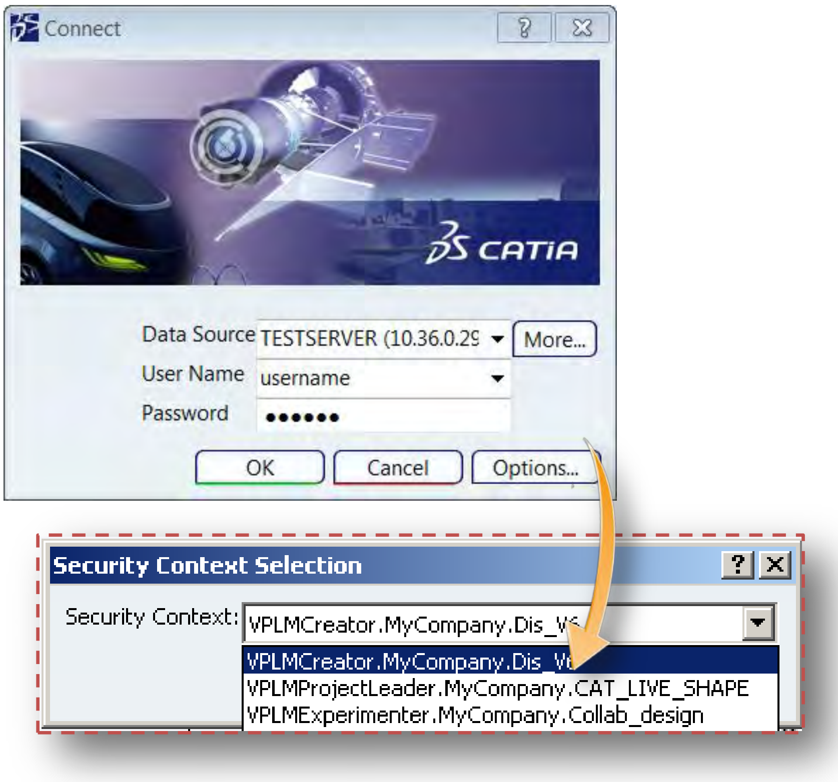
The Micro Engine.

During this courseware we will use a micro engine of RC car as a training model .

Connecting to V6

- When you start V6, you will be asked to log on to a server (Data Source).
- You will be prompted to fill in three fields:

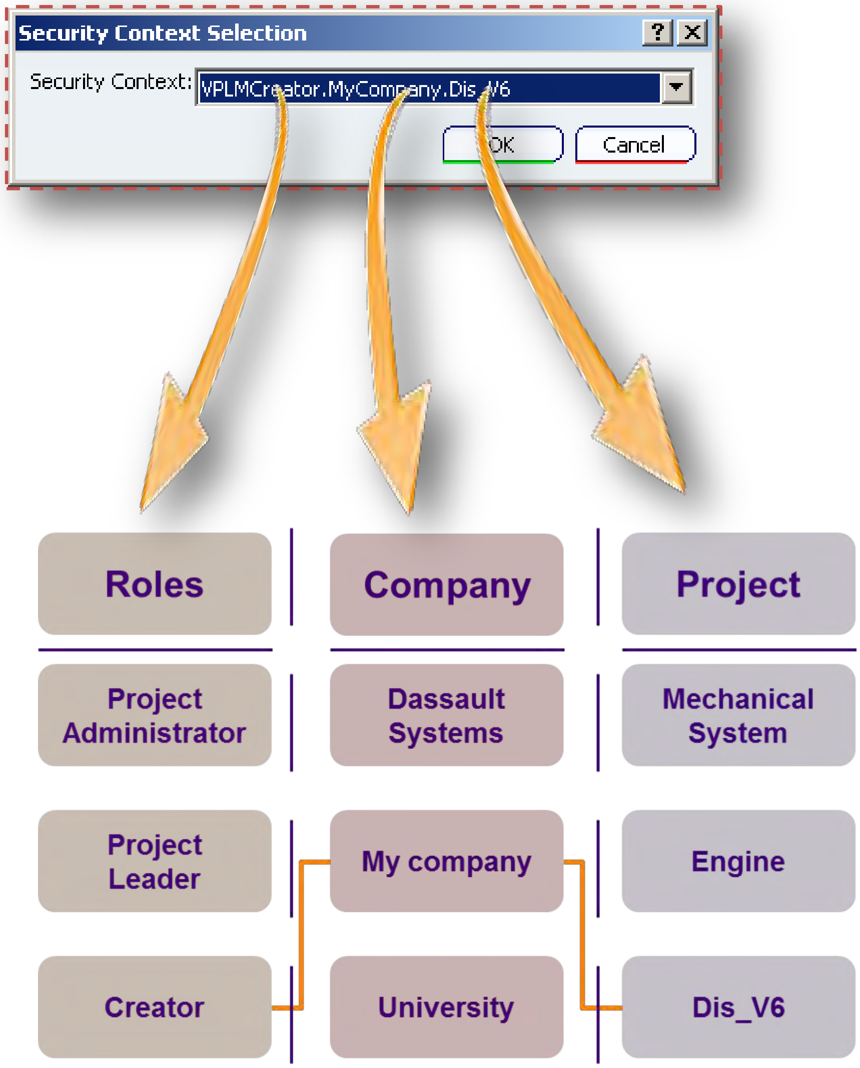
  Data Source
  User Name and Password
  Security context. (You are going to select the context in which you will work….)



Security Context and Roles
In V6, access rights to the project's data and to the features of the application are based on the role assigned to you.
The following default roles in V6 for Academia are predefined:

- Project Admin
- Project Leader
- Creator
- Experimenter
- Viewer

The combination of the role, company and project forms the **Security Context.**
You must select a designated security context in order to login and connect to the V6 database.
*In the example shown, the person who is a '**Creator**', is logging in '**My Company** ' for the '**Dis_V6**' project. He will have access rights based on this combination.*

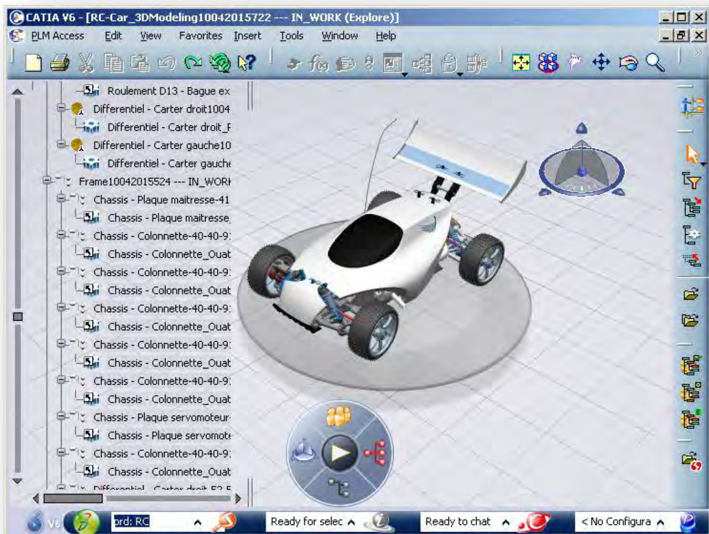


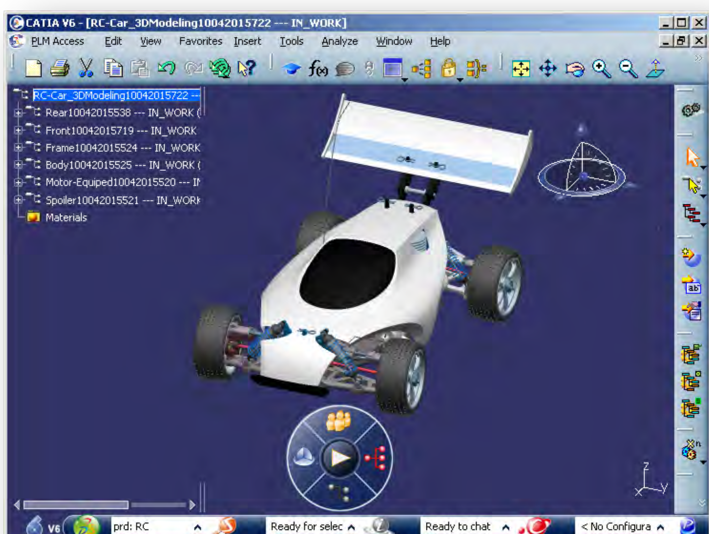
Security Context Selection
Security Context:
VPLMCreator.MyCompany.Dis_V6
OK
Cancel
Roles
Company
Project
Project Administrator
Dassault Systems
Mechanical System
Project Leader
My company
Engine
Creator
University
Dis_V6

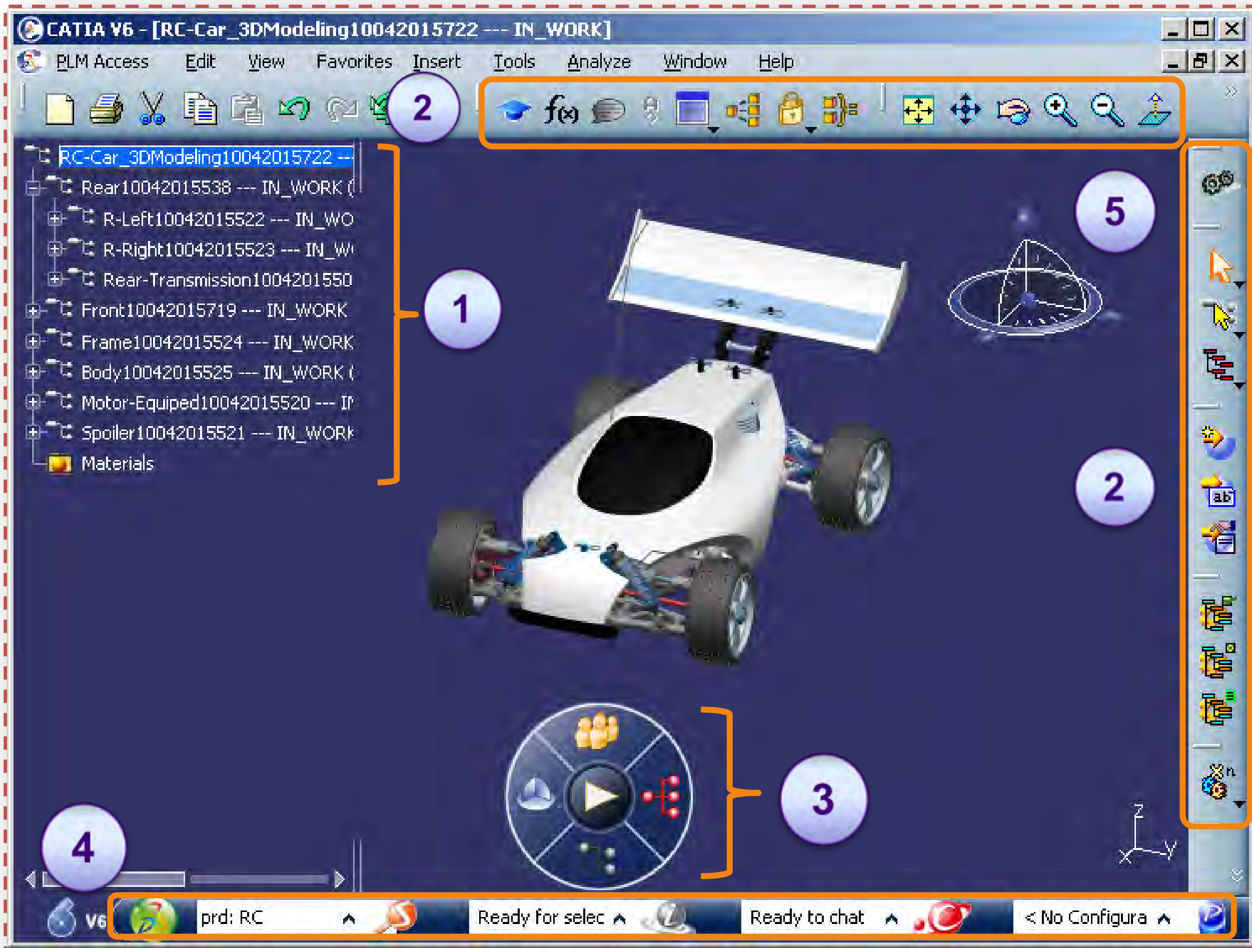
Presentation of V6 GUI
In this step you will be introduced to the V6 PLM 2.0 interface.
V6 includes two user interface windows, called Navigation and Authoring.

You can explore V6 objects in the VPM Navigator (Silver layer). You can view graphical representations of objects on the turntable, search for PLM objects, navigate product structure and view the object.

You can modify V6 objects in the Authoring window (blue layer). CATIA will allow one to edit data in 3D (design, structure, ...) and publish the changes to the database server.







1. Specification Tree
2. Toolbars
3. The compass
4. The bar
5. The Robot

The Compass provides you with instant access to PLM information at any time, on any object. It is present in all document windows and is composed of four quadrants(North, South, East, West).

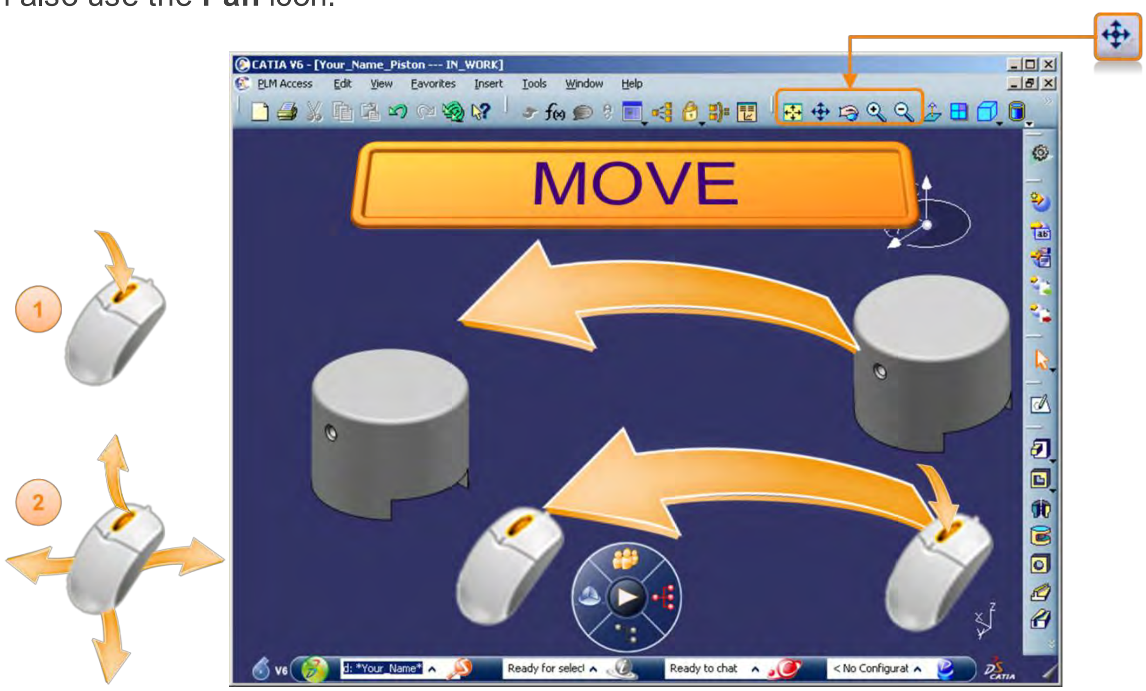
4. The Bar serves as a quick access tool for:
a. Searching for data
b. Examining the impacts of modifications
c. Collaborating with people
d. Saving the modifications to the database

Mouse Manipulation

- To pan the view:

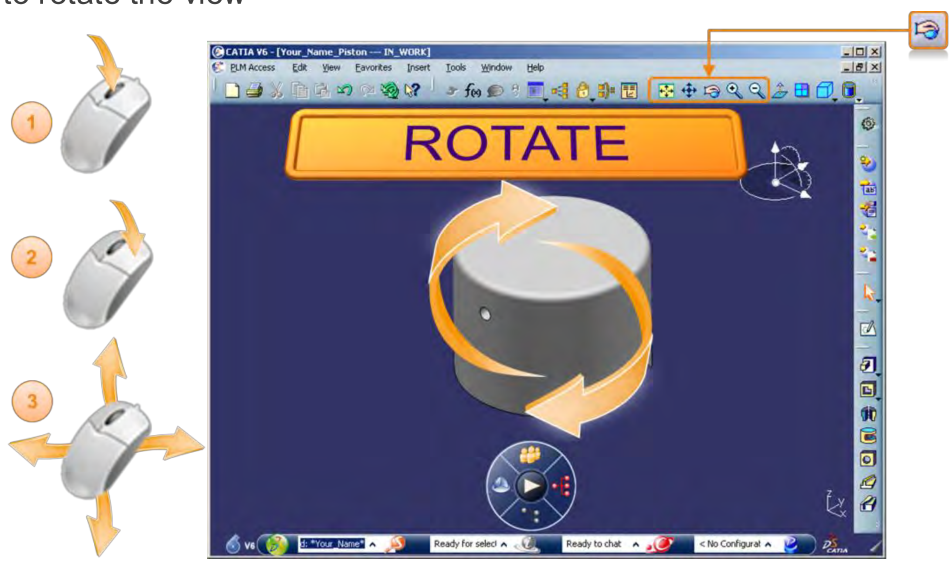
1. Hold-click on the middle button.
2. Then, while holding the middle button pushed, move your mouse to pan the view.

You can also use the **Pan** icon.



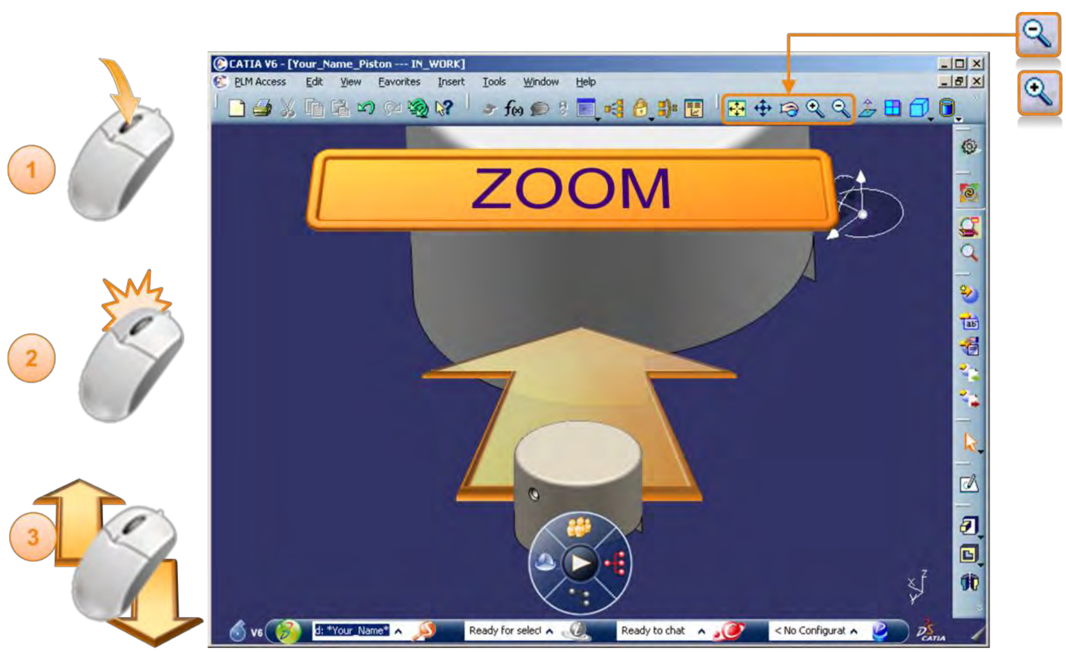
- To rotate the view:

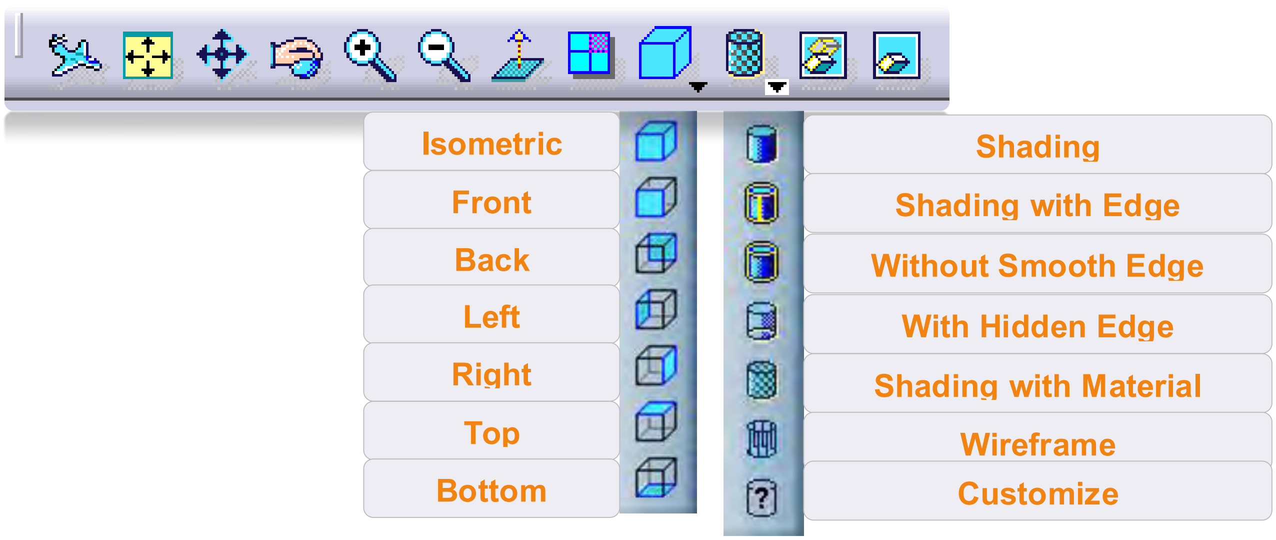
1. Click and hold the middle button.
2. Then, while middle button is pushed, click and hold the right button.
3. Then, while holding middle and pushing the right button, move your mouse to rotate the view



- To zoom in the 3D environment :

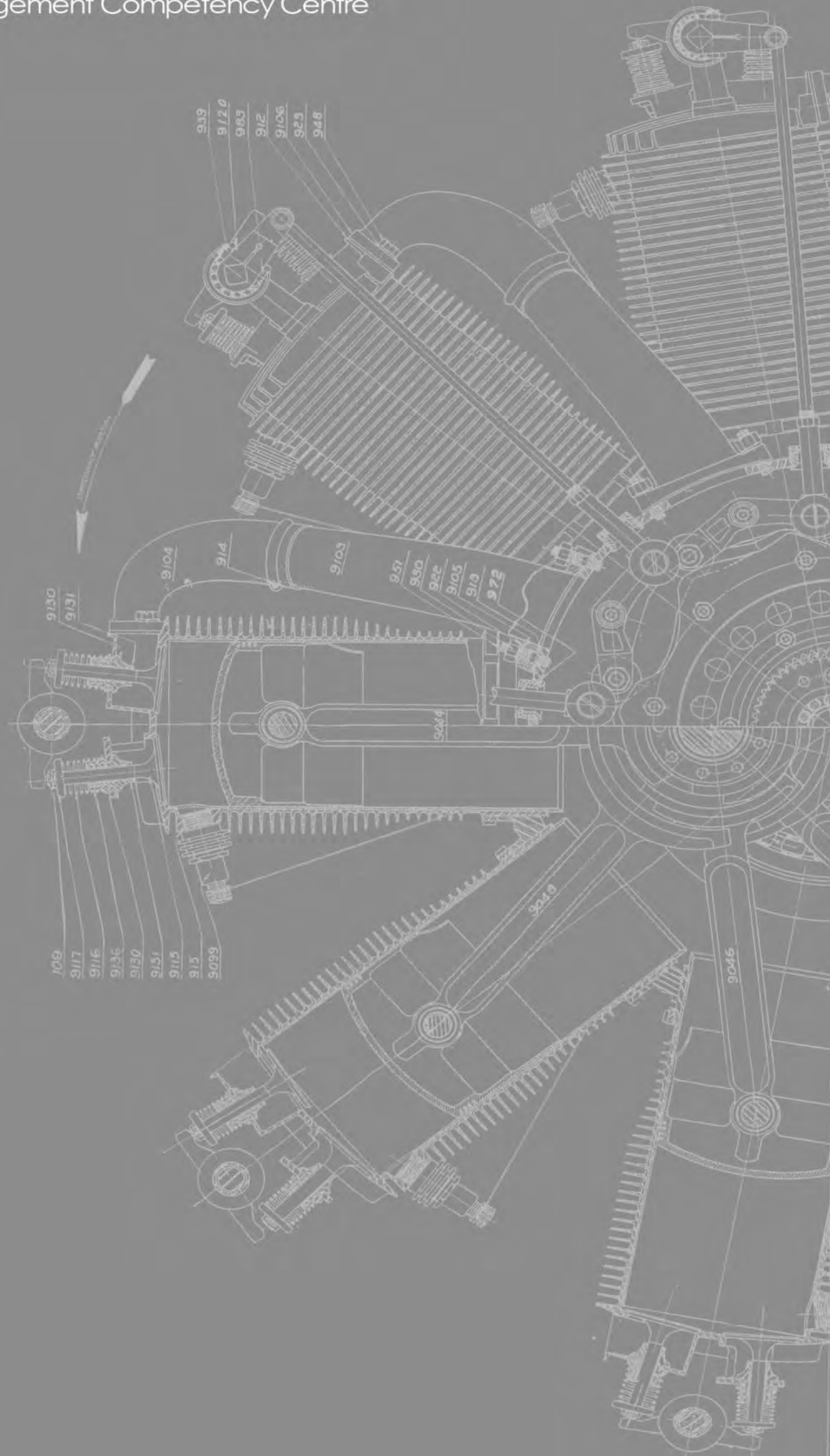
1. Hold-click on middle button.
2. Then, while middle button is pushed, click (and release) the right button.
3. Then, while holding middle button, move your mouse to zoom in or out.




Isometric
Front
Back
Left
Right
Top
Bottom
Shading
Shading with Edge
Without Smooth Edge
With Hidden Edge
Shading with Material
Wireframe
Customize

PLMCC

Product Lifecycle Management Competency Centre



Part Design

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Part Design

During these steps, you will design the piston using Part Design.
Part Design makes it possible to design precise 3D mechanical shapes, from simple to advanced.

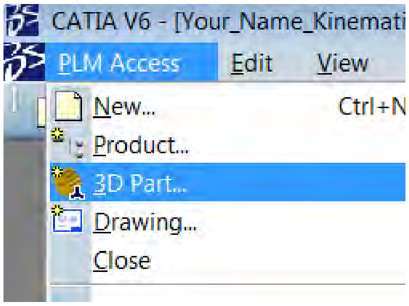
Log onto the **TESTSERVER** to start the part design tutorial.

(The data to fill in these three fields will be supplied by your administrator.)

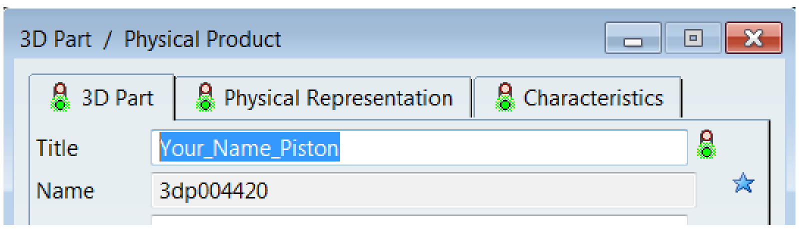
Design Process of the Piston

1. Create a new part

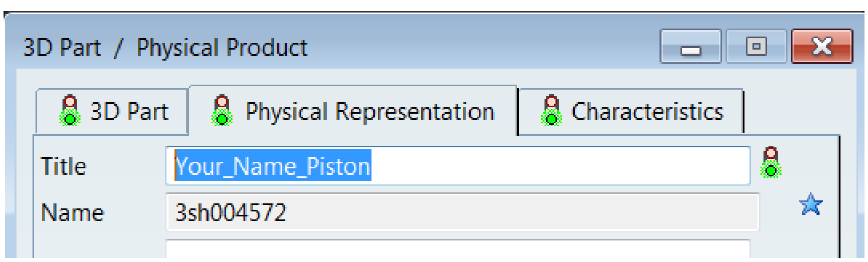
a. Select „PLM Access" > „3D Part…"



c. Type „Your_Name_Piston" in the field „3D Part Title",

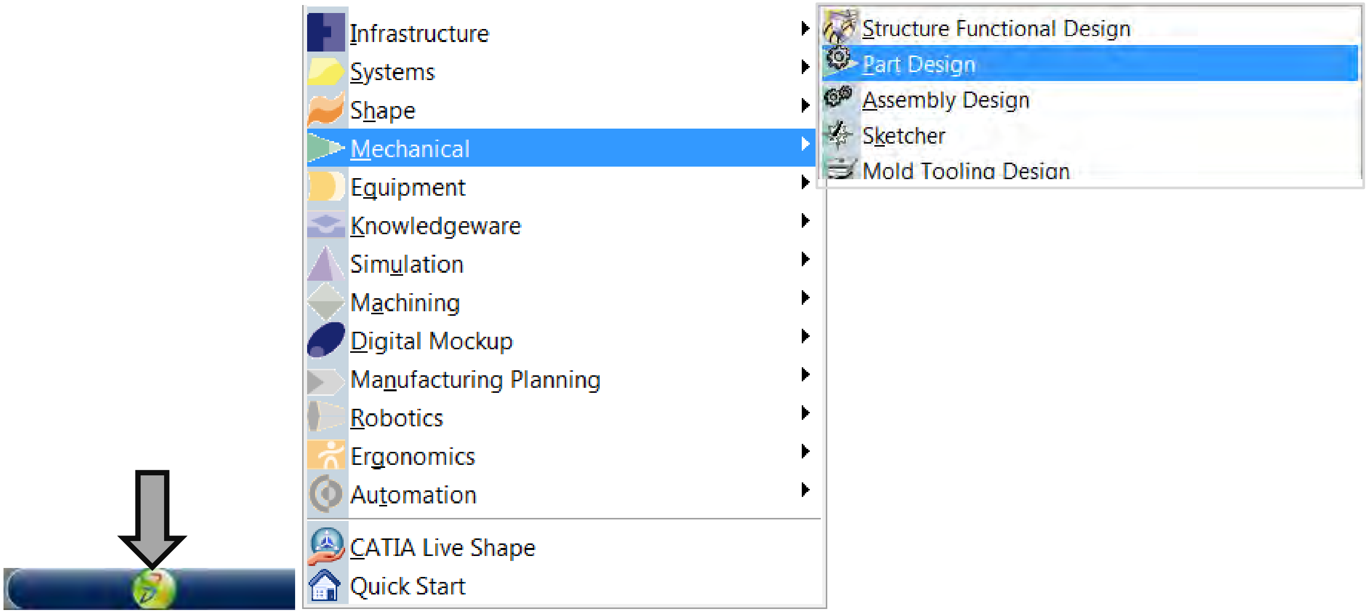


d. Type „Your_Name_Piston" in the field „Physical Representation Title",

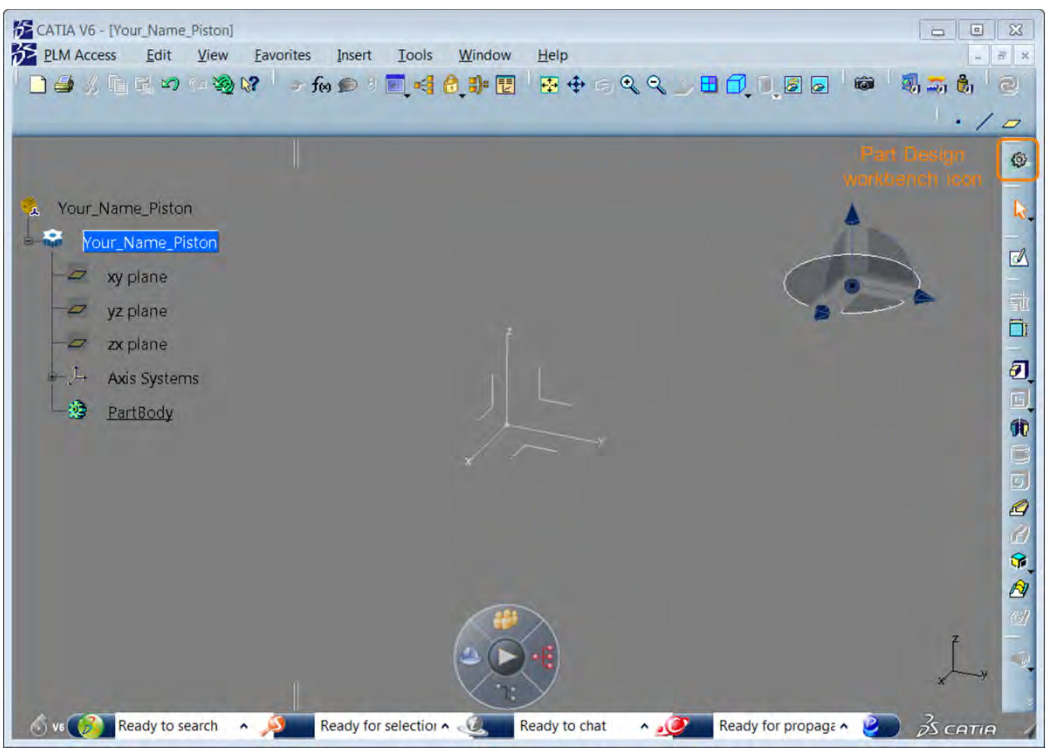


Click „Finish"

2. **To access ‘Part Design’ click ‘Start’ in the bar > ‘Mechanical’ > ‘Part Design.**

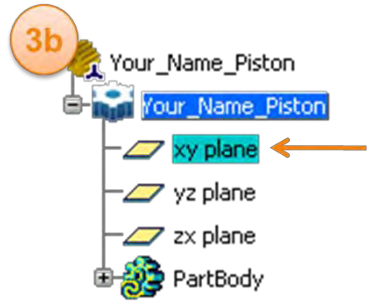


The Part design workbench is opened as shown below.

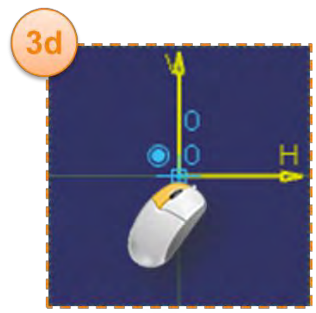


3. Sketch the primary shape of the piston

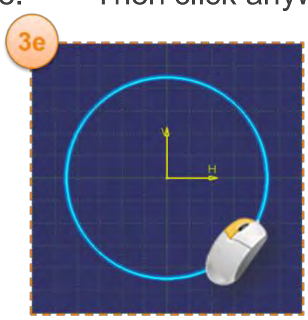
a. Click on the „Positioned Sketch“ icon
b. Select the „xy plane“



c. Click on the „Circle“ icon
d. Select the origin to position the center of the circle

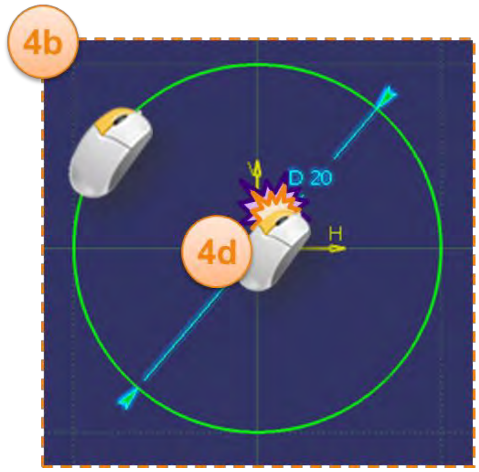


e. Then click anywhere in the graphic area to validate the circle.

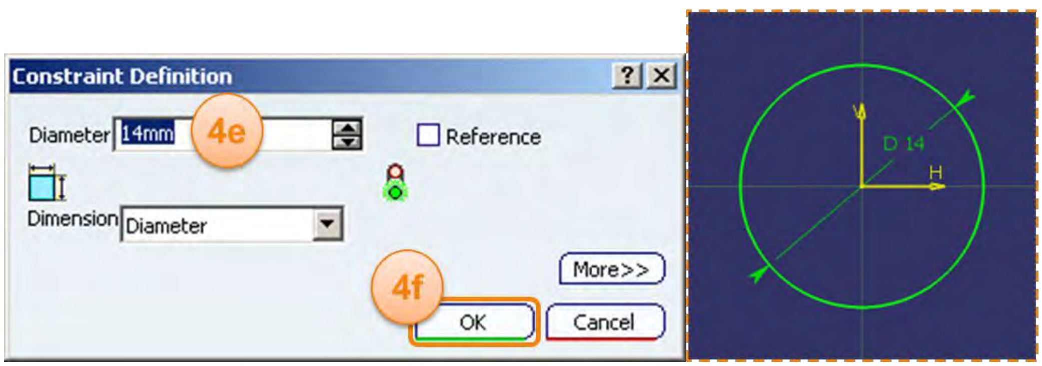


4. Define the diameter of the circle

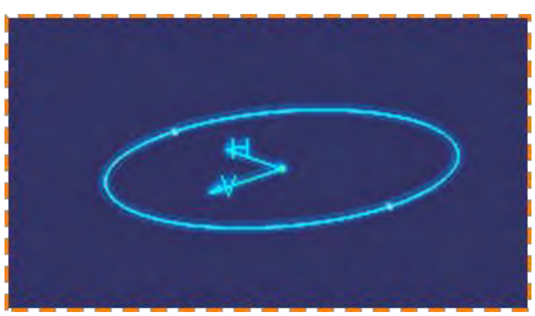
a. Click on the „Constraint“ icon.
b. Select the circle.



c. Click anywhere on the graphic area to validate the dimension.
d. Double click the dimension to modify
e. Enter [14mm]
f. Click „OK“



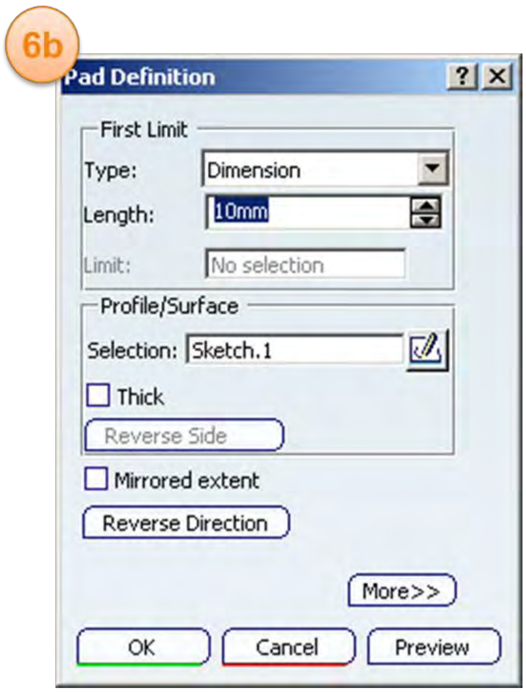
5. Click on the ‘Exit Workbench’ to exit the sketch

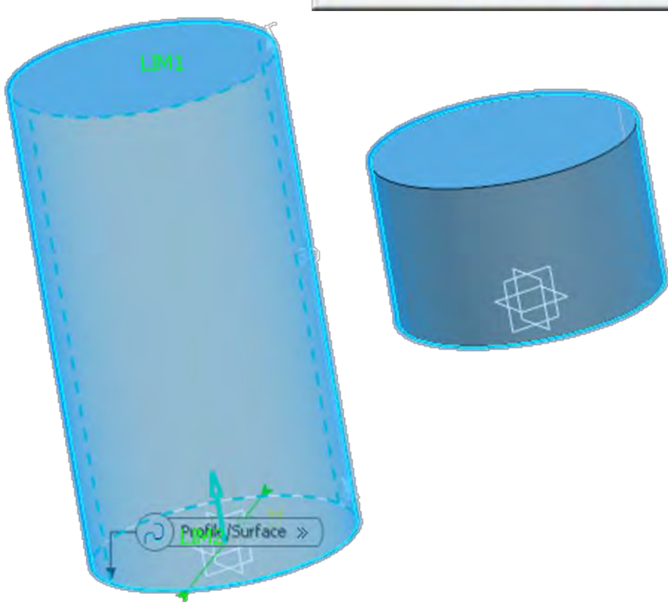


6. Now extrude previously created sketch and generate a cylinder

a. Click on the „Pad" icon to define the extrusion
b. Define the dialog box as shown opposite

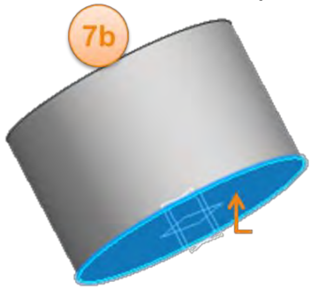
„Type": „Dimension"
„Length" as [10 mm]
„Selection": Click on the „Sketch.1"





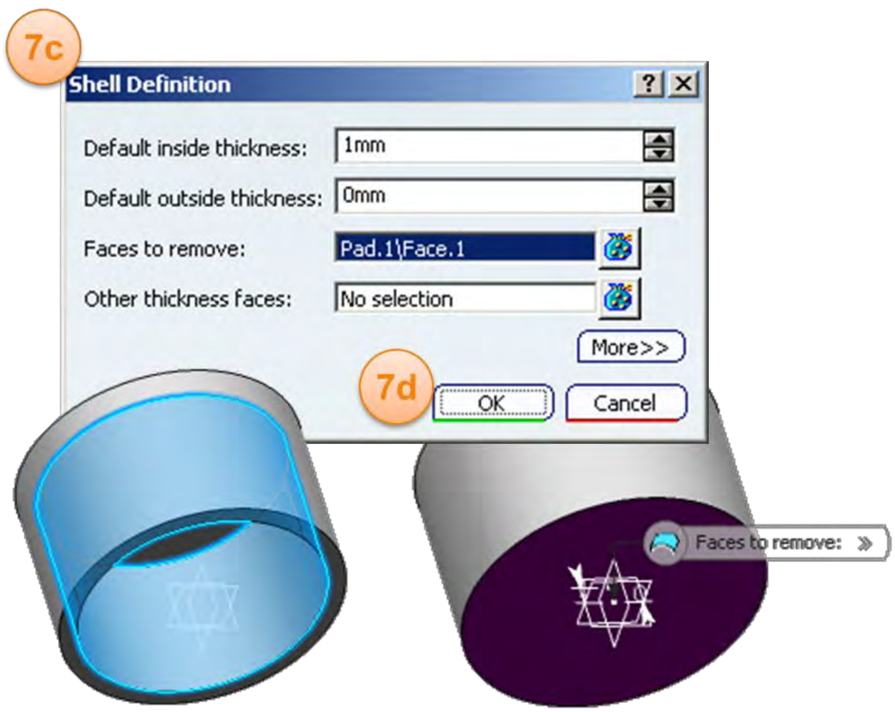
7. Now perform a shell to remove matter inside the piston.

a. Click on the „Shell" icon
b. Select the bottom face of the piston



c. Define the panel as shown below.
Enter [1 mm] as „Default inside Thickness"
„Face to remove" : „Pad.1\Face.1"

d. Click OK

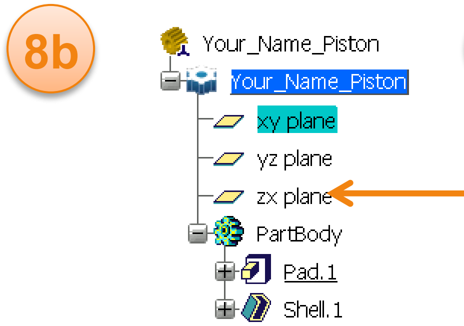


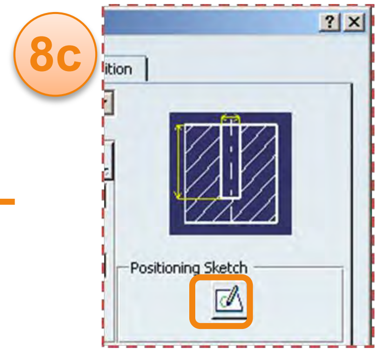
8. Define the hole for the axle between the piston and the connecting rod.

a. Click on the „Hole“ icon.

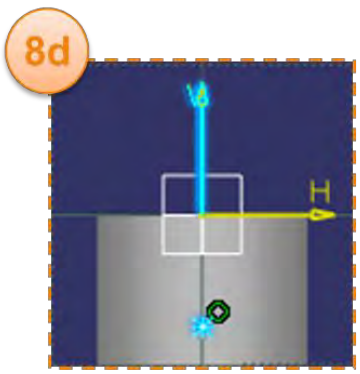
b. Select „zx plane“ .

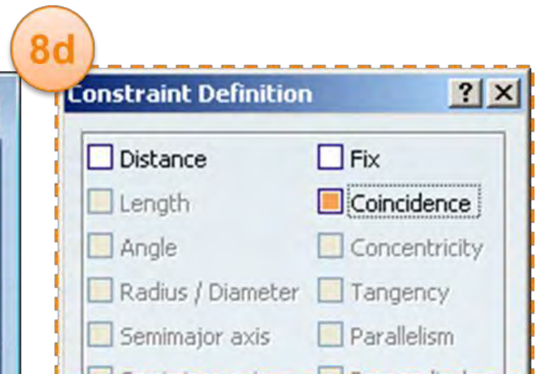
c. Click on the „Positiong Sketch“ icon to position the center of the hole.



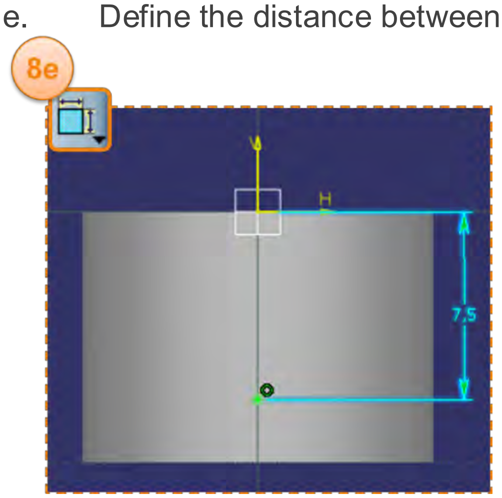


d. Select the hole center, the „yz plane“ and click „Constraints Defined in Dialog Box“ icon then select „Coincidence“.





e. Define the distance between „xy plane“ and the center of the hole as 7,5 mm.



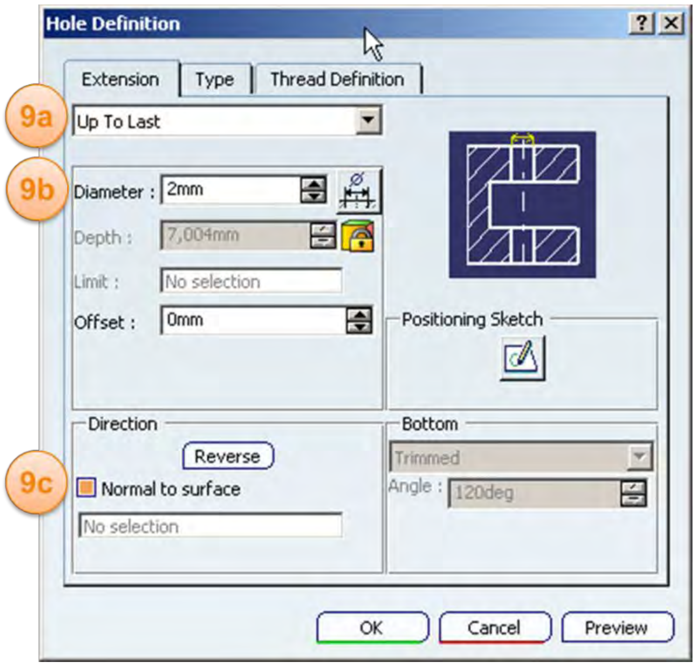
f. Then exit the sketch by clicking on the „Exit“ icon. 

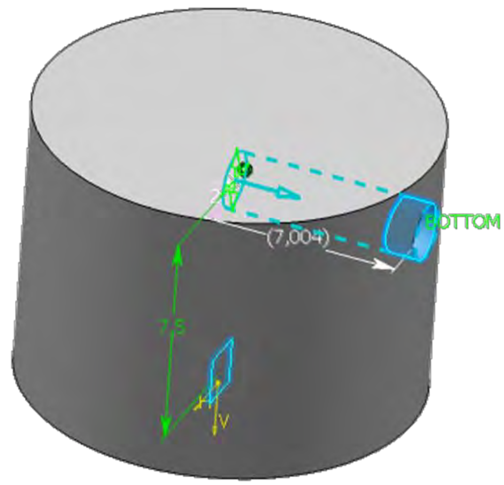
9. **Define the hole as indicated opposite.**

a. „Extension": „Up To Last" .

b. „Diameter" as [2 mm] .

c. „Check „Normal to surface" .

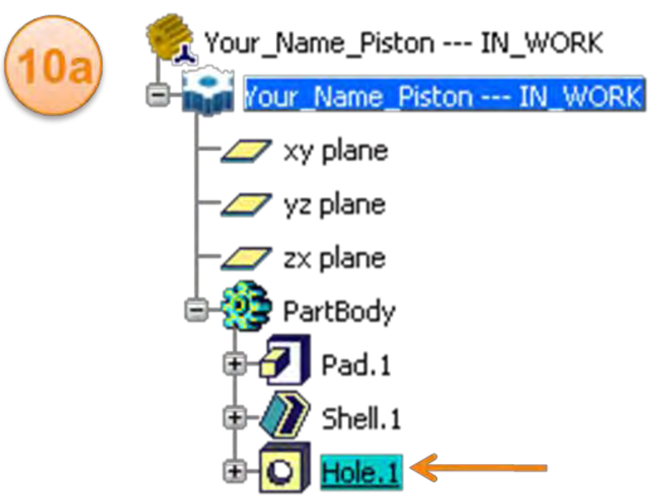




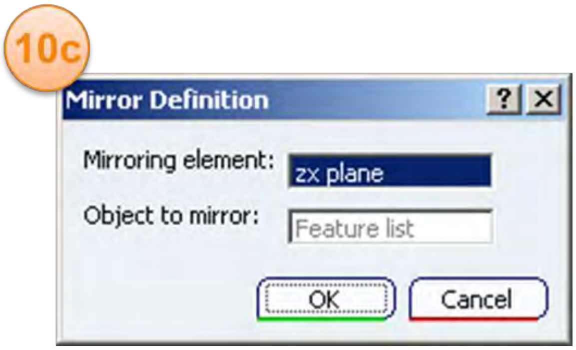
10. **Perform a symmetry to define the hole on the other side of the piston.**

a. Select the hole created previously on the feature tree.

b. Click on the „Mirror" icon. It should be on the lower right corner of your screen.

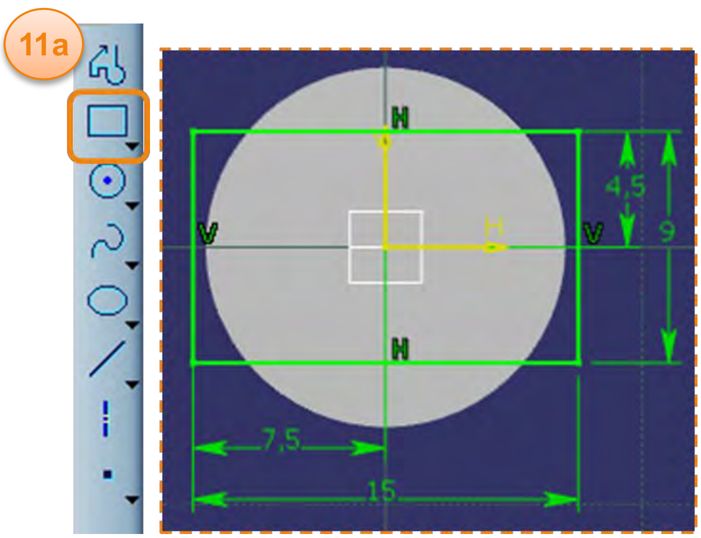


c. Select the „zx plane“ as „Mirroring element“ as indicated opposite.

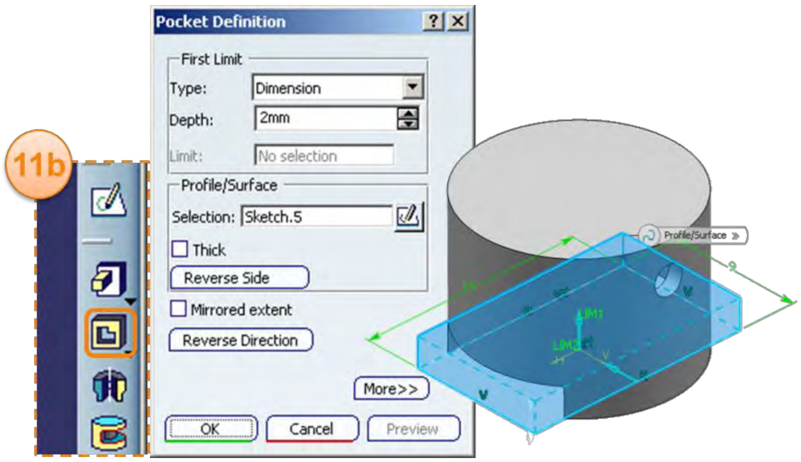


11. Define the bottom face of the piston.

a. Sketch the following sketch on the „**xy plane**“ then exit the sketch.

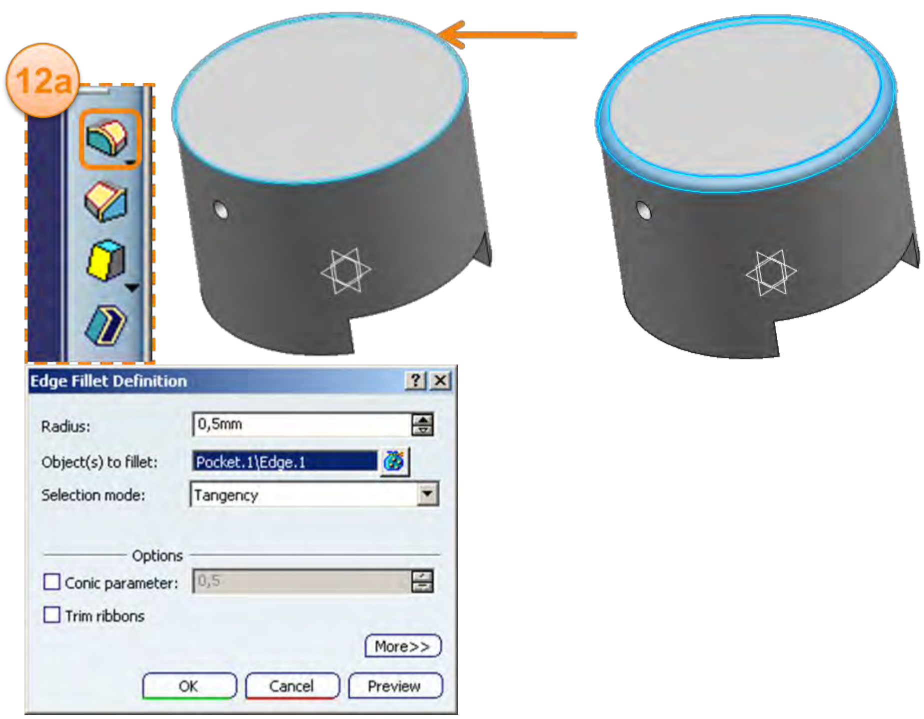


b. Click on the „Pocket“ icon, then define the dialog box as indicated opposite

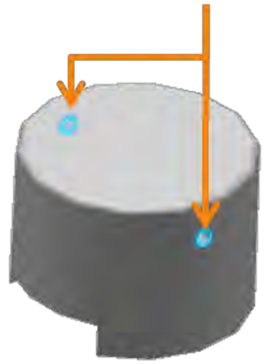


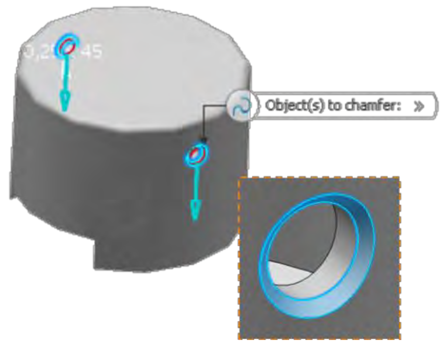
12. To finish, define dress up features.

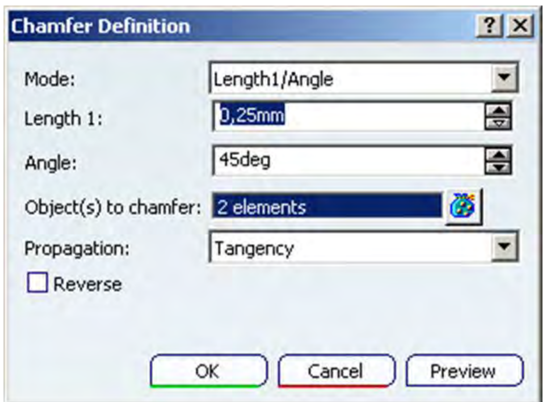
a. Click on the „Fillet" icon, select the top edge and define a radius of 0,5mm.



b. Click on the „Chamfer“ icon, select the 2 edges as shown opposite and define the Chamfer definition as below.







13. Don’t forget to ‘Propagate’ your work to save it in the data base.

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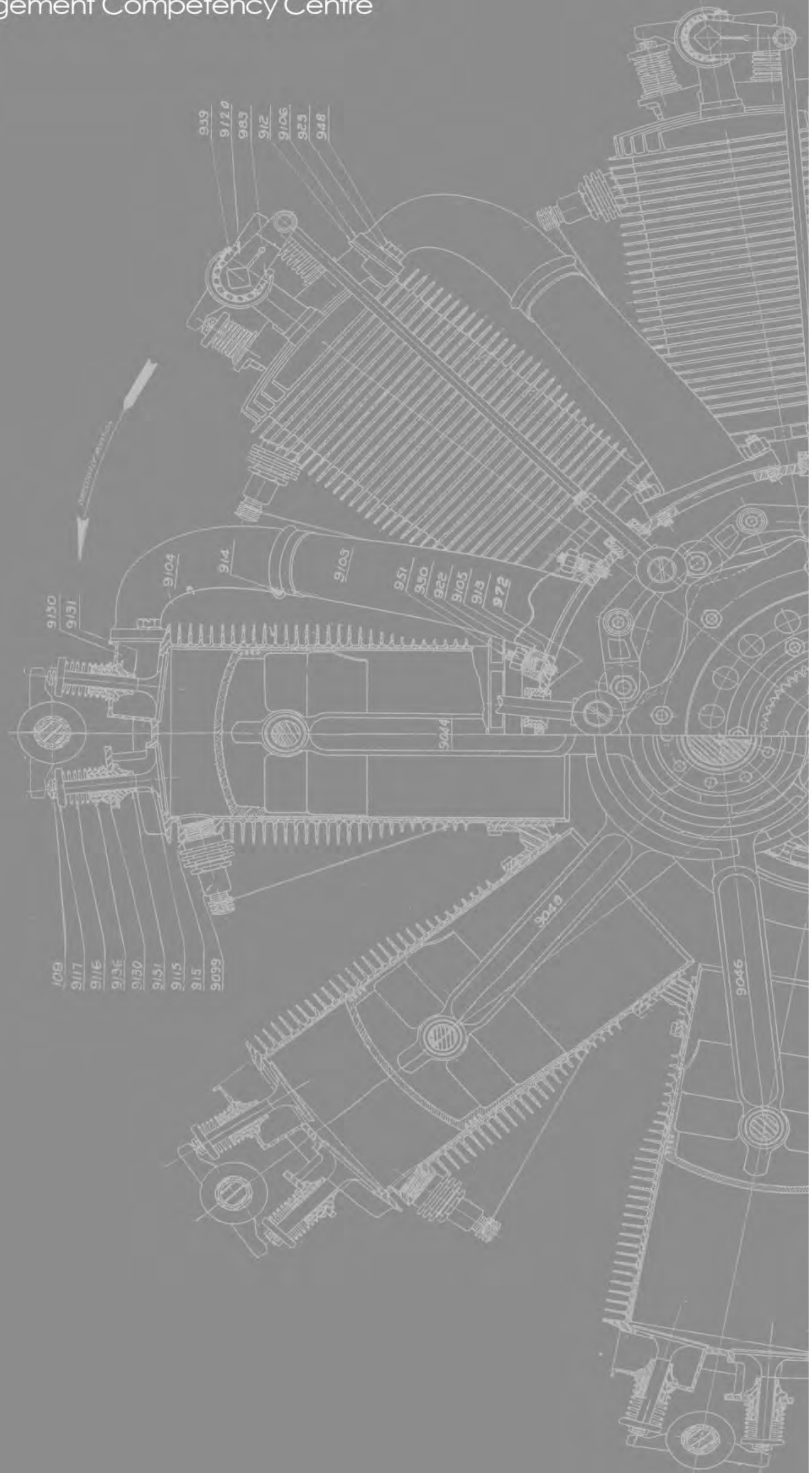

Part Design

Exercises

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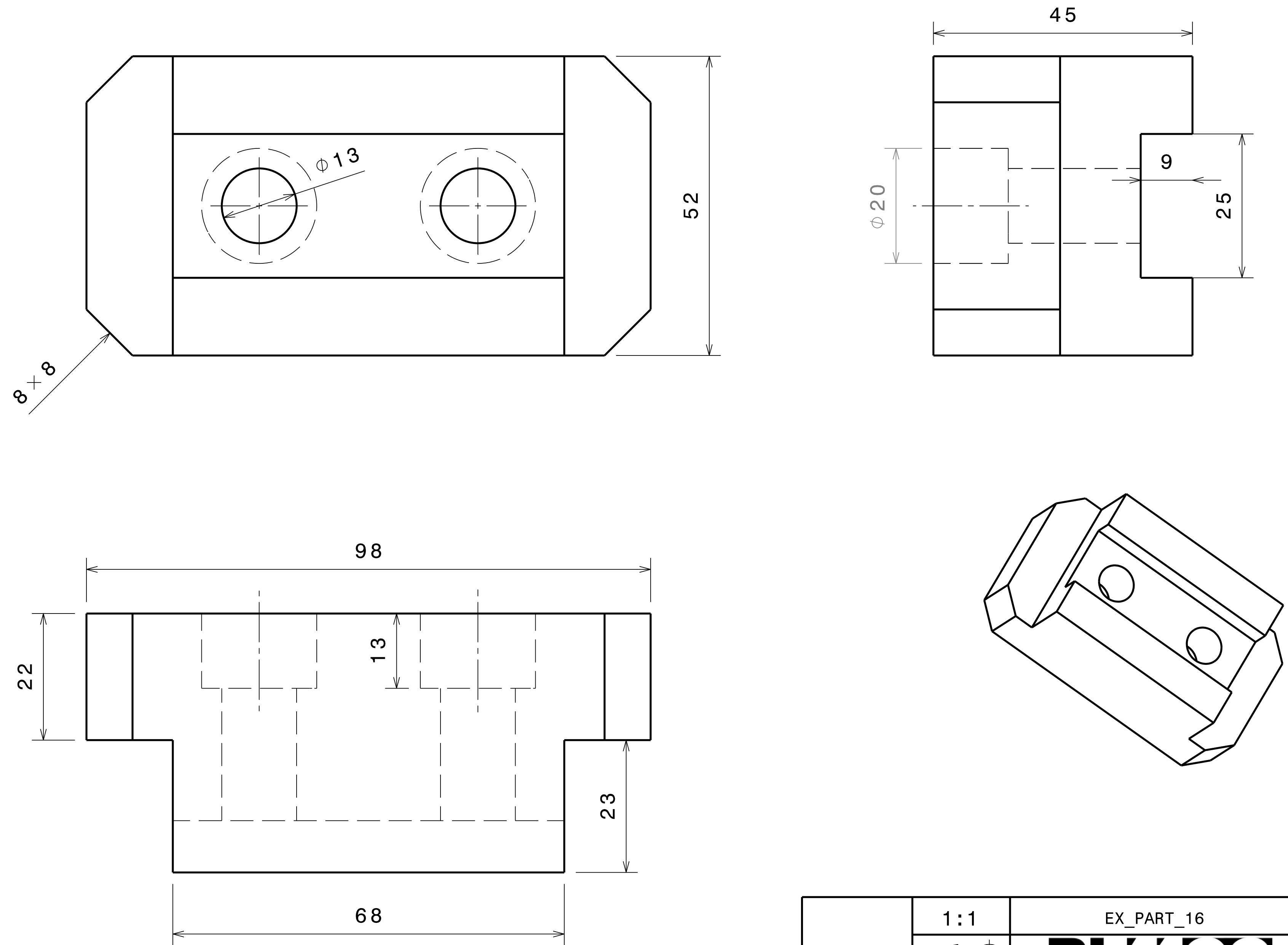
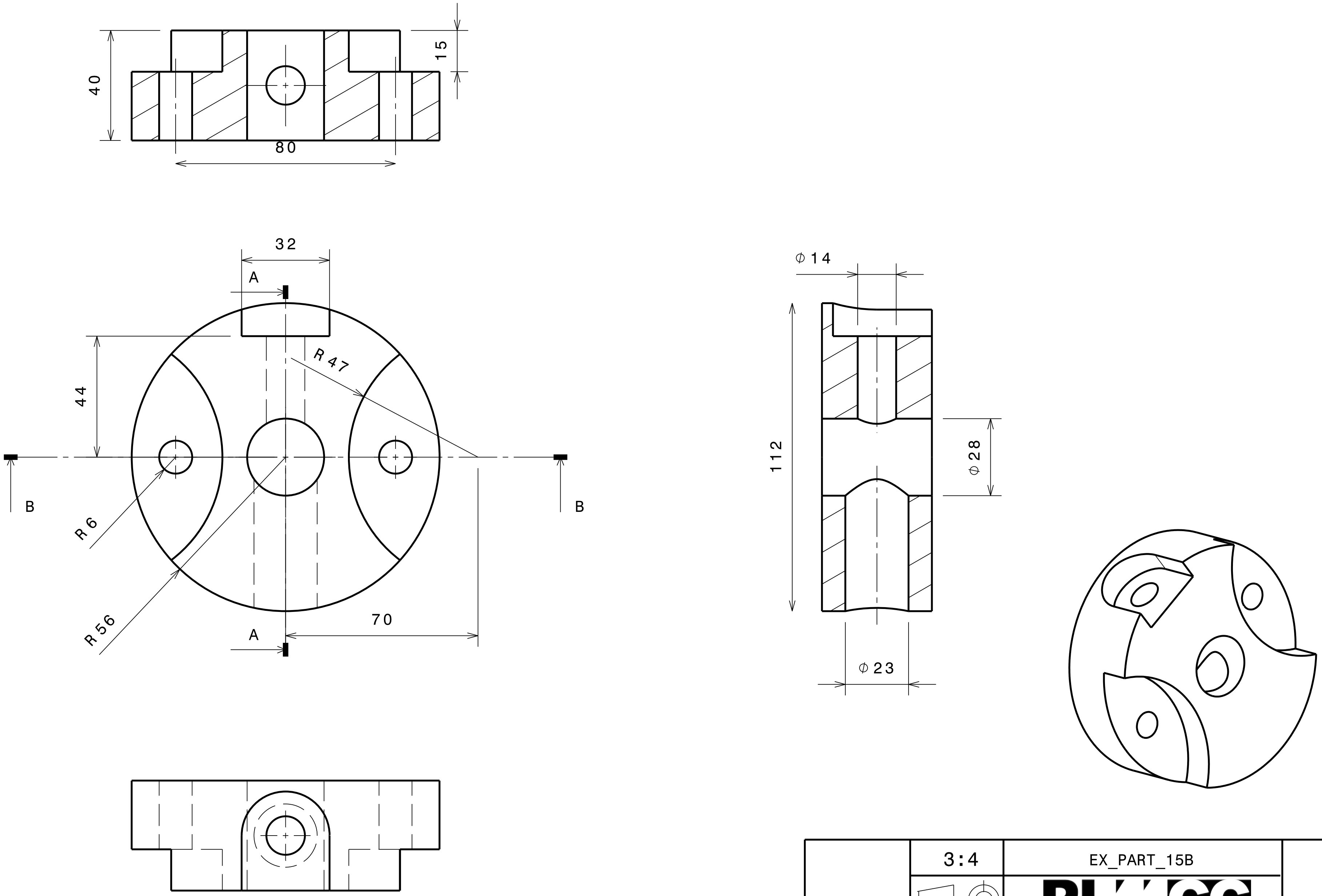
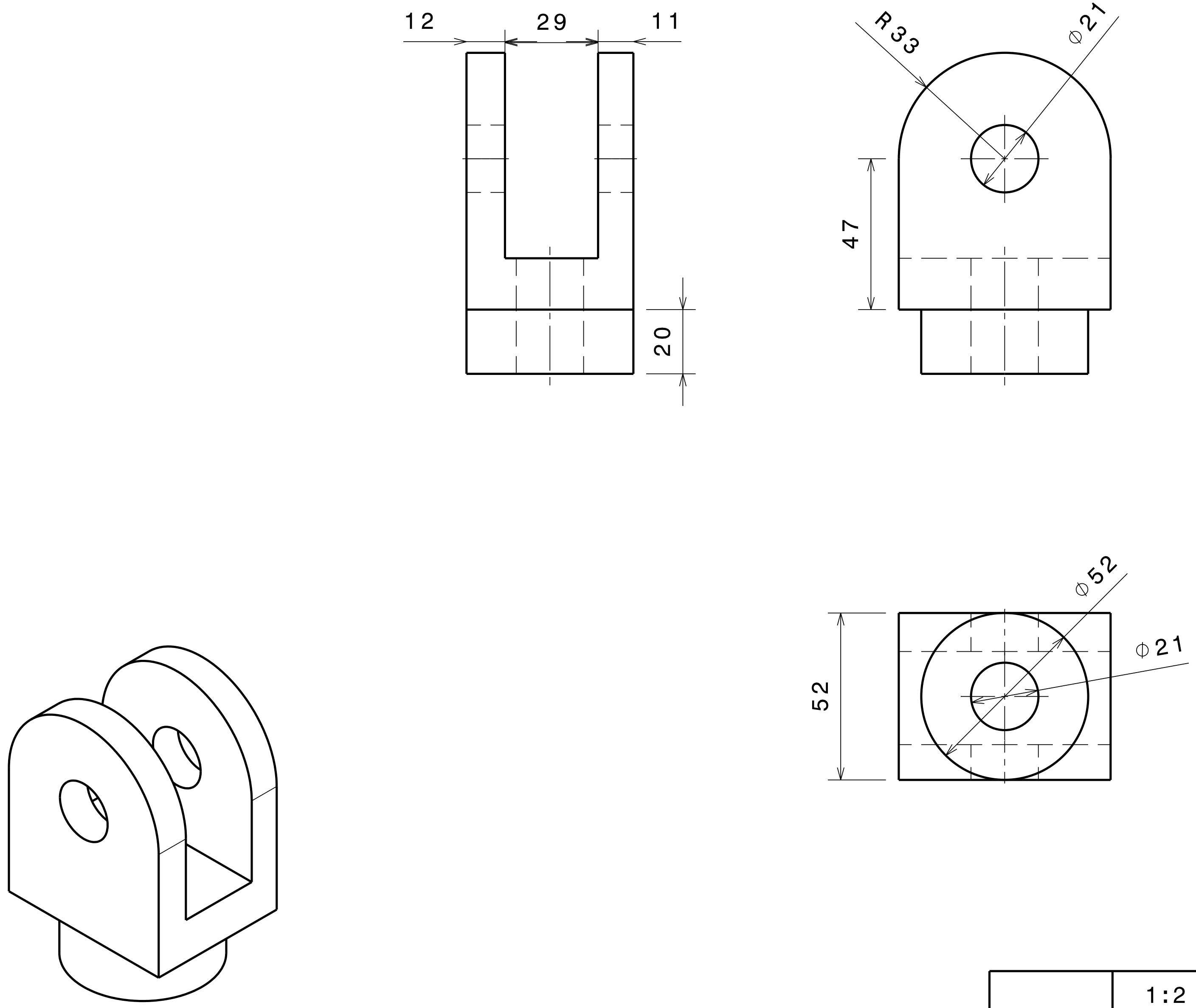


1:1	EX_PART_16	
	PLMCC	
A4	Drawing by : Aneesh	Date:10.09.2008



3:4	EX_PART_15B	
A4	Drawing by : Aneesh	Date: 10.09.2008



1:2

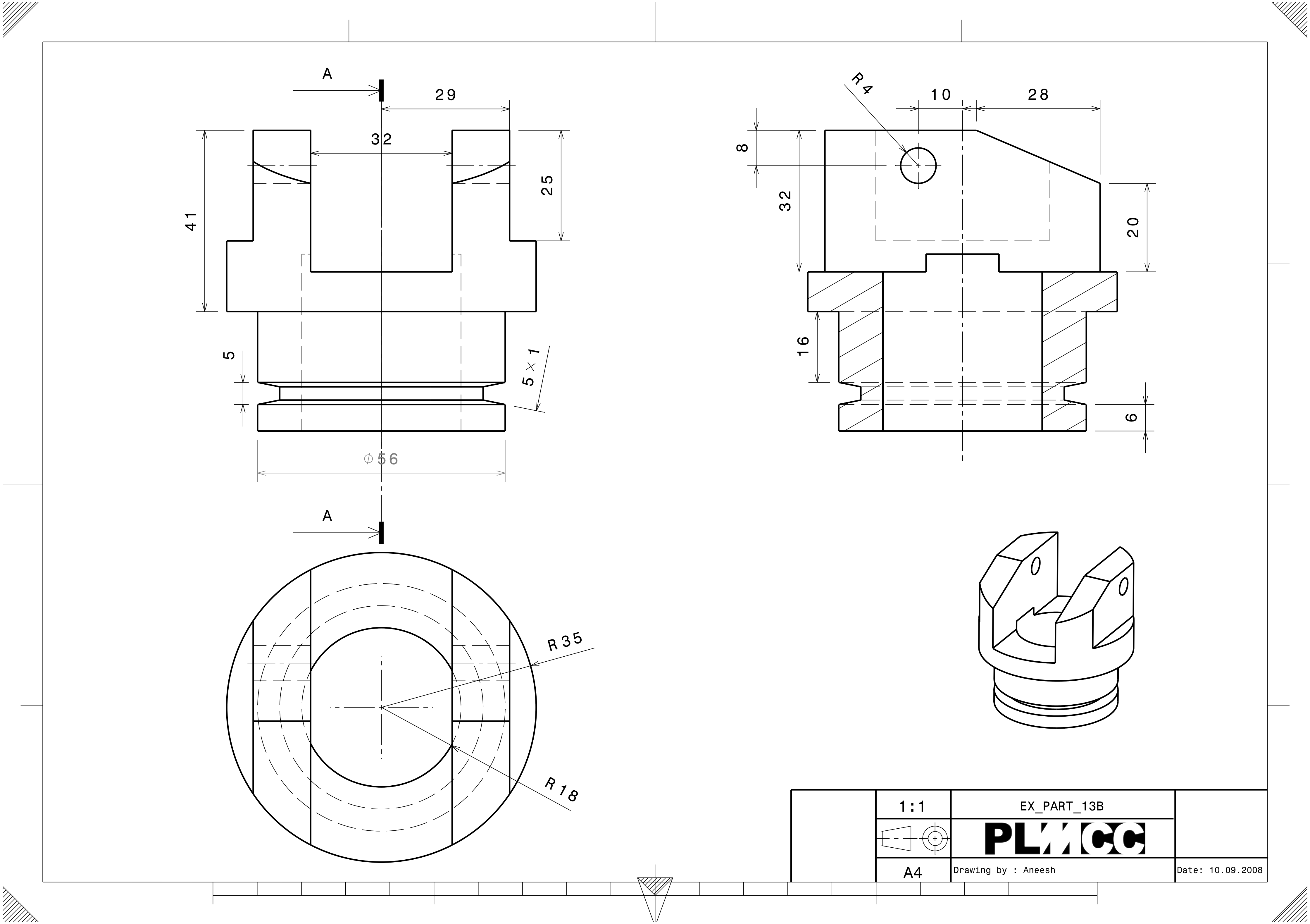
EX_PART_15

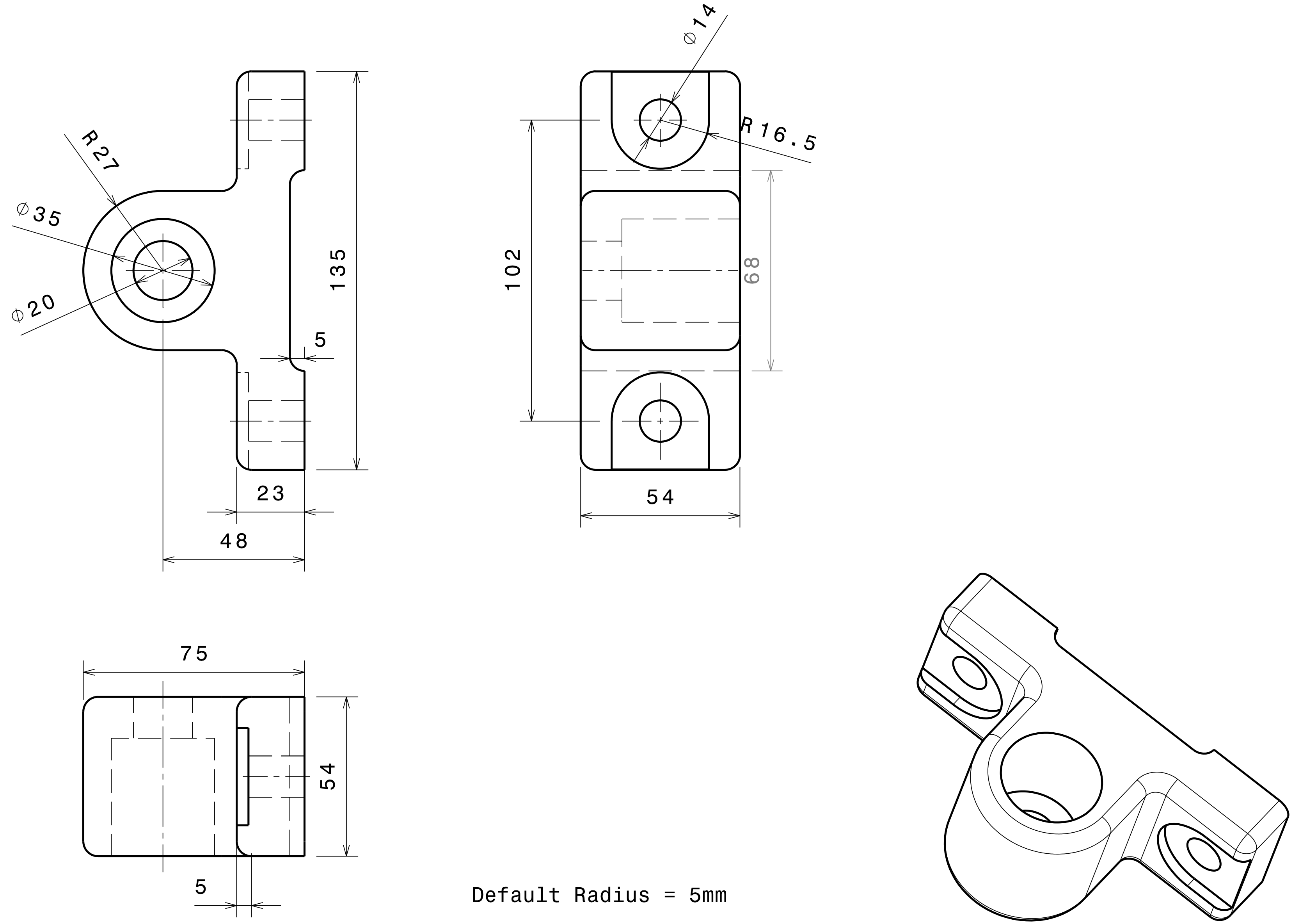
PLMCC

A4

Drawing by :Rakesh KUMAR

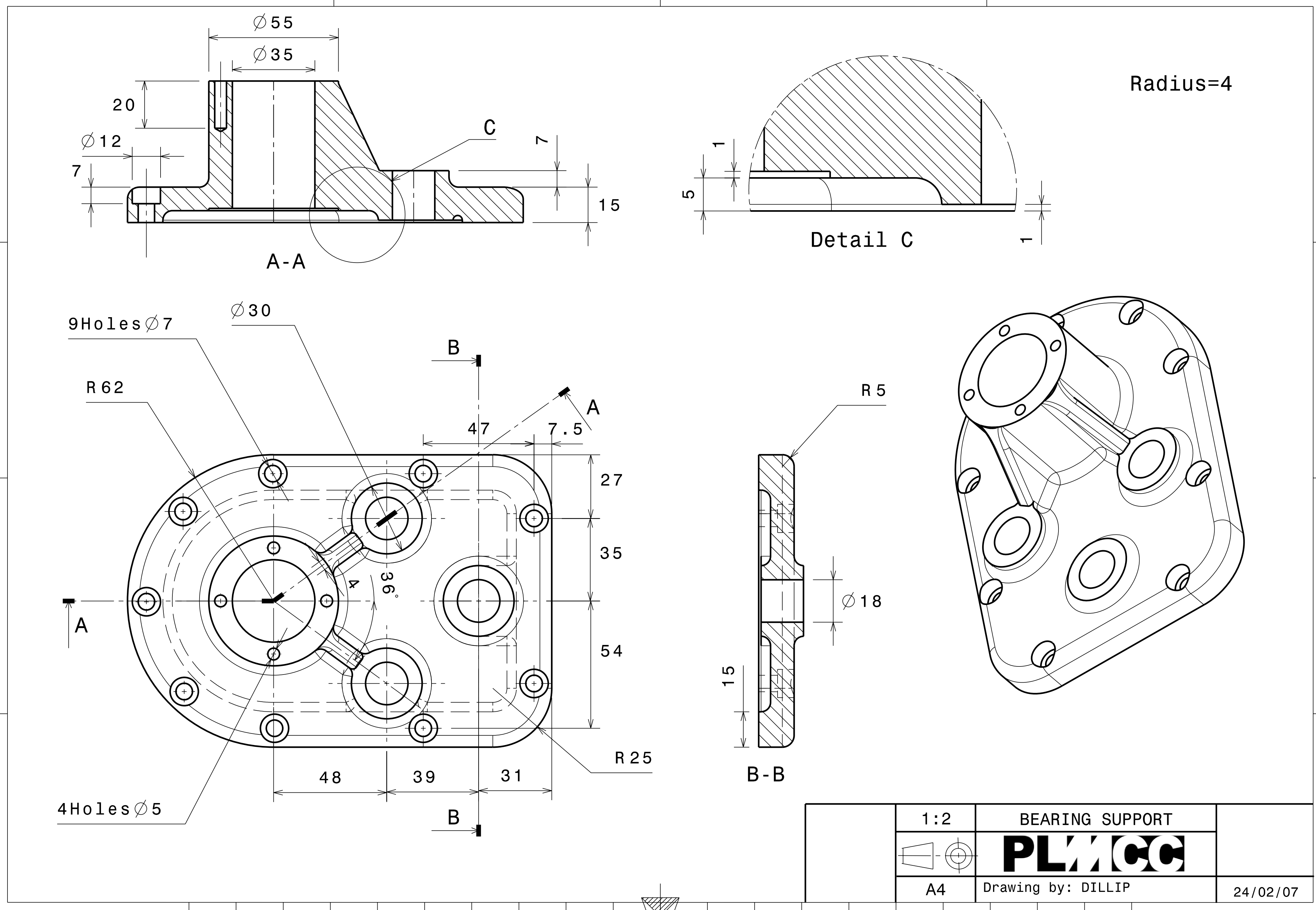
Date:09.07.2008


A
29
32
41
25
5
5 × 1
⌀56
A
R35
R18
R4
10
28
8
32
20
16
6
1:1
EX_PART_13B
A4
Drawing by : Aneesh
Date: 10.09.2008



Default Radius = 5mm

1:2	EX_PART_18	
	PLMCC	
A4	Drawing by :Rakesh KUMAR	Date:09.07.2008



Ø55
Ø35
20
Ø12
7
C
7
15
A-A
1
5
Detail C
1
Radius=4
9HolesØ7
Ø30
R62
B
A
47
7.5
27
35
54
4
36°
A
R5
Ø18
15
B-B
R25
48
39
31
4HolesØ5
B
1:2
BEARING SUPPORT
A4
Drawing by: DILLIP
24/02/07

PLMCC

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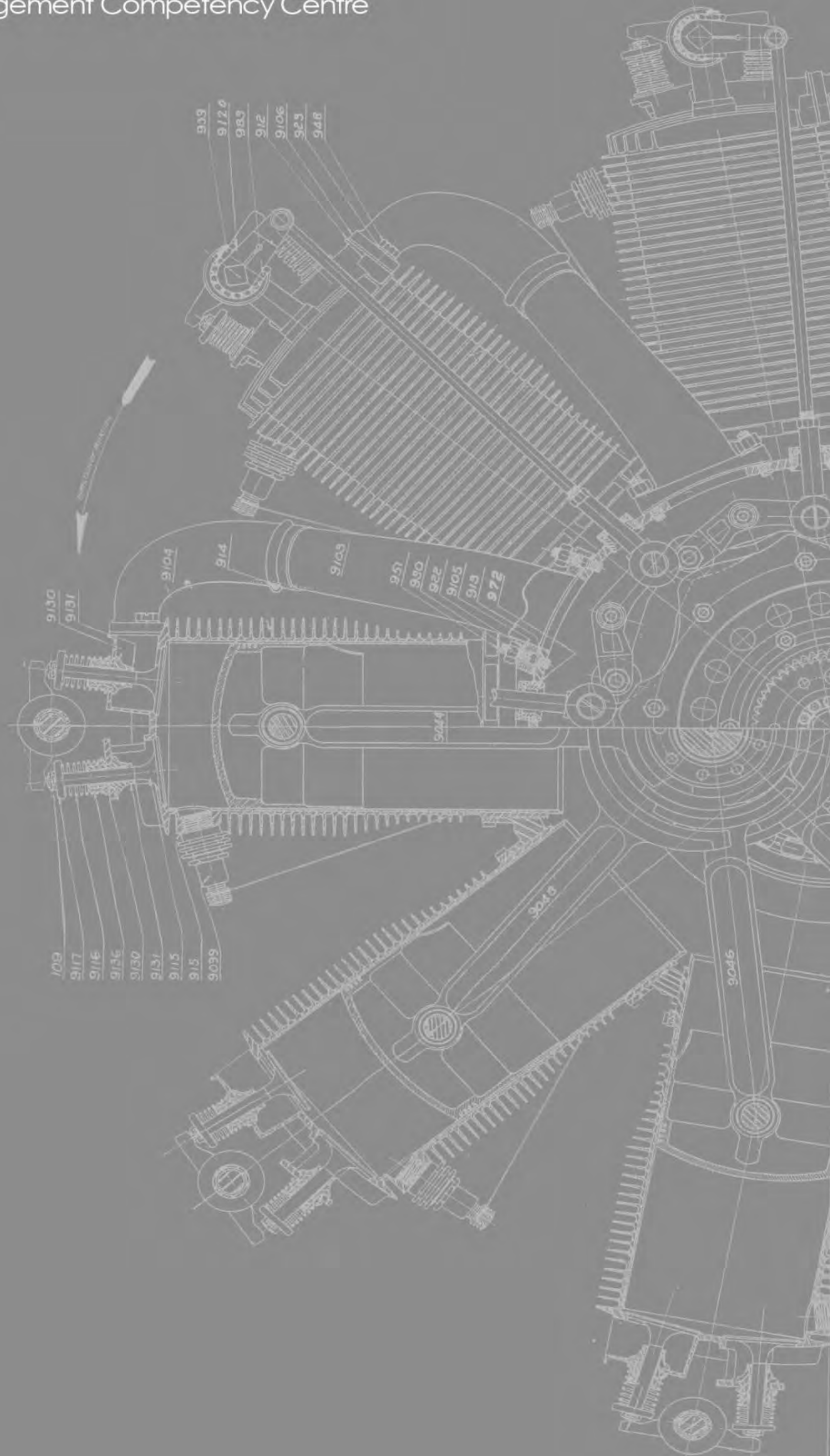


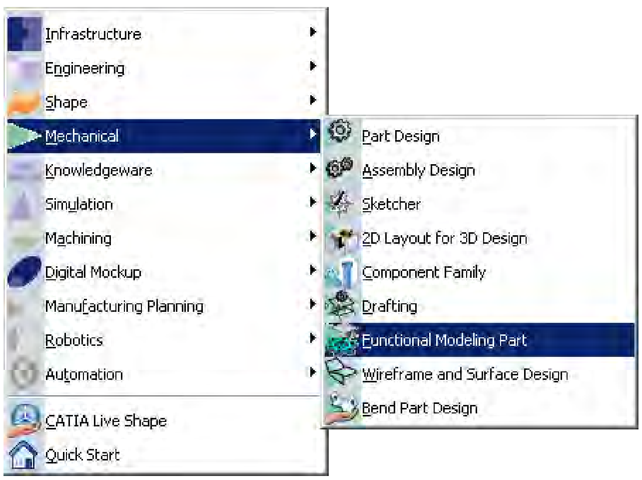
Functional Modelling

Functional Modelling Part

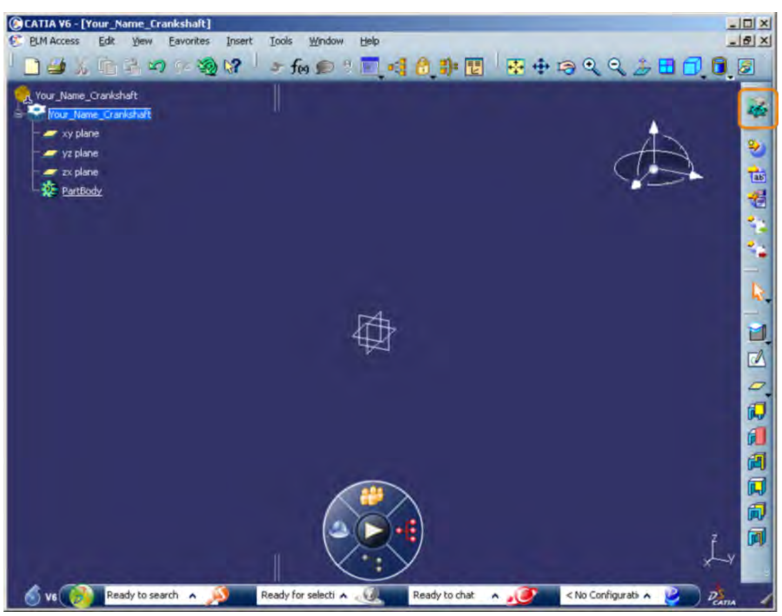
During these steps, you will design the Crankshaft. Functional Modeling Part offers a new approach to the development of 3D digital models.
The objective of Functional Modeling Part is to enable product designers to focus on the functional goals and design constraints of their product.

1. **Create a new part**

2. To access „**Functional Modeling Part**" click „**Start**" in the bar **>** „**Mechanical Design**" **>** „**Functional Modeling Part**"



The „Functional Modeling Part" workbench opens as shown below.



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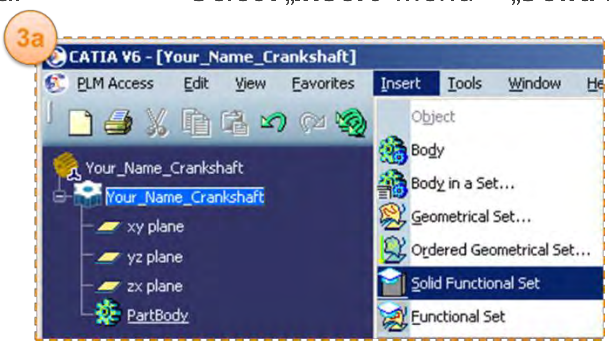
We will define 3 functional sets to design the „Crankshaft":

- General Form
- Connecting Rod
- Fuel_Air

We will finish the design by dress up features

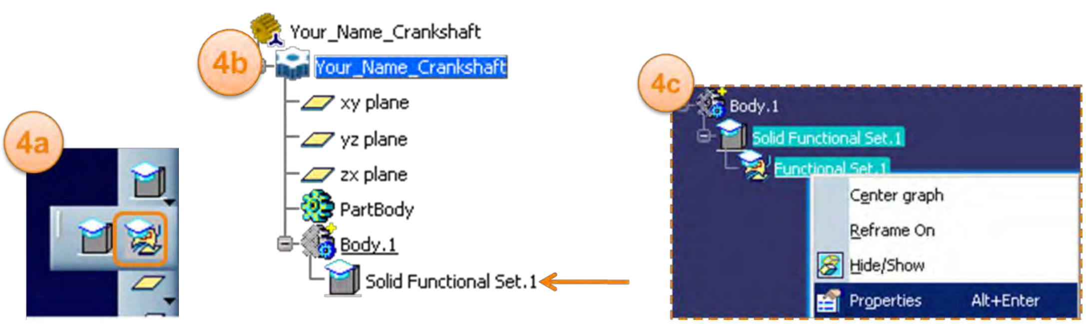
3. Create a „**Solid Functional Set**" where all „**Functional Set**" will be stored.

a. Select „**Insert**" Menu **>** „**Solid Functional Set**"



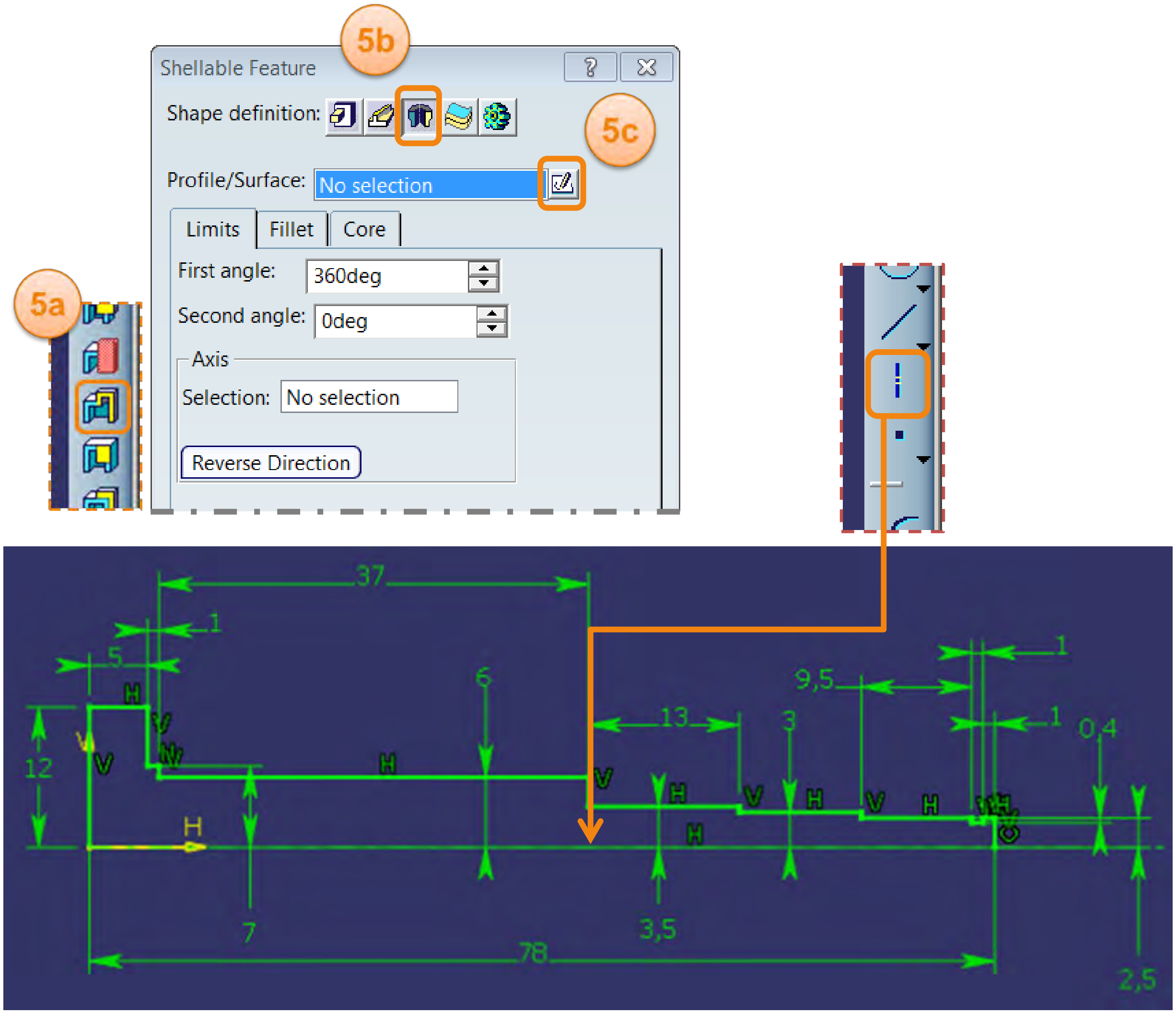
4. Create the first „**Functional set**"

a. Click on the „**Functional Set**" icon
b. Select „**Solid Functional Set**"
c. Right click „**Functional Set.1**" > „**Properties'**
d. Rename the „**Functional Set.1**" to **General Form**.

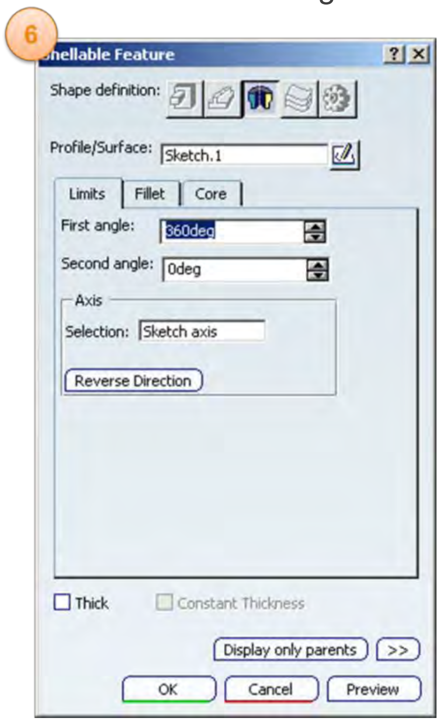


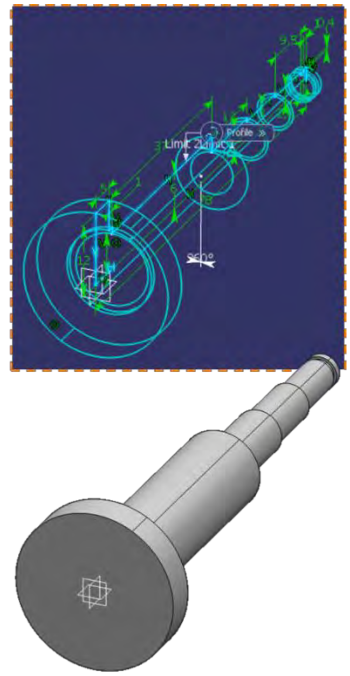
5. **Create the first shape of the Crankshaft**

a. Click on the „**Shellable Feature**" icon
b. Select „**Revolve**" as „**Shape definition**"
c. Select „**Sketch**" icon and select „**zx plane**"
d. Sketch as indicated below
e. Exit the Sketch

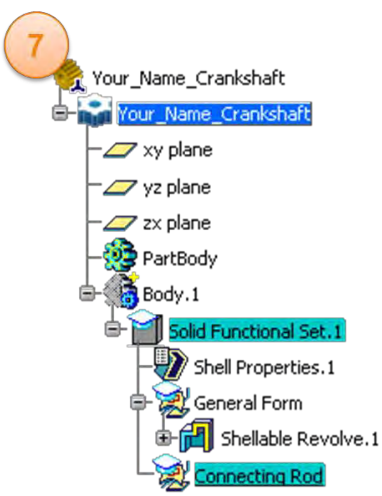

5a
5b
5c
Shellable Feature
Shape definition:
Profile/Surface:
No selection
Limits
Fillet
Core
First angle:
360deg
Second angle:
0deg
Axis
Selection:
No selection
Reverse Direction
37
1
5
6
9,5
13
3
0,4
12
7
78
3,5
2,5

6. Define the dialog box as indicated to create a 3D part using **shaft.**



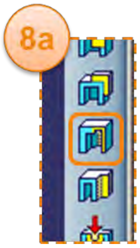


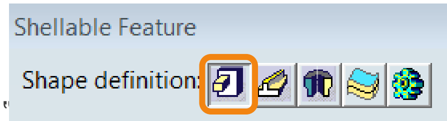
7. Create a new „**Functional Set**“ and rename it „**Connecting Rod**“



8. Create a „**Core Feature**"

a. Click on the „**Core Feature**" icon

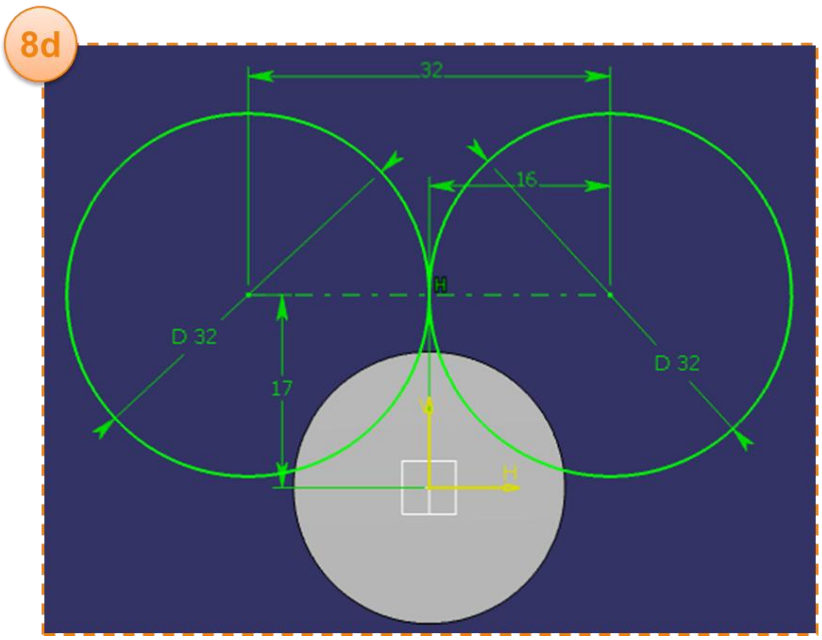




b. Select „**Prism**" as „**Shape definition**"

c. Select the „**Sketch**" icon and select the face of the crankshaft as shown opposite.

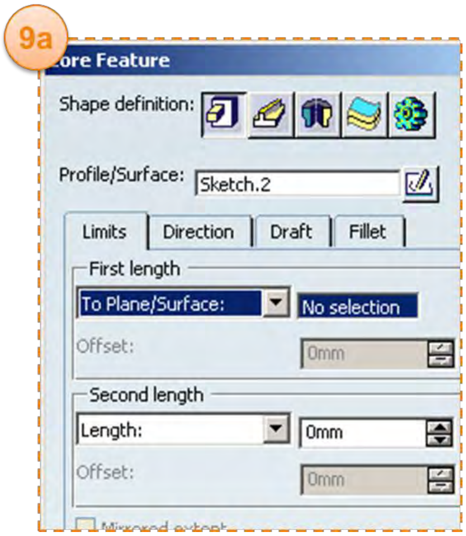
d. Sketch as indicated below

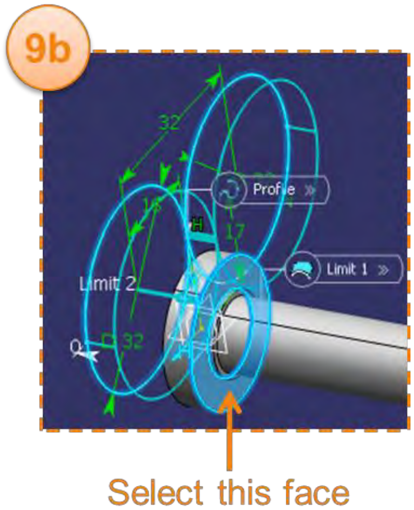


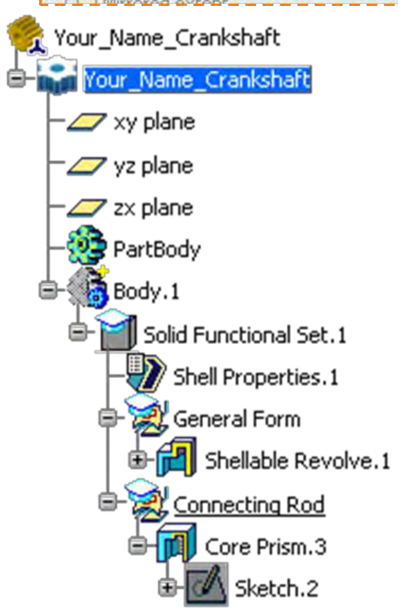
e. Exit the Sketch

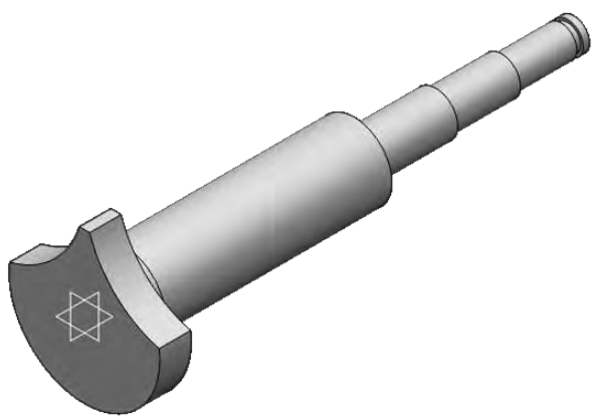
9. **Define the 'Core Feature' as indicated opposite.**

a. Select „**To Plane/Surface**" as „**First Length**" under „**Limits**" Tab
b. Select the face as shown as „**First length**"

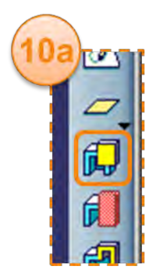






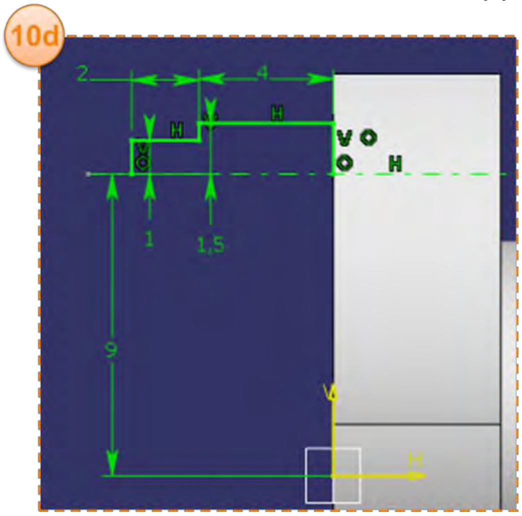


10. Create an „**Added Feature**"



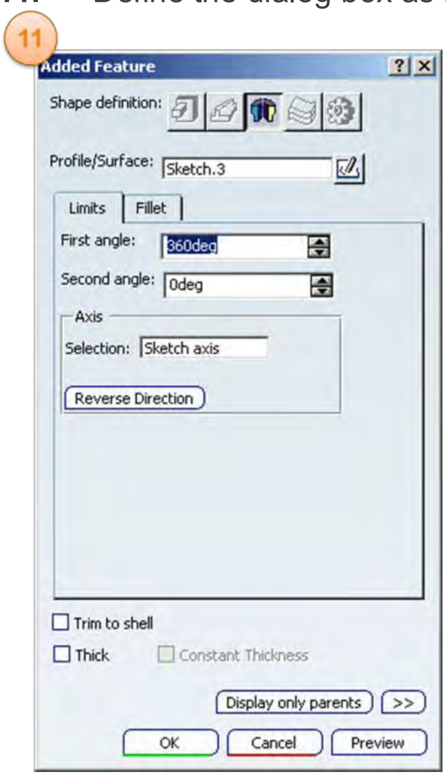
a. Click on the „**Added Feature**" icon
b. Select „**Revolve**" as „**Shape definition**"
c. Select „**Sketch**" icon and select the „**ZY plane**"

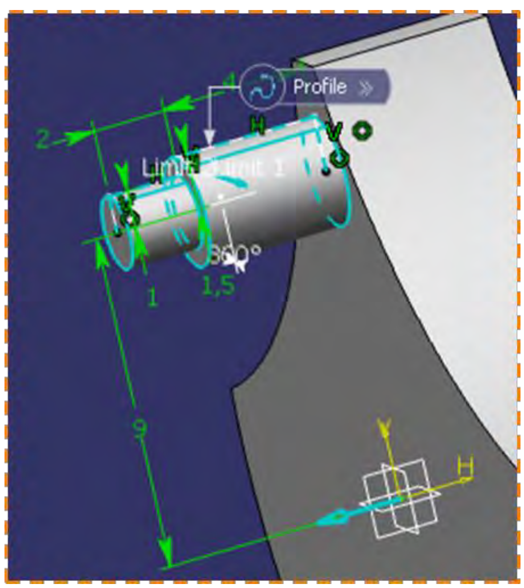
d. Sketch as indicated opposite.



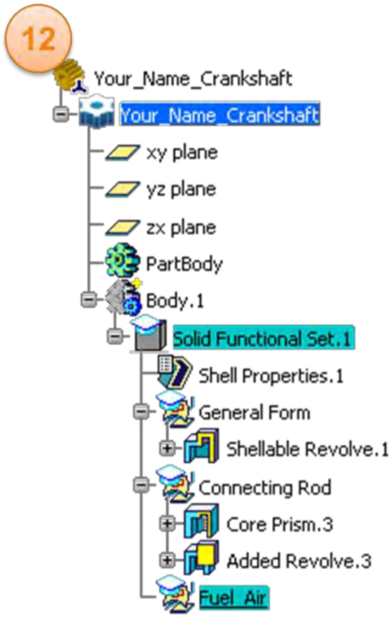
e. Exit the Sketch

11. Define the dialog box as indicated to create a 3D part using **shaft.**



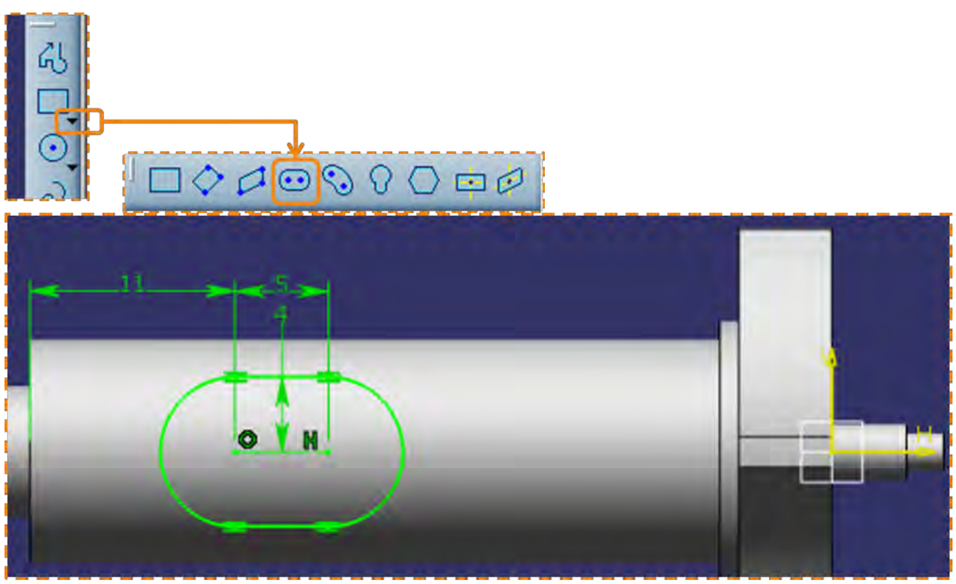


12. Create the last „**Functional Set**" and rename it „Fuel_Air"



13. Create „**Protected Feature**"

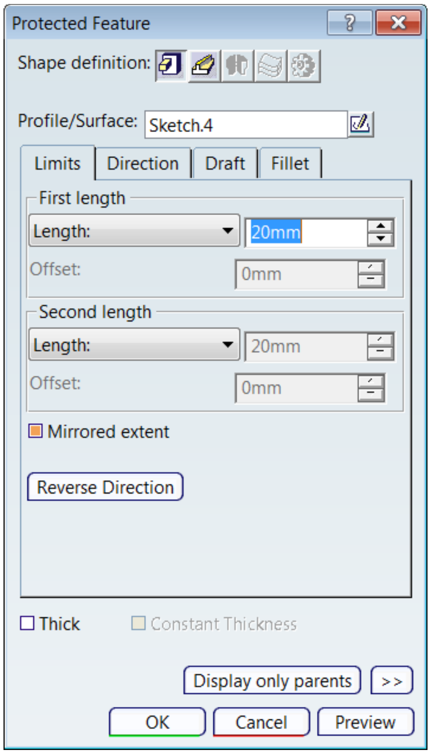
a. Click on the „**Protected Feature**" icon
b. Select „**Prism**" as „**Shape definition**"
c. Select the „**Sketch**" icon and select the „**xy plane**"
d. Sketch as indicated.

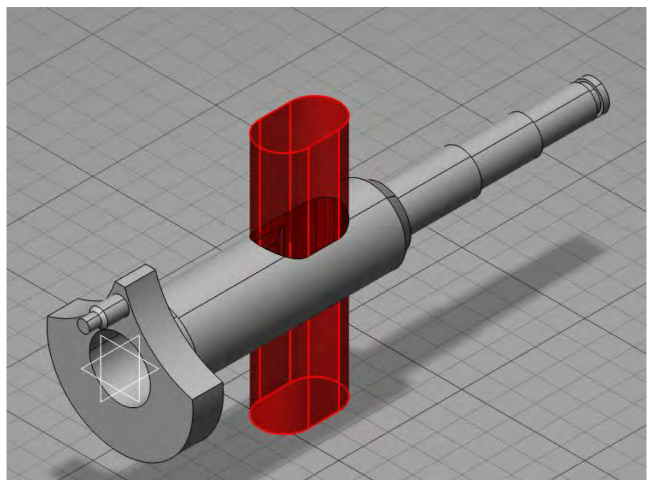


e. Exit the Sketch

14. Define the dialog box as indicated opposite.

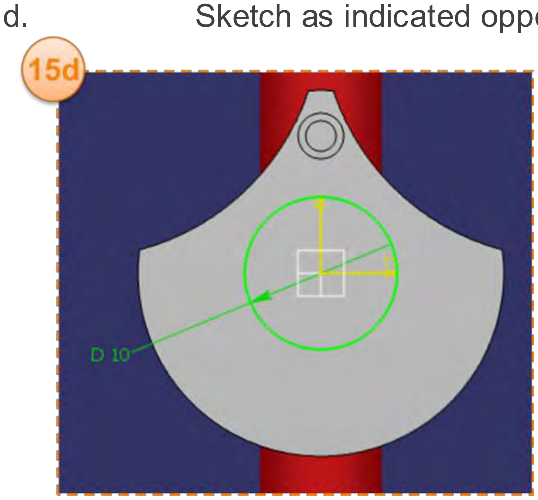
a. Check „**Mirrored extent**"
b. Specify the „**First length**" as [20mm]





15. Create the last „**Protected Feature**"

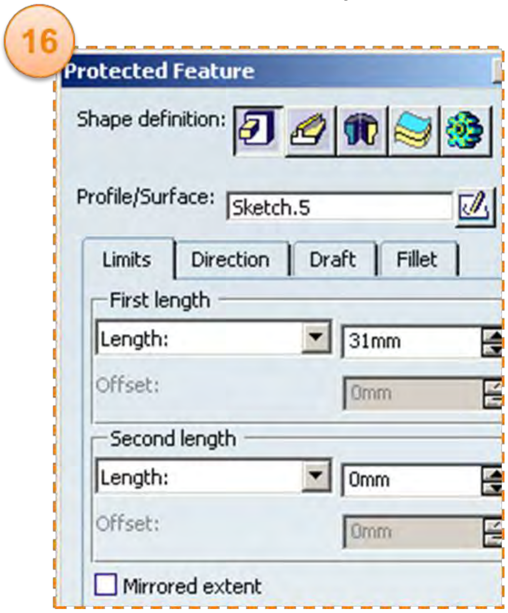
a. Click on the „**Protected Feature**" icon
b. Select „**Prism**" as „**Shape definition**"
c. Select the „**Sketch**" icon and select the „**yz plane**"
d. Sketch as indicated opposite.

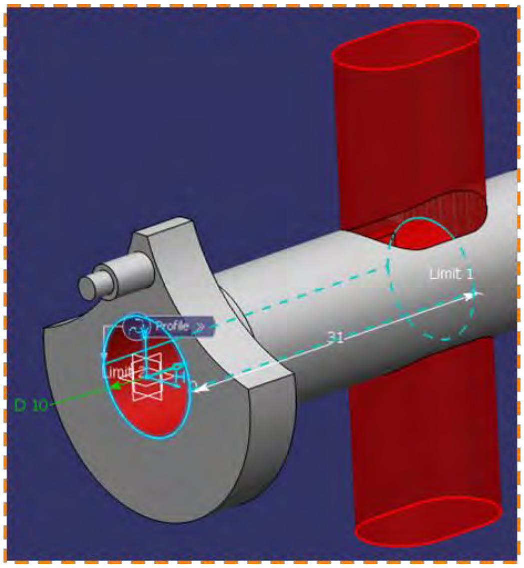


e. Exit the Sketch

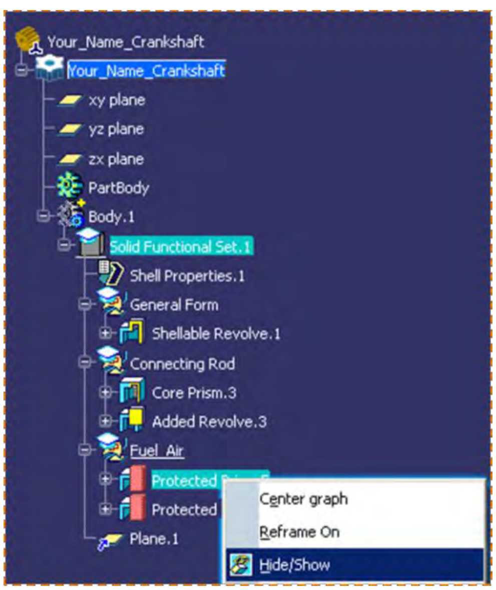
16. Define the dialog box as indicated opposite.

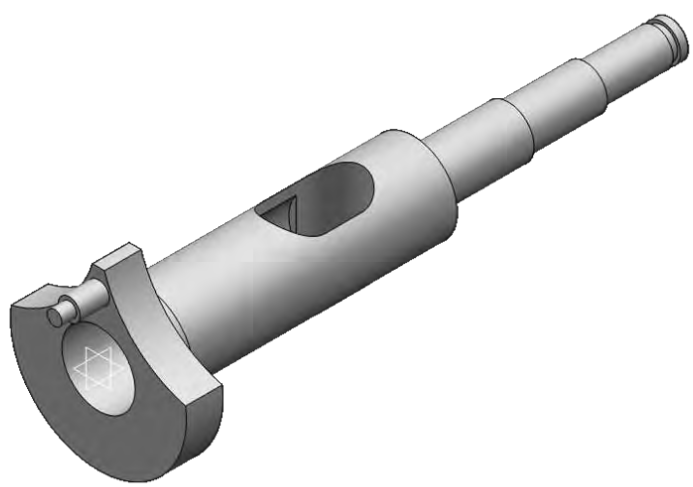
a. Uncheck „**Mirrored extent**"
b. Specify the „**First length**" as [31mm]
c. If necessary use „**Reverse Direction**





17. Hide previously created „**Protected Prism**"
a. Right click „**Protected Prism.5**" > „**Hide/Show**"
b. Repeat this step for „**Protected Prism.6**"





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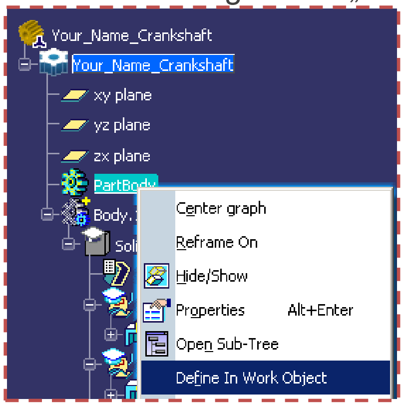
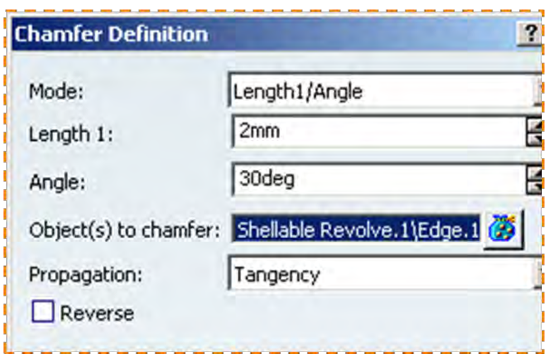
18. Finish the design by „**Dressup**“ Features

a. Right click „**Part Body**“ > „**Define In Work Object**“

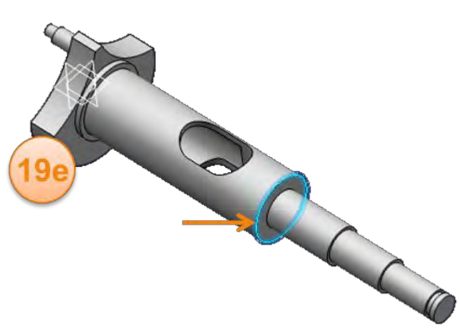


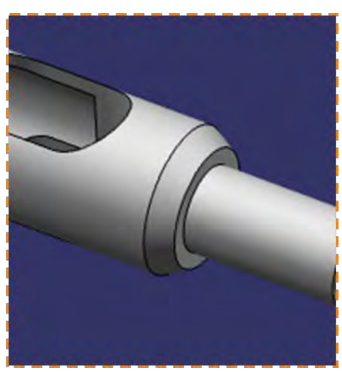
19. We will now perform 2 chamfers to dress up the Crankshaft

a. Click on the „**Chamfer, local modifier**“ icon.
b. Select „**Length1/Angle**“ as „**Mode**“
c. Specify „**Length1**“ as [2mm]
d. Specify „**Angle**“ as [30deg]



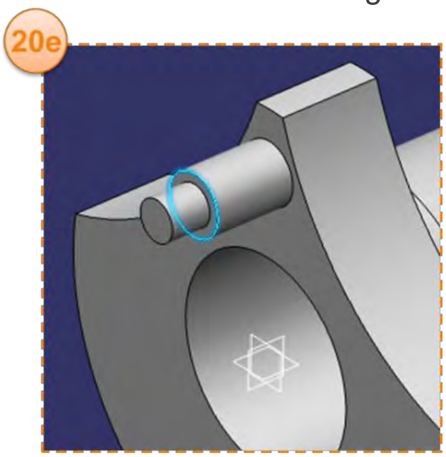
e. Select the edge as shown opposite.

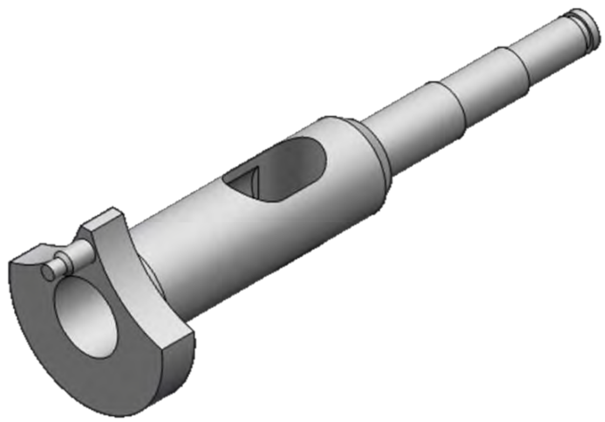




20. Perform the last chamfer to finish the Crankshaft

a. Click on the „**Chamfer, local modifier**" icon
b. Select „**Length1/Angle**" as „**Mode**"
c. Specify „**Length1**" as [0.5mm]
d. Specify „**Angle**" as [45deg]
e. Select the edge as shown

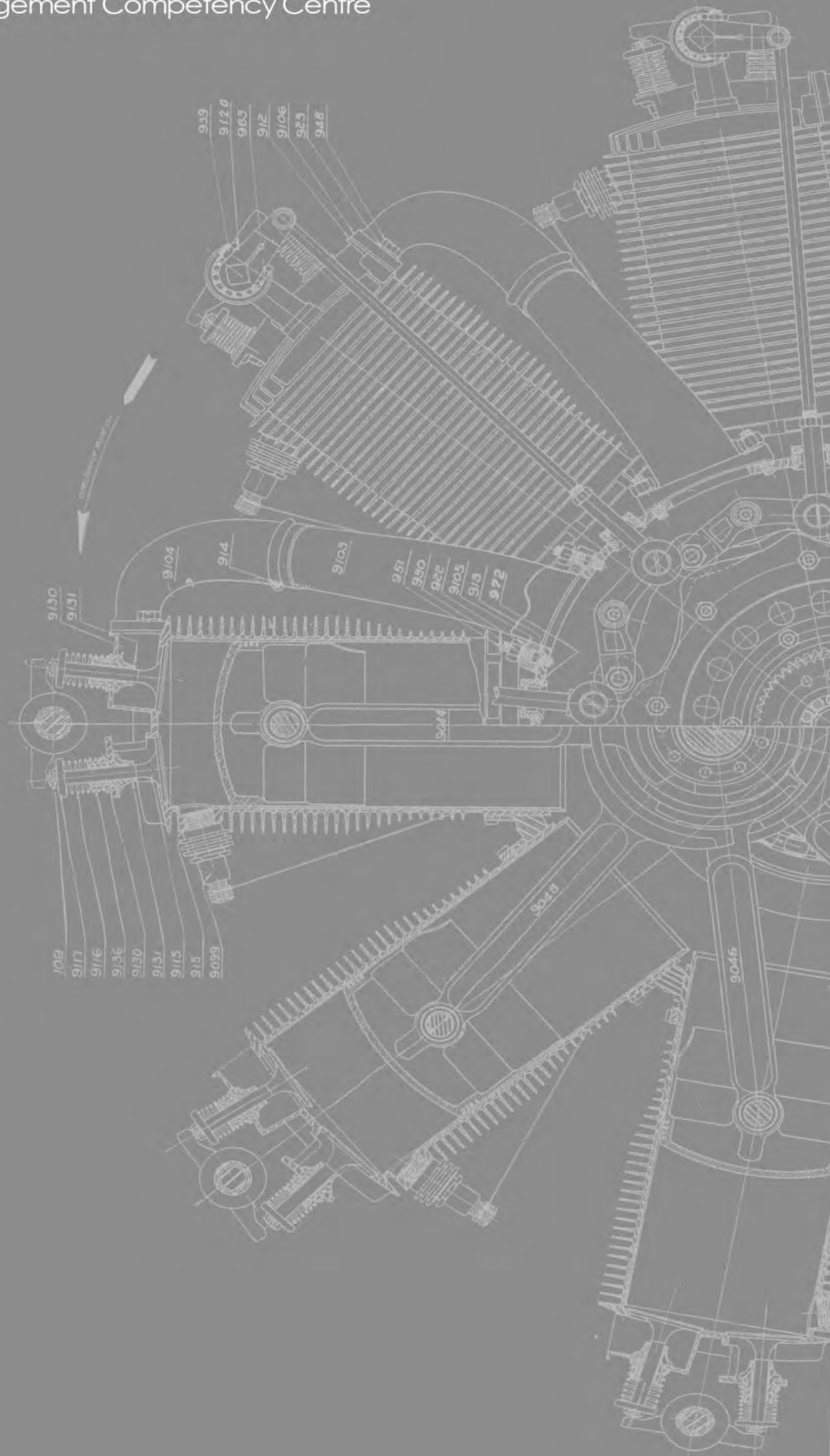




Don't forget to „Propagate" your work to save it in the data base.

PLMCC

Product Lifecycle Management Competency Centre



Assembly Design

DASSAULT SYSTEMES

Cape Peninsula University of Technology

Liberté • Égalité • Fraternité
RÉPUBLIQUE FRANÇAISE

French ministry of education and research

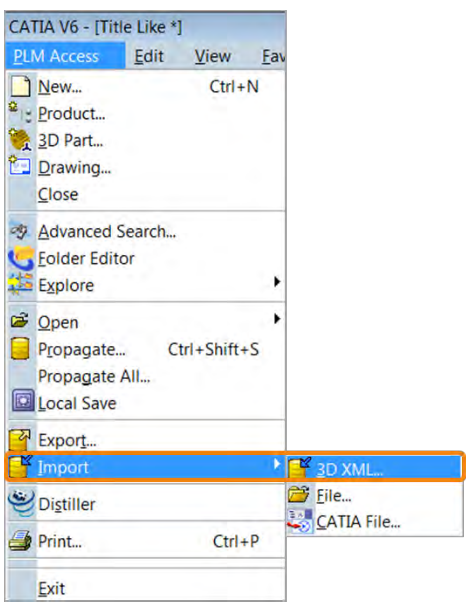
ASSEMBLY DESIGN

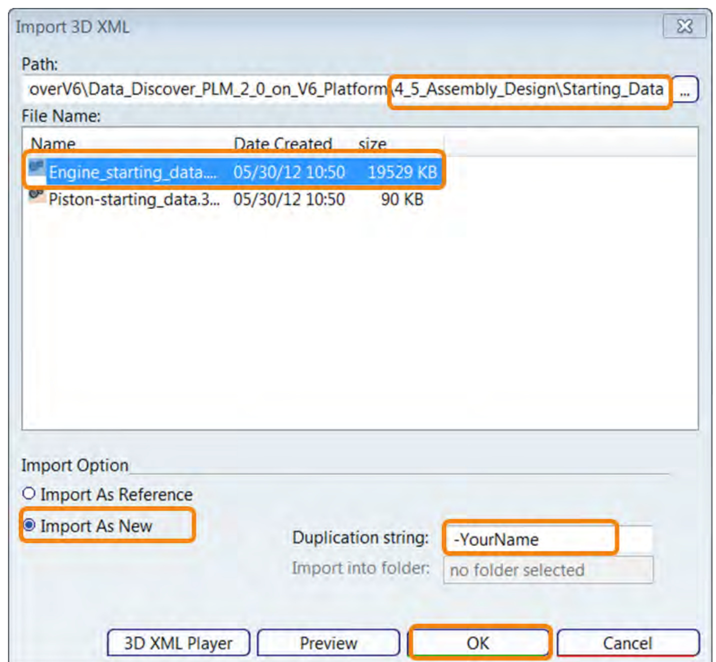
During these steps, you will assemble a section of the micro motor using Assembly Design.

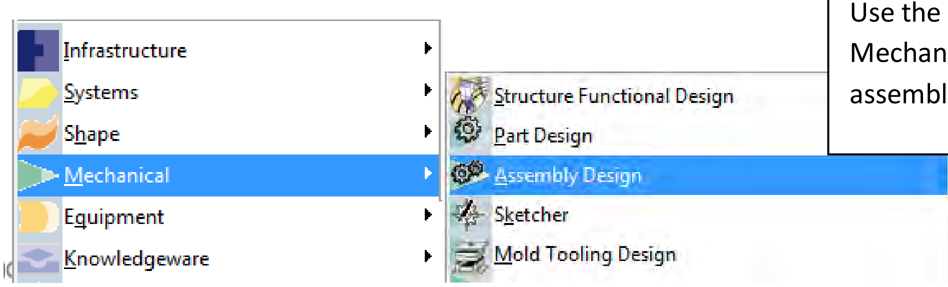
The Assembly Design application allows the design of assemblies with an intuitive and flexible user interface.

1. Import the micro motor 3DXML

a. Click **PLM Access > Import > 3D XML….**
b. You will find it in the **Assembly Design > Starting_Data>Assembly** folder.
c. Select the file named : **Engine_Starting_Data.3dxml**
d. Check **Import As New.**
e. As duplication string enter *„your name_'*
f. Click **OK.**



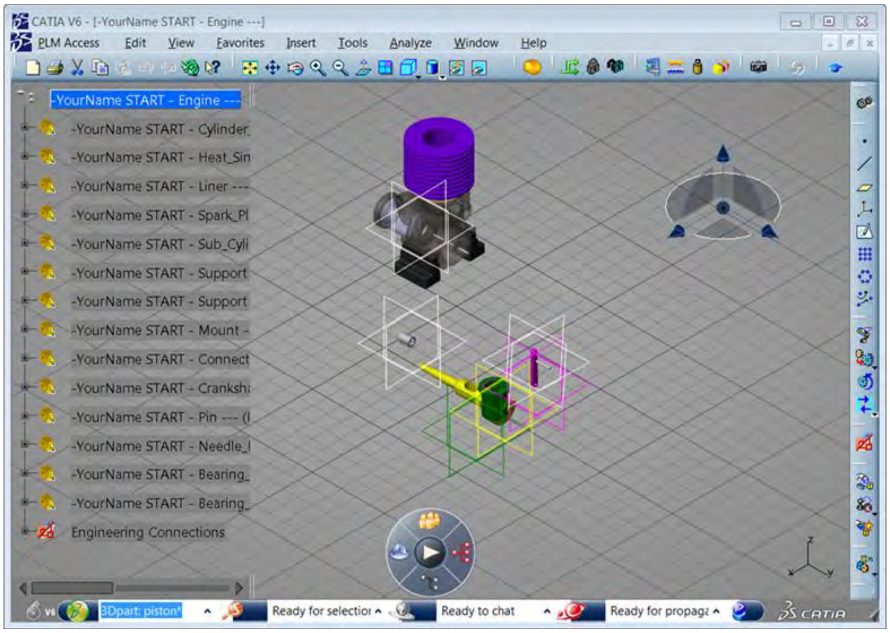




Use the Assembly Design (under Mechanical) Workbench to create the assembly.

2. **The assembly will open in a Navigator window.**

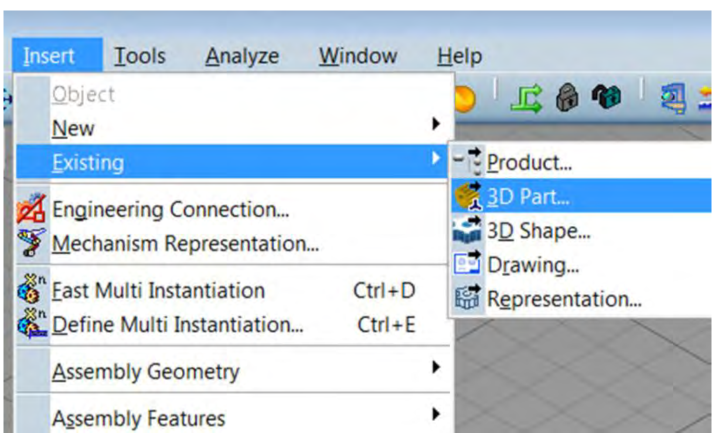
   a. Right click on the file **Your_Name START– Engine** and select **Open**
   b. The Product is opened in an Authoring window as shown below.



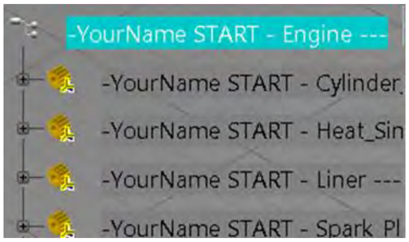
   c. Verify that you are in Assembly Design
   if not:
   Click on Start -> Mechanical-> Assembly Design

3. **Insert the piston into the Product**

   a. Click **'insert' > 'Existing' > '3D Part**"

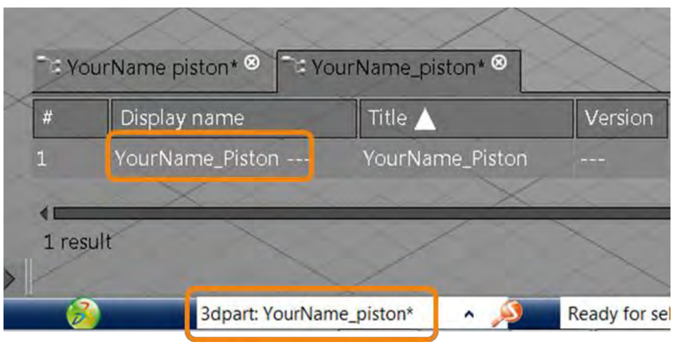


b. Select the root Product **Your_Name START – Engine** in the specification tree.

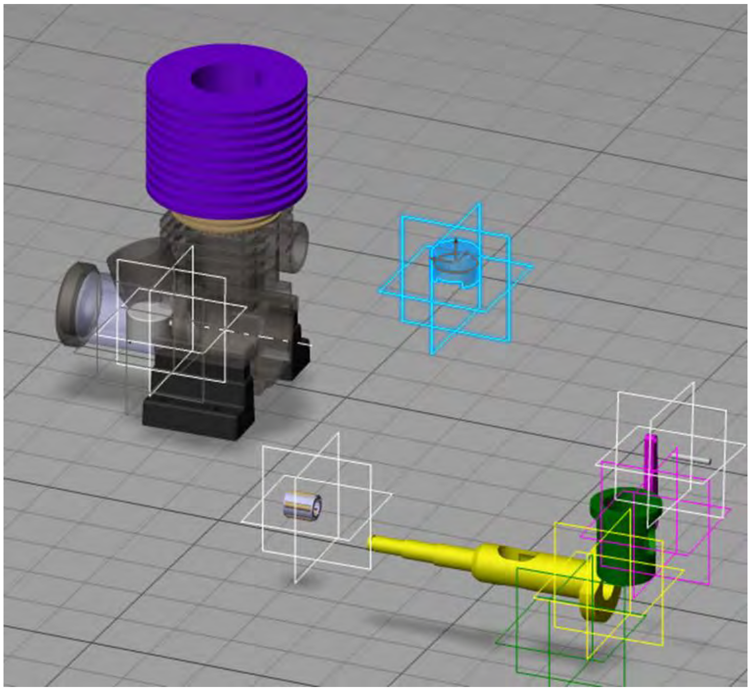


c. A dialogue box appears prompting the use of the search bar to select an existing 3D part.

d. Enter „*3dpart: YourName_piston**" in the search bar.
e. Click Search.
f. Double click on the matching search result.

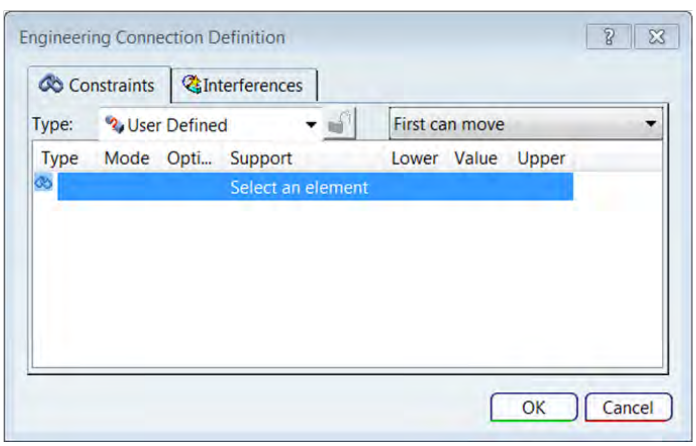


g. Place the piston in the assembly space.
h. Click **OK** on the **Engineering Connection Definition** dialogue box.
i. Close the search results table.

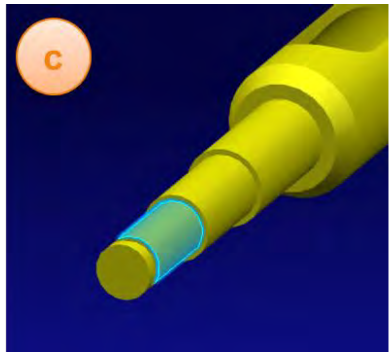


4. Assemble the crankshaft and the needle bearing.

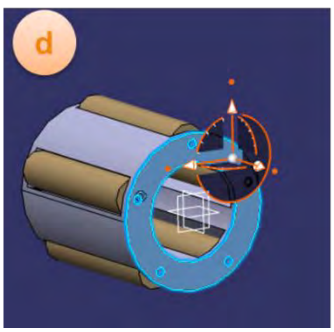
a. Click on the **Engineering Connection** icon.
A panel will open.

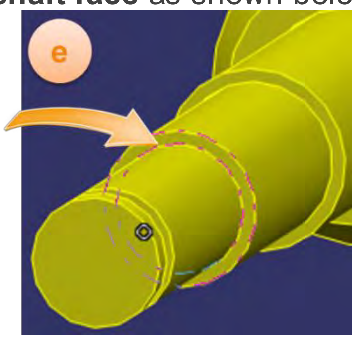


b. Click on the **Surface** of the needle bearing to select its **axis**.
c. Click on the **Surface** of the crankshaft to select its **axis.**



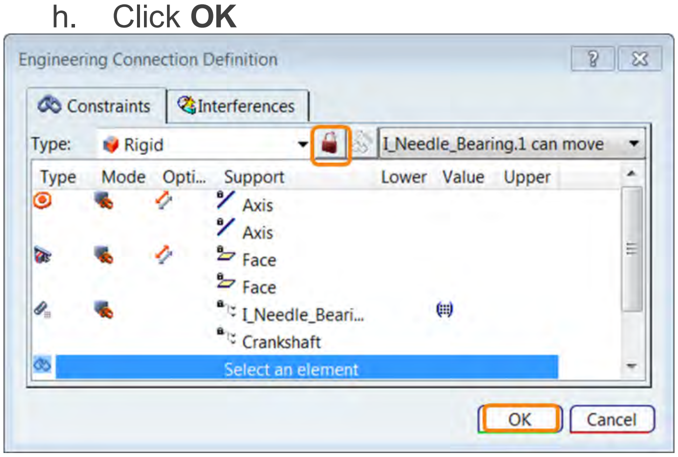
d. Click on the **needle bearing face** as shown below
e. Click on the **crankshaft face** as shown below

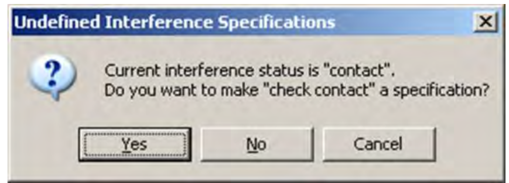




f. Select the the crankshaft and the needle bearing in the **specification tree.**

g. The type of engineering connection has been changed to **Rigid**, click on the **unlocked padlock** to fix the engineering connection as Rigid.
h. Click **OK**

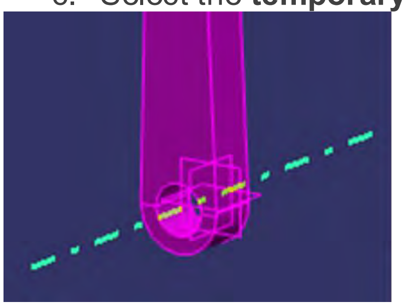


i. Then click **Yes** to validate the interference. 

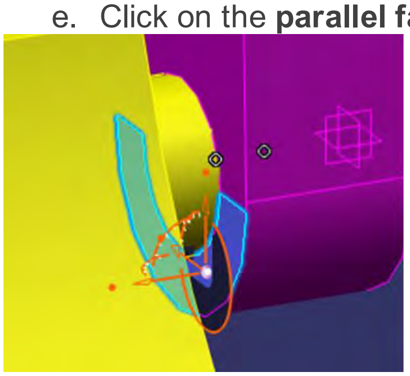
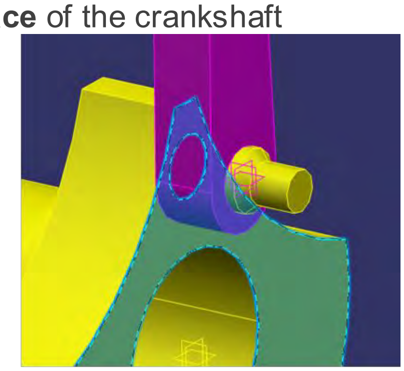
5. Assemble the Crankshaft and the connecting rod

a. Click on the **Engineering connection** icon. 

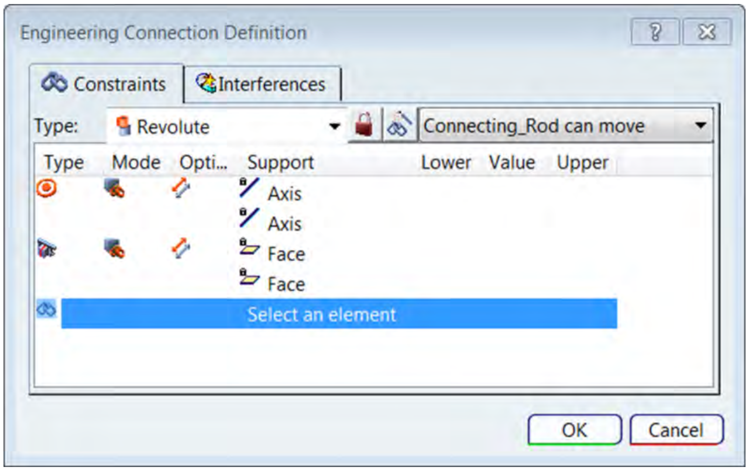
b. Click on the **temporary axis** in the hole of the connecting rod.
c. Select the **temporary axis** of the rotational axis of the crankshaft




d. Click on the **face** of the connecting rod
e. Click on the **parallel face** of the crankshaft

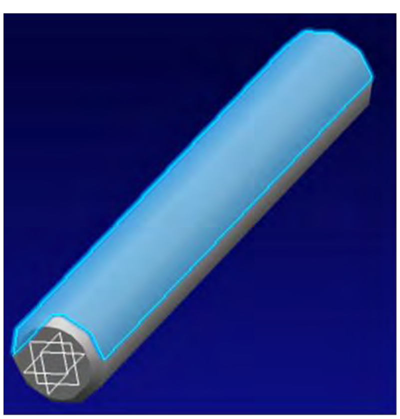
f. Click on the unlocked padlock
g. Click **OK** to close the pannel

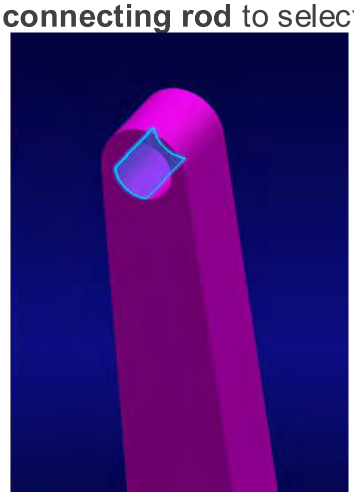


h. Click **Yes** to validate the interference

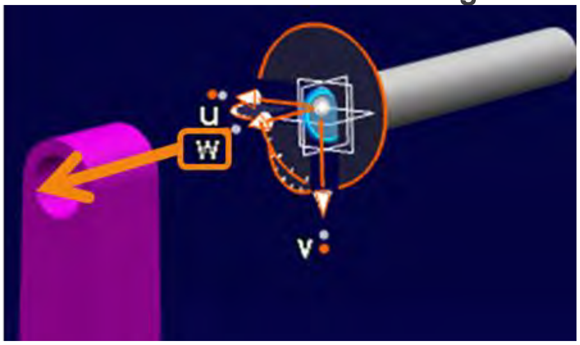
6. Create a cylindrical joint with the pin and the connecting rod

a. Click on the **Engineering connection** icon
b. Click on the **Surface** of the pin to select its axis.
c. Click on the **Surface** of the **connecting rod** to select its axis.

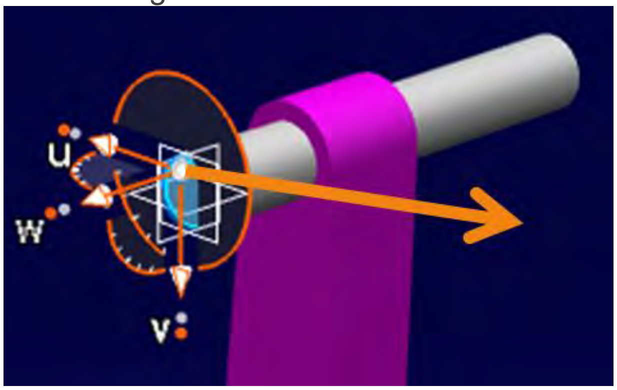




d. Take the **Robot** and put it on the surface of the **Pin.**
e. When the **Robot** is on the surface, pull the **axis w** to move the **pin** between the two surfaces of the **connecting rod.**

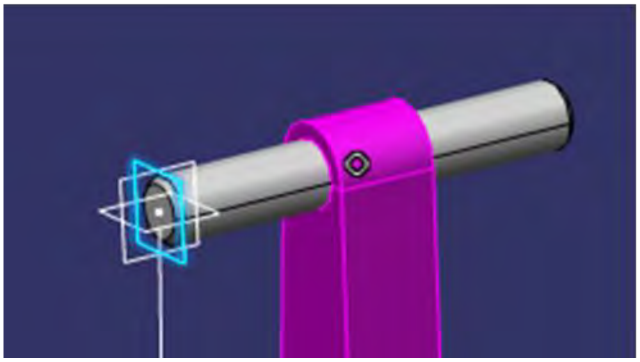


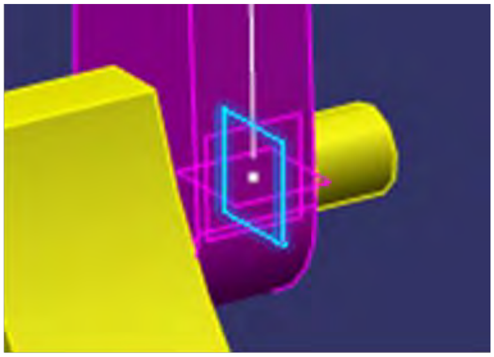
f. After having moved the **Pin,** remove the robot by dragging it off the part to the background.



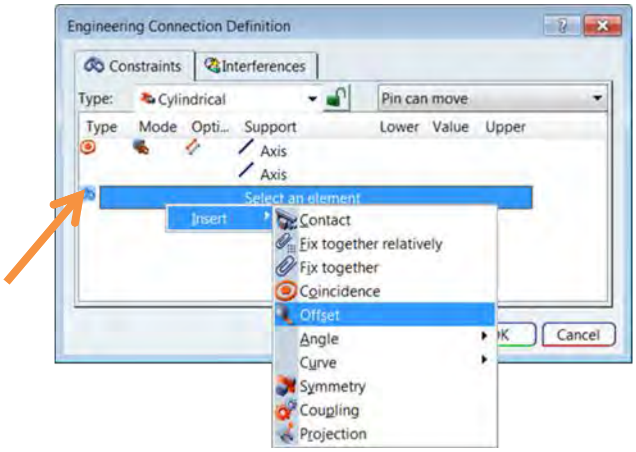
7. Create a revolute joint the pin and the connecting rod (to remove)

a. Click on the **first plane** of the pin as shown opposite.
b. Click on the **parallel plane** of the connecting rod as shown opposite.

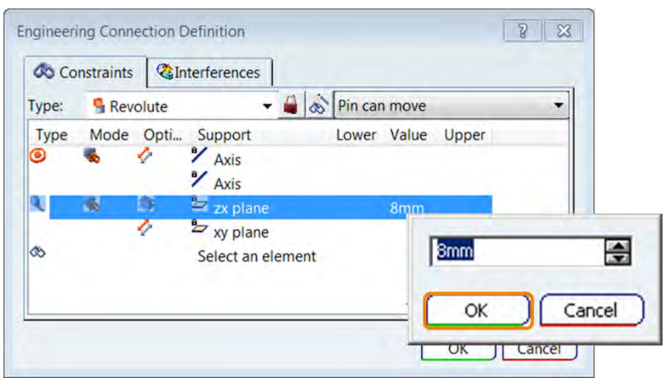




c. Right click on the connection type and select **Offset.**

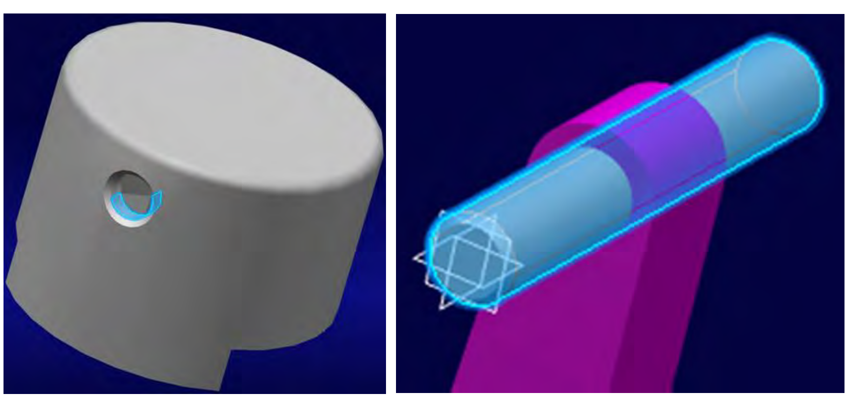


d. Double click on the value written and type **8mm** in the panel. Click **OK**
e. Click on the unlocked **padlock**
f. Click **OK** and **Yes** to close the panel and validate the interference

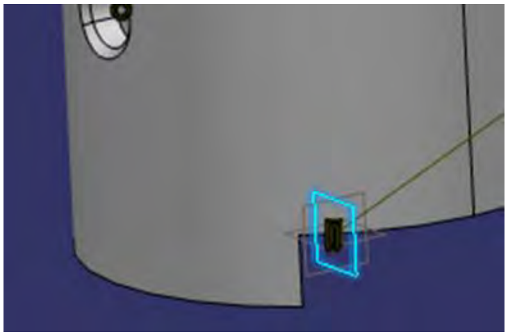


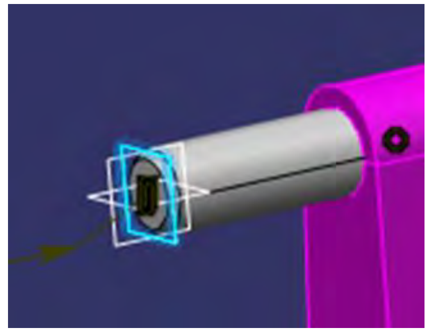
8. Create a rigid joint between the piston and the pin

a. Click on the surface of the **piston hole** and the **Pin**

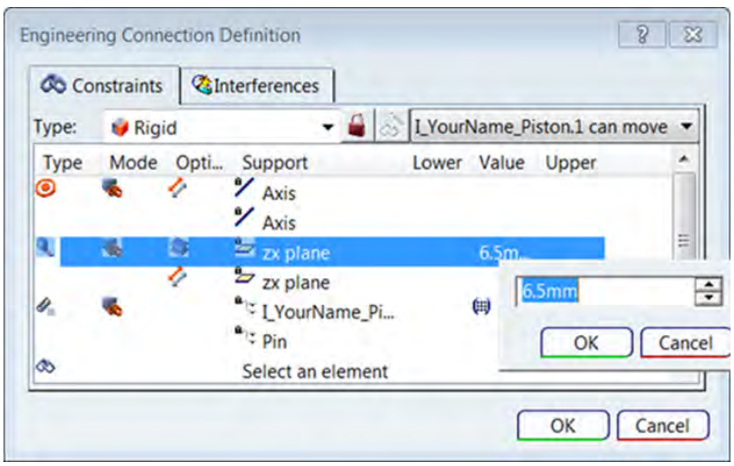


b. Select the **plane** of the piston as shown.
c. Select the **parallel plane** of the pin as shown.

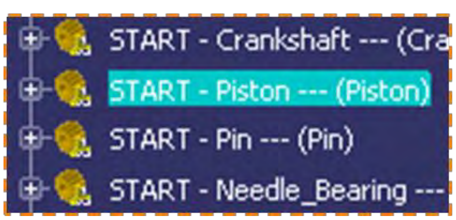




d. Right click on the connection type and select **Offset.**
e. Change the offset value to **6.5** in the panel. Click **OK**



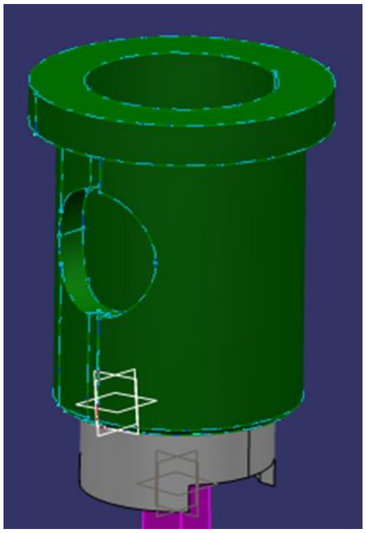
f. Click on the piston and the pin product in the specification tree .The Engineering connection type has been changed to **rigid.**



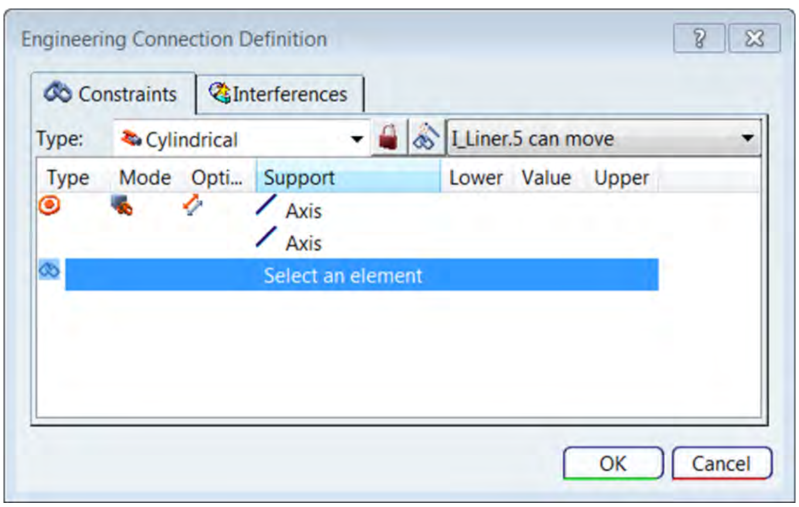
g. Lock the padlock and click **OK**

9. Create a cylindrical joint between the liner and the piston

a. Create an **axis coincidence** between the axis of the liner and the piston

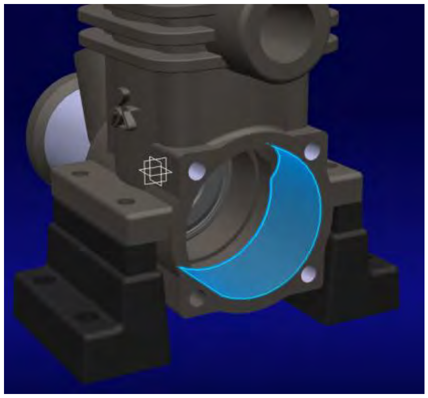


b. Lock the padlock
c. Click **OK**

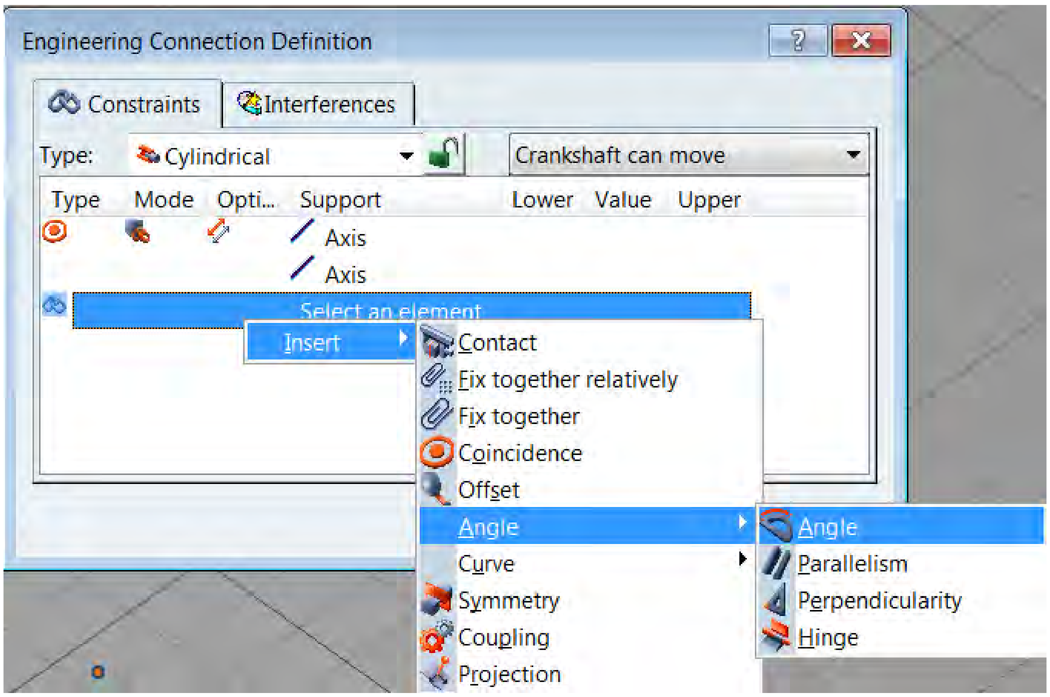


10. Create a cylindrical joint between the crankshaft and the mount

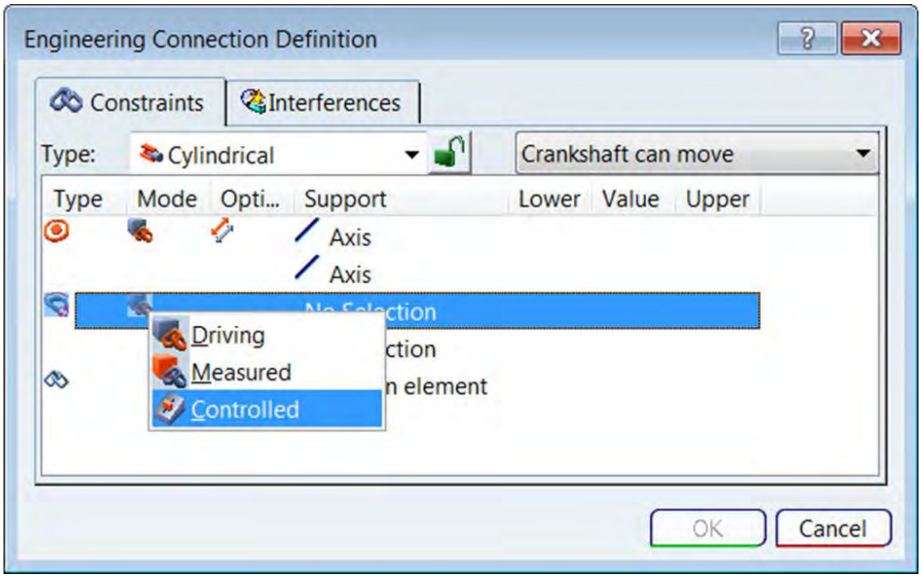
a. Create an **axis coincidence** between the axis of the liner and the piston



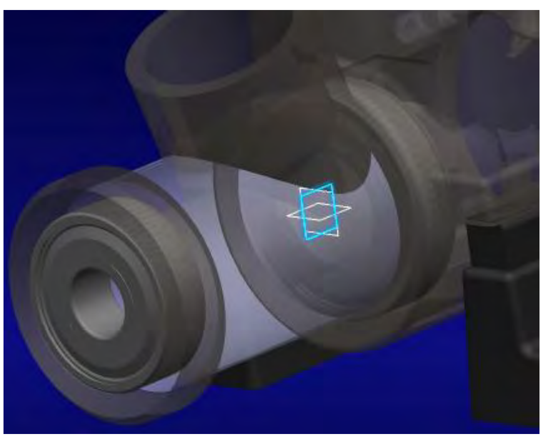
b. Right click as shown below and select „**Insert**“>’**Angle**“>’**Angle**“

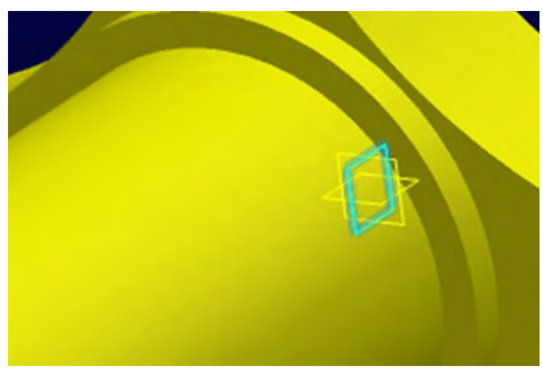


c. Right click the connection mode as shown below and select **Controlled.**



d. Select the **two parallel planes** of the **crankshaft** and the **mount**, as shown below.



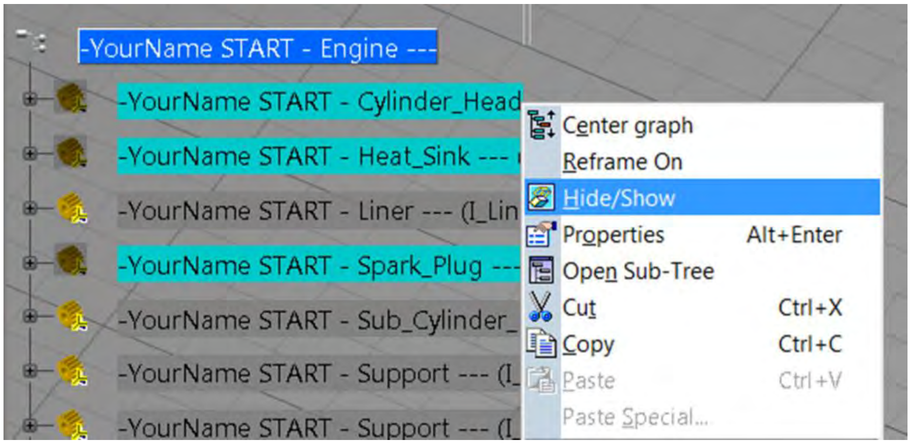


e. Validate the Engineering connection

f. Don't forget to **Update.** 

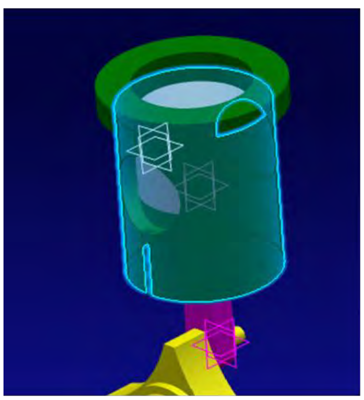
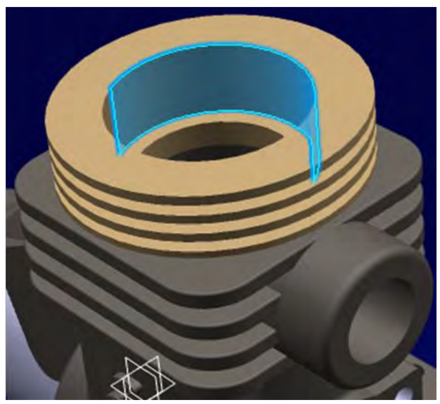
11. Hide the Cylinder_head, the Heat_sink, and the Spark_plug.

a. Select the Cylinder_Head, the Head_Sink and the Spark_Plug.
b. Right-click and select Hide/Show.

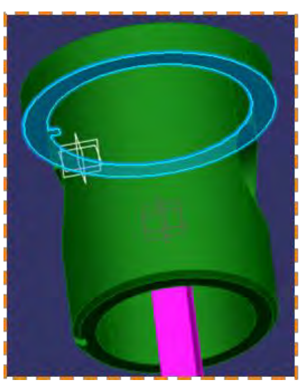
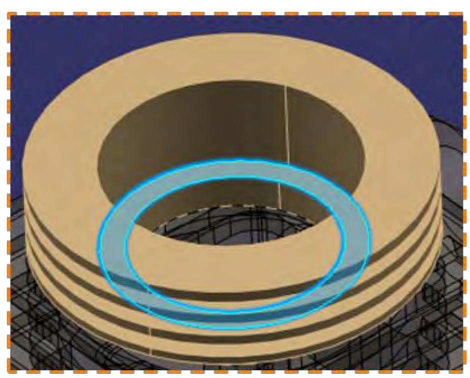


12. Create a rigid joint between the 'liner' and the Sub_Cylinder_head

a. Create an axis coincidence between the liner and the **Sub_Cylinder_head.**

b. Create a planar coincidence between the liner and the **Sub_Cylinder_head.**

c. Select the 2 products concerned to make a **rigid** joint.

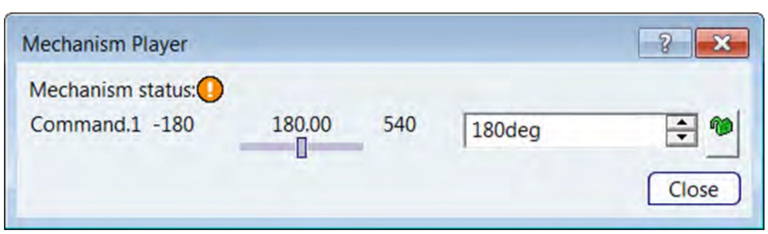
d. Validate the Engineering connection

13. Create a mechanism Representation

a. Click on **Mechanism Representation** to create a virtual mechanism of your micro motor.

b.

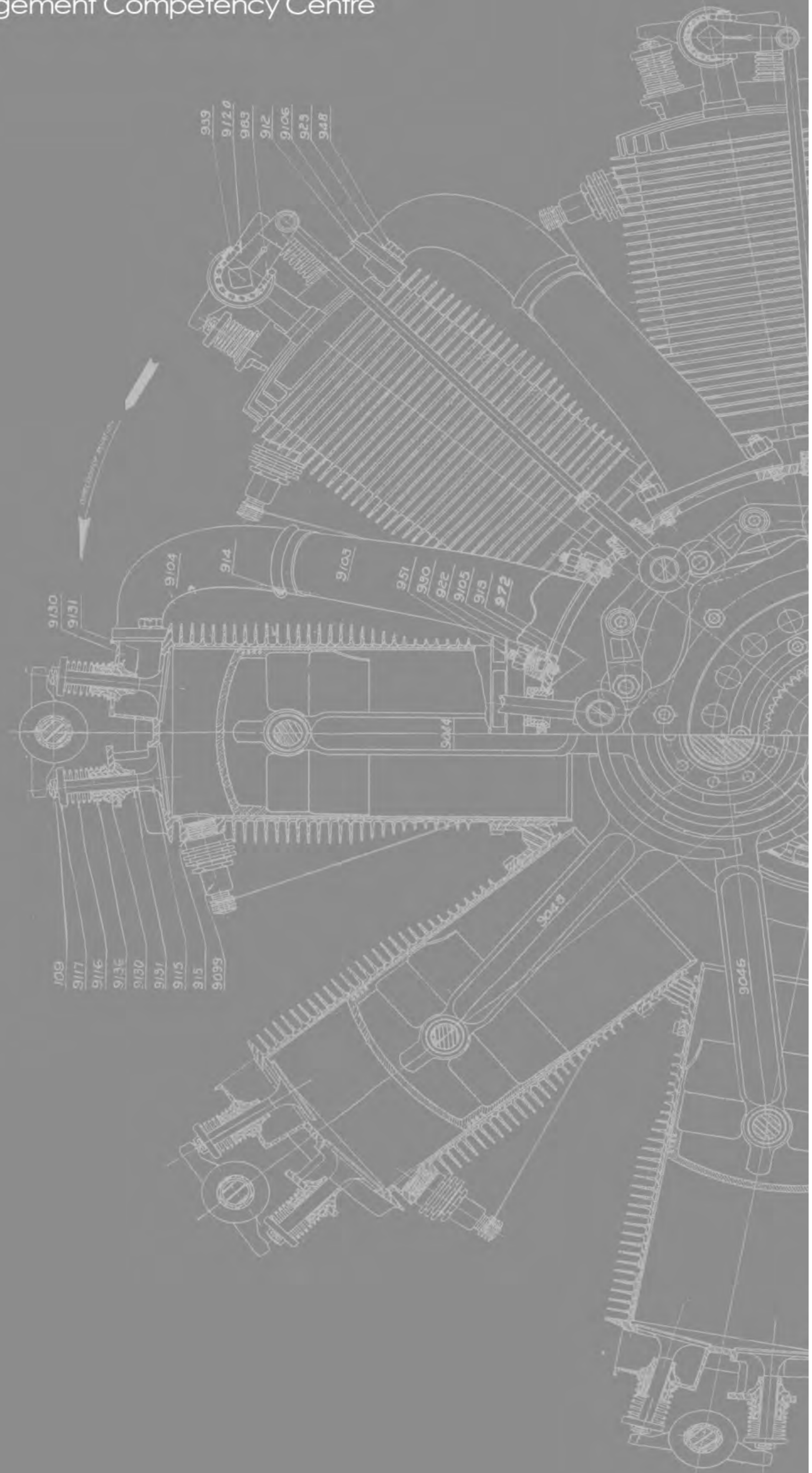
c. Simulate by clicking on **Mechanism Animation**



Don't forget to 'Propagate' your work to save it in the data base

PLMCC

Product Lifecycle Management Competency Centre



Assembly Design

Exercises

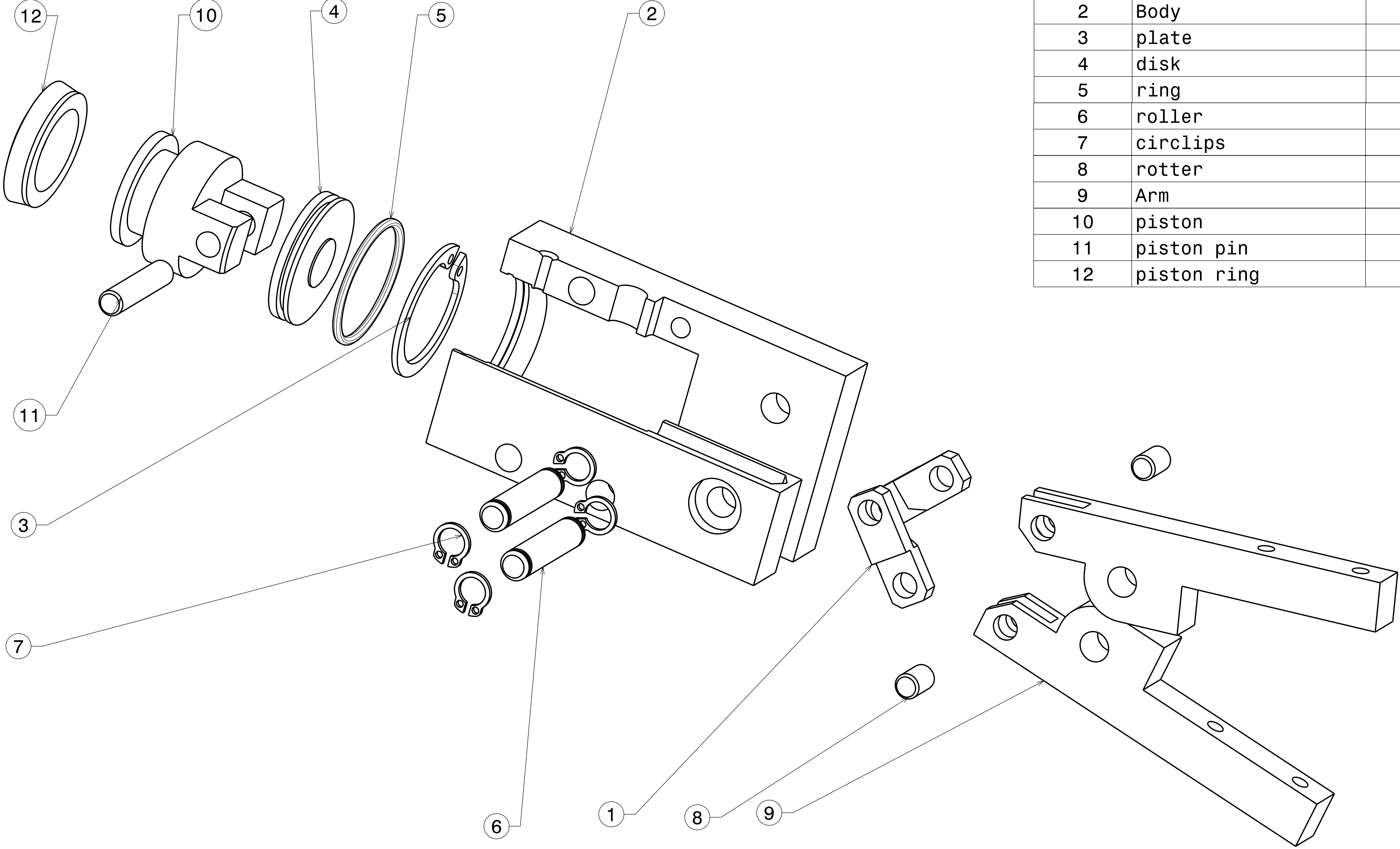
DASSAULT SYSTEMES

Cape Peninsula University of Technology

Liberté • Égalité • Fraternité
RÉPUBLIQUE FRANÇAISE

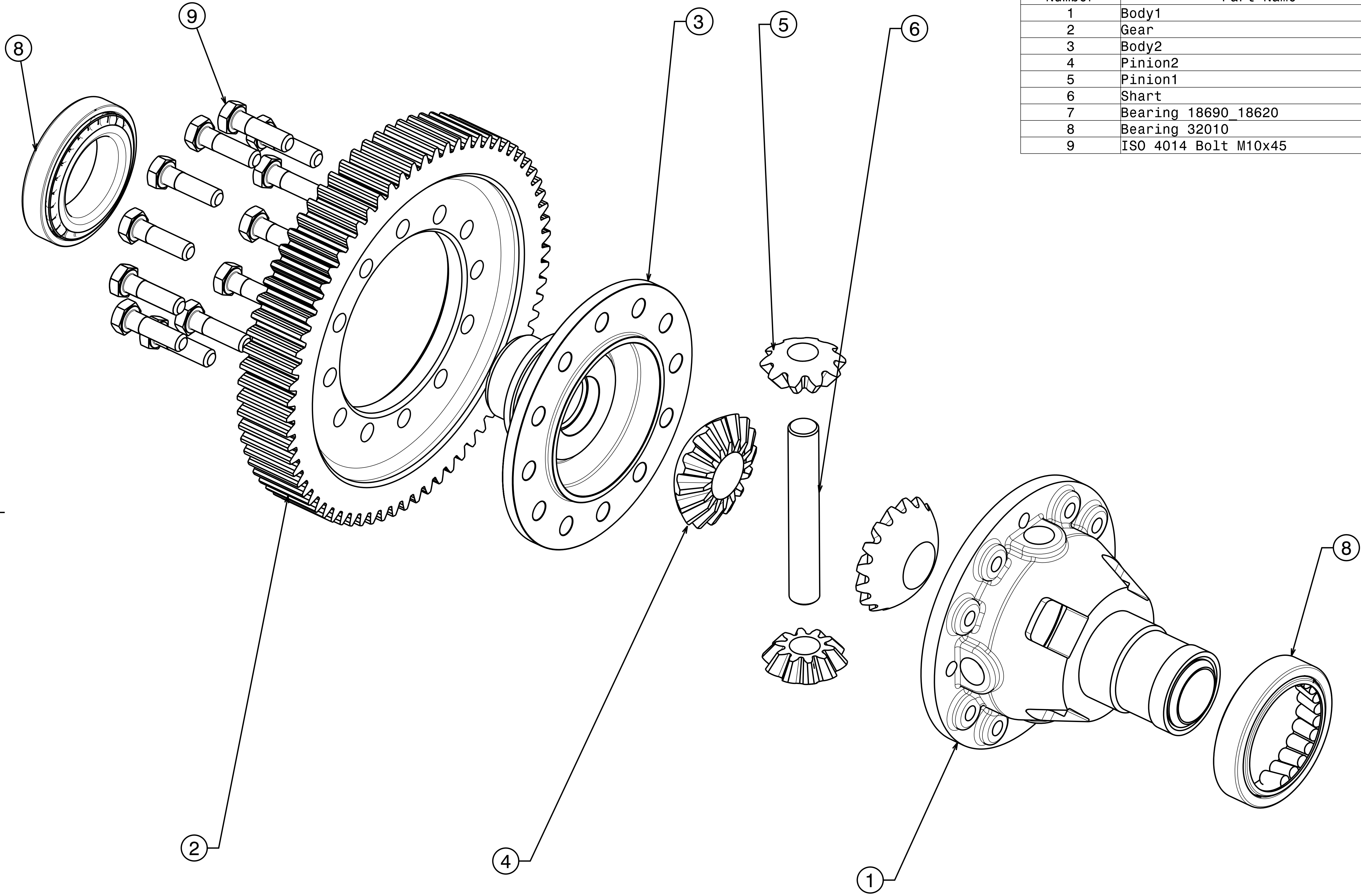
French ministry of education and research

Number	Part Name	Quantity
1	connector	2
2	Body	1
3	plate	1
4	disk	1
5	ring	1
6	roller	1
7	circlips	2
8	rotter	1
9	Arm	1
10	piston	1
11	piston pin	1
12	piston ring	1

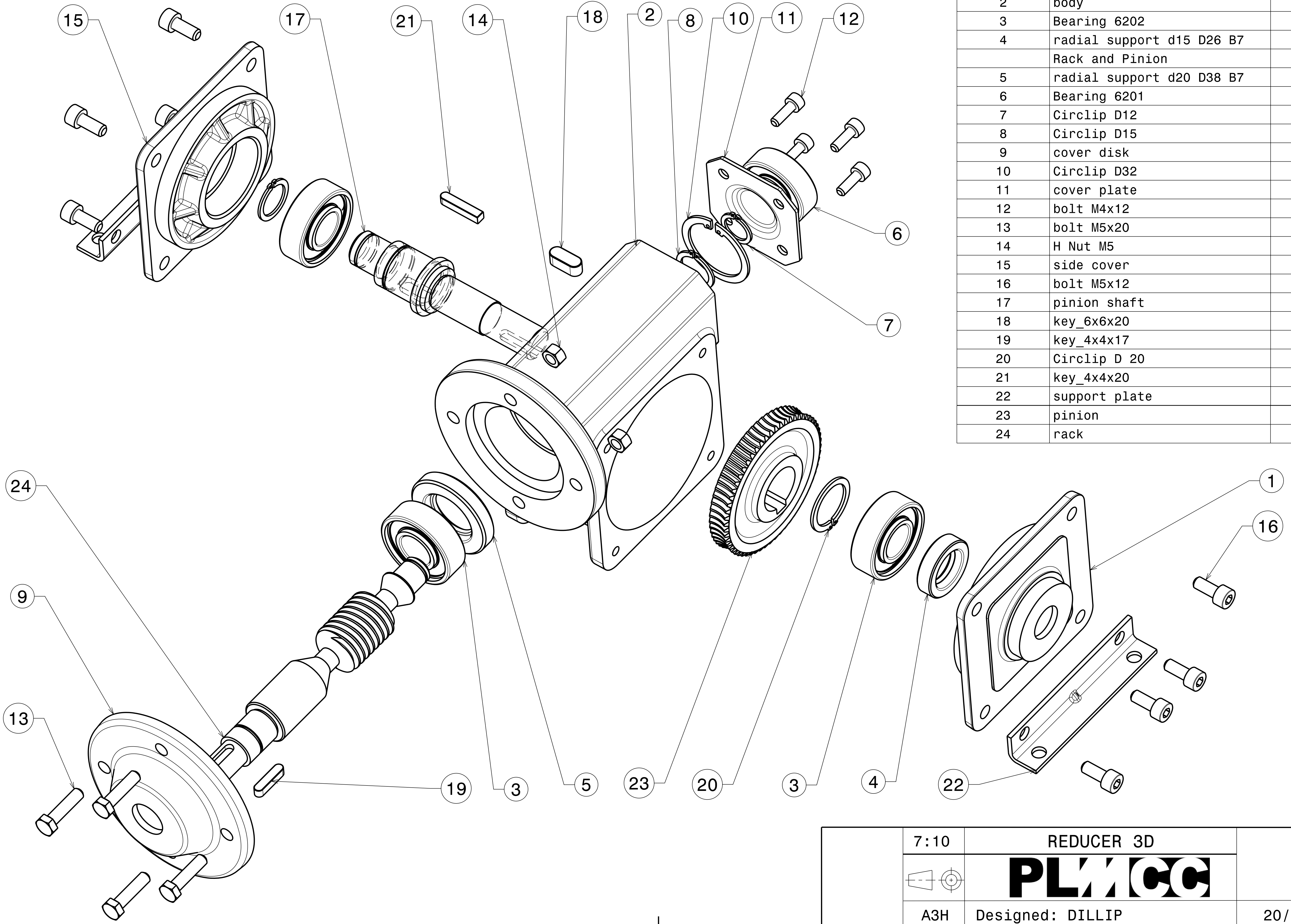


7:5	GRIPPER	
A3H	Designed: DILLIP	20/03/07

Number	Part Name	Quantity
1	Body1	1
2	Gear	1
3	Body2	1
4	Pinion2	2
5	Pinion1	2
6	Shart	1
7	Bearing 18690_18620	1
8	Bearing 32010	1
9	ISO 4014 Bolt M10x45	13

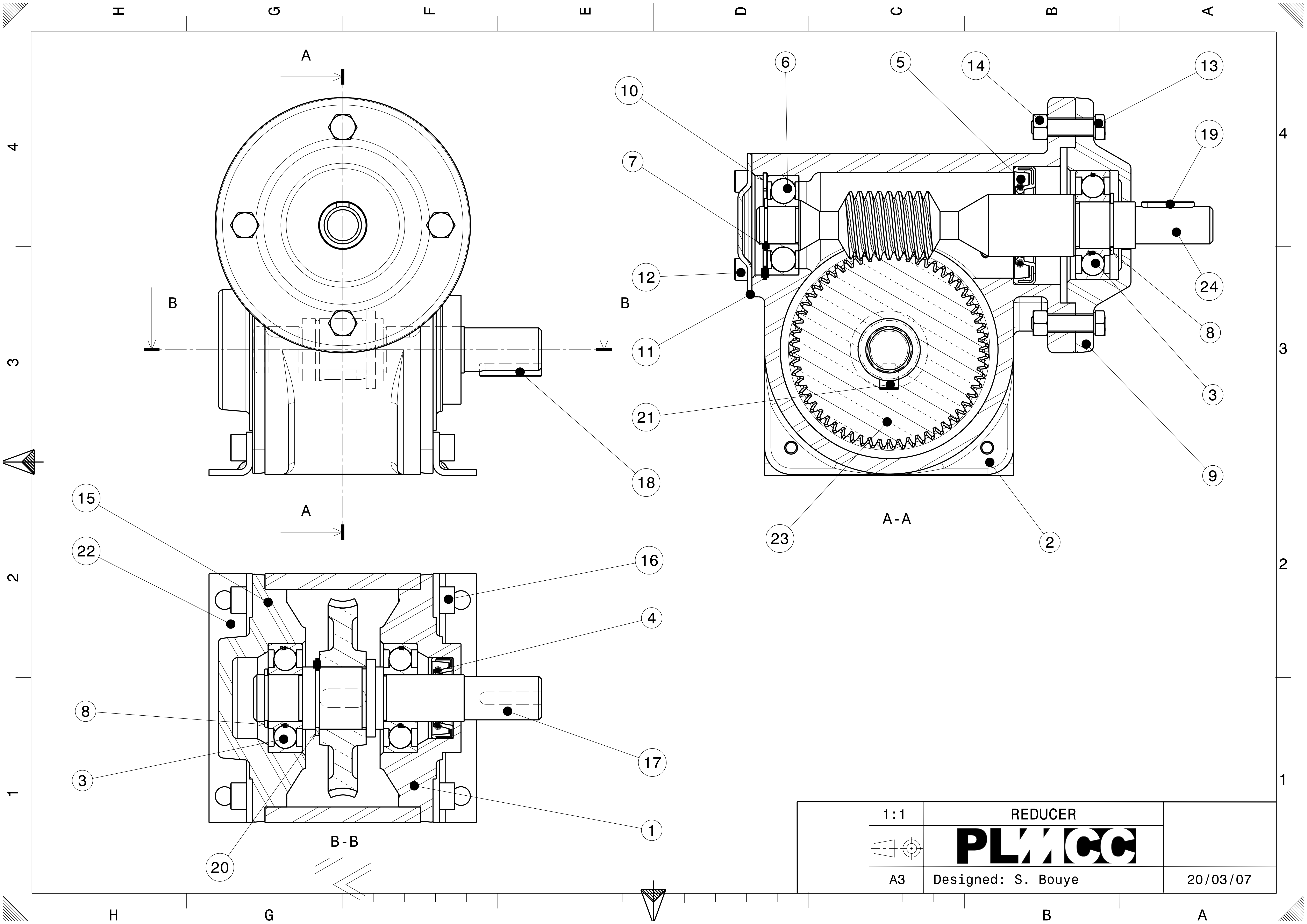


1:2	DIFFERENTIAL	
	PLMCC	
A3H	Designed: DILLIP	20/03/07



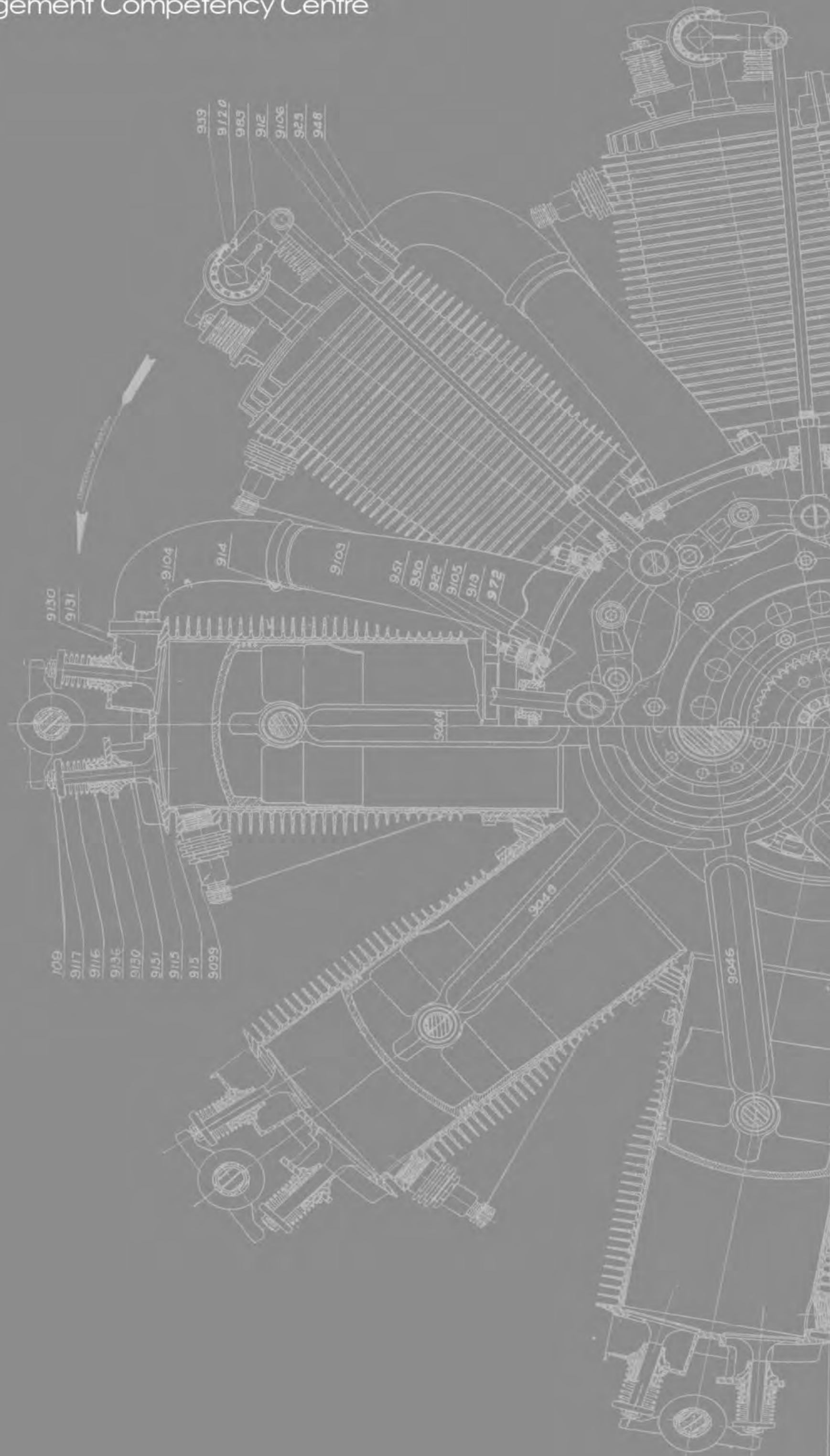
Number	Part Number	Quantity
1	side plate	1
2	body	1
3	Bearing 6202	3
4	radial support d15 D26 B7	1
	Rack and Pinion	1
5	radial support d20 D38 B7	1
6	Bearing 6201	1
7	Circlip D12	1
8	Circlip D15	2
9	cover disk	1
10	Circlip D32	1
11	cover plate	1
12	bolt M4x12	4
13	bolt M5x20	4
14	H Nut M5	4
15	side cover	1
16	bolt M5x12	8
17	pinion shaft	1
18	key_6x6x20	1
19	key_4x4x17	1
20	Circlip D 20	1
21	key_4x4x20	1
22	support plate	2
23	pinion	1
24	rack	1

7:10	REDUCER 3D	
	PLMCC	
A3H	Designed: DILLIP	20/03/07


A
B
B
A
A-A
B-B
1
2
3
4
5
6
7
8
9
10
11
12
13
14
15
16
17
18
19
20
21
22
23
24
1:1
REDUCER
A3
Designed: S. Bouye
20/03/07

PLMCC

Product Lifecycle Management Competency Centre



Kinematic Analysis

DASSAULT SYSTEMES

Cape Peninsula University of Technology

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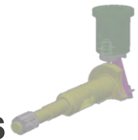
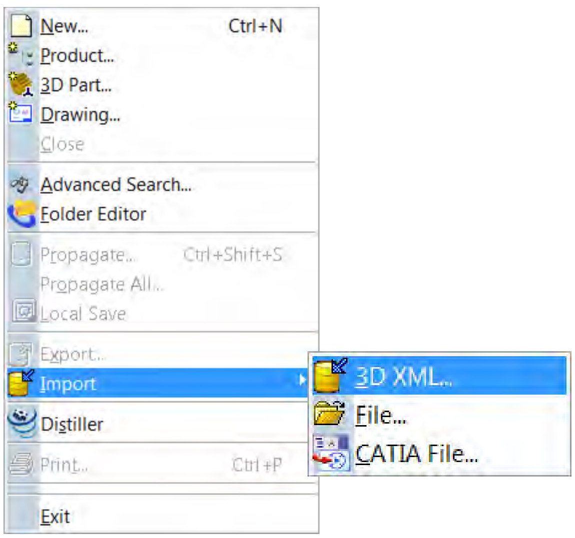
French ministry of education and research

KINEMATIC ANALYSIS 

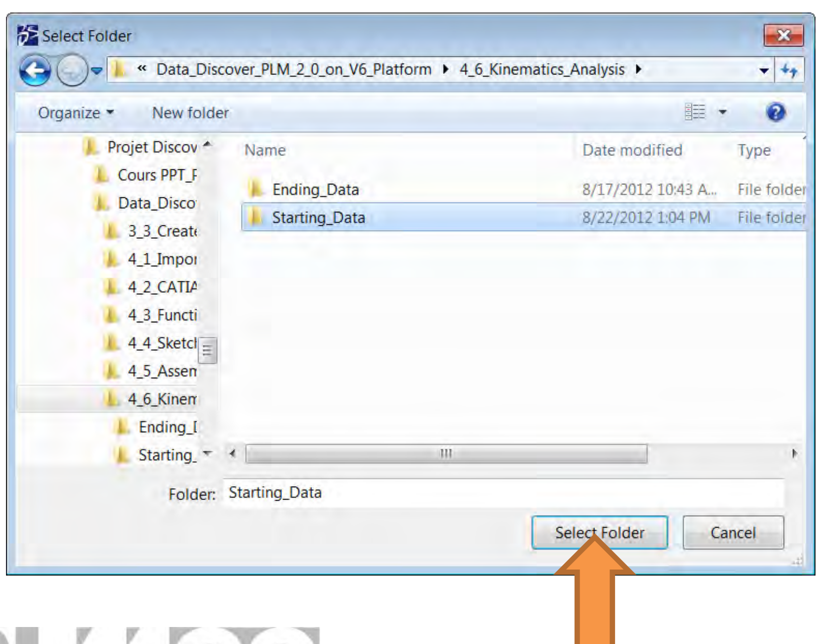
During these steps, you will simulate the movement of the piston and the crankshaft using kinematics analysis.
Kinematics Simulation provides a set of tools to simulate the motions of assembly mechanisms.

1. **Import the 3D XML file**

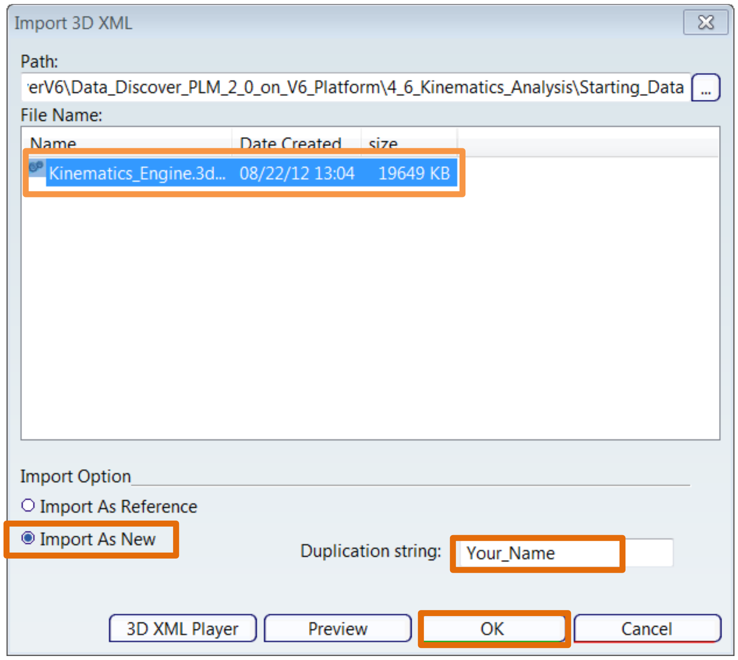
   a. Click **PLM Access > Import > 3D XML….**



   b. You will find it in the **Kinematics_Analysis** folder.
   c. Click on folder named **Starting Data**, and click **Select Folder**



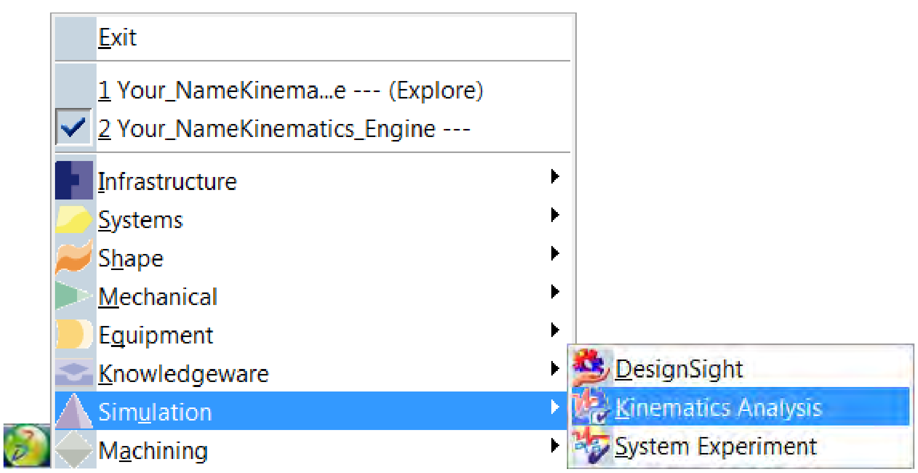
d. Select **Kinematics_Engine.3D XML** file to import.
e. Check **Import As New.**
f. As duplication string enter **Your_Name**
g. Click **OK.**



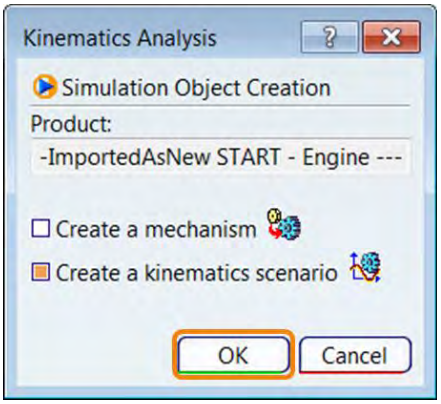
h. A navigator window opens.
i. Right click on the product"s name at the top of the Specification tree and select Open. The product is transferred to an authoring window.

2. Access the workbench

a. Click on the **start** button
b. Select **Simulation > Kinematics Analysis**

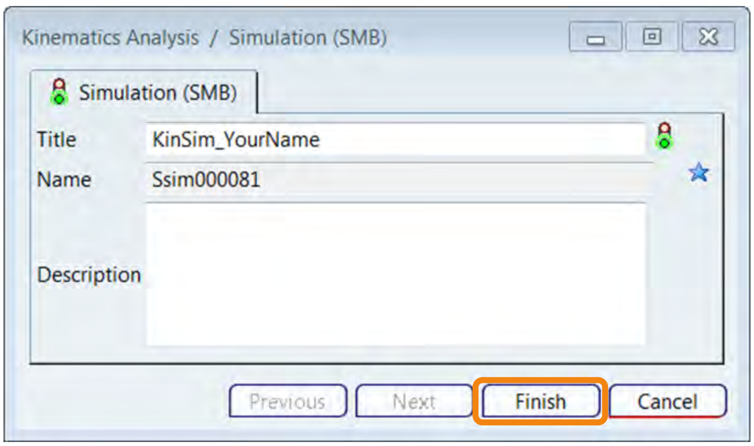


c. Select **Create a kinematics scenario** and click **OK** to the **Kinematics Analysis** dialogue box.



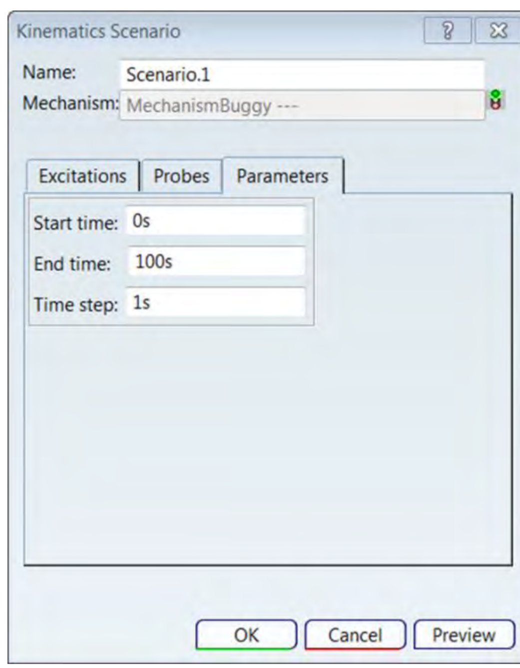
d. Change Title (KinSim_YourName)

e. Click **Finish**



3. Create a simulation

a. The Kinematics Scenario window is open.
b. Enter the **name** of the scenario
c. In the field **End time** enter **100s**
d. Click **OK**



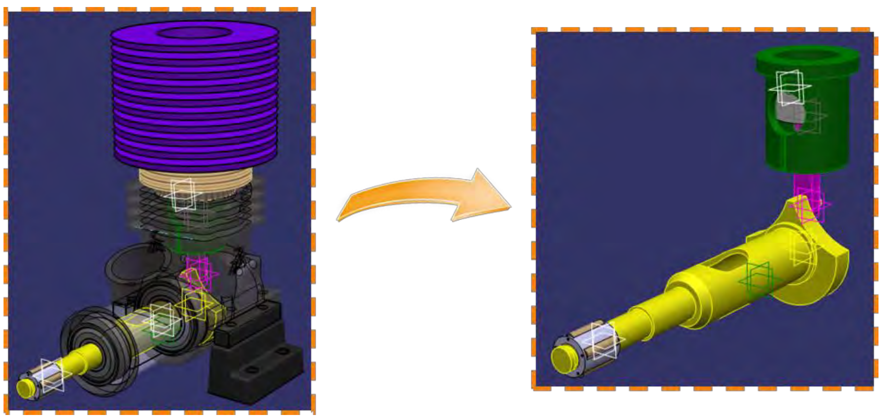
A **Simulation** has been created.

PLMCC

4. Hide parts

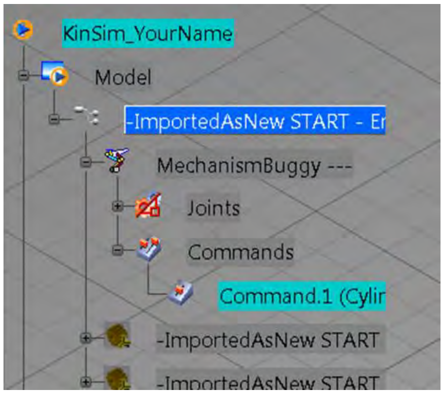
a. Hide the following parts by multi selecting them and right clicking Hide/Show :

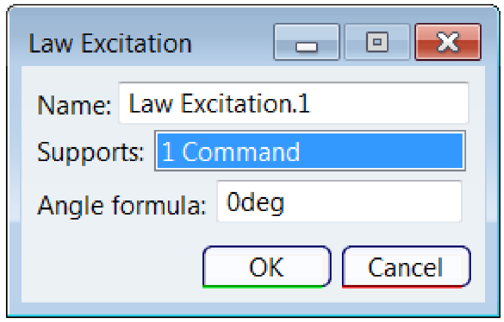
Cylinder_Head
Heat_Sink
Spark_Plug
Sub_Cylinder_head
Support
Support
Mount
Bearing_D24
Bearing_D19



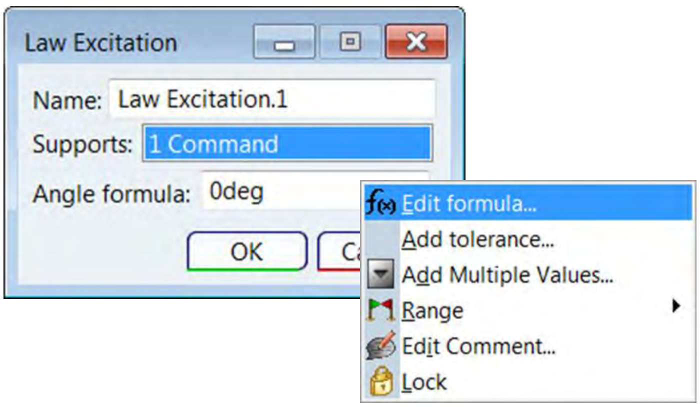
5. Create a law excitation

a. Click on the **law excitation** icon.
b. Select **Command.1** under **Commands** in the Specification tree as **Supports**.





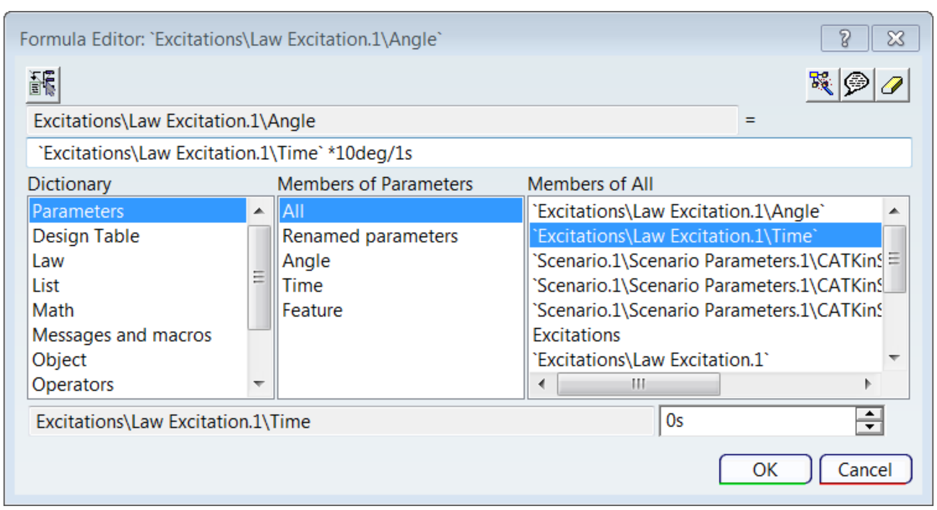
c. Right Click on the **Angle formula** and select **Edit formula**



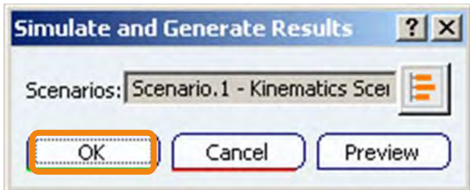
d. Create a formula by double clicking on in the „Members of All" column **`Excitations\Law Excitation.1\Time`**

e. Write ***10deg/1s** with the keyboard to complete the formula.

f. Click **OK** in the formula editor and **OK** in the law excitation panel

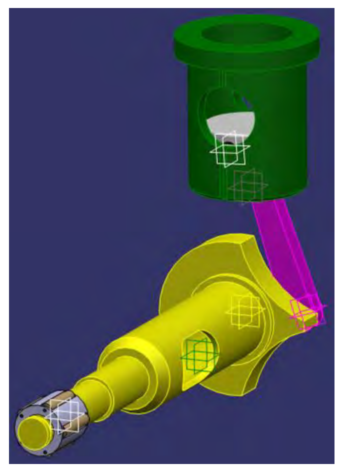


g. Click **simulate and generate results** icon
h. Click **OK** and wait till the end of the calculation.



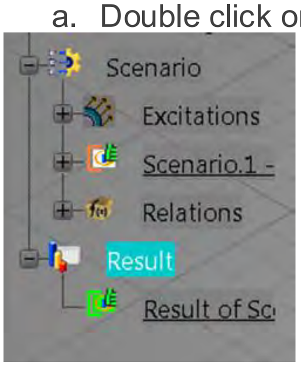
6. View the results of the simulation

a. Click on the **play button** of the compass to see the motion of the piston
b. You can control the **motion** of the kinematics by clicking and holding the cursor at the top of the compass



7. Detect the Clash

a. Double click on **Result** in the Specification tree.



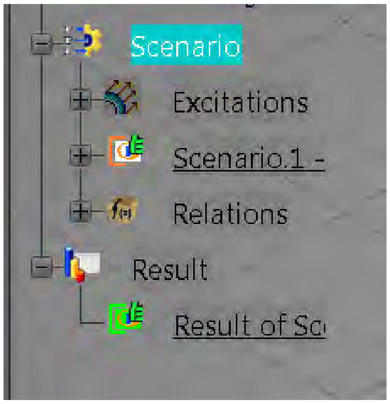
b. Click on the **clash detection** icon

c. The new toolbar will appear.
Click **clash detection (on)**

d. Click on the **play button** of the compass. The kinematics will highlight the parts that are moving with clashes

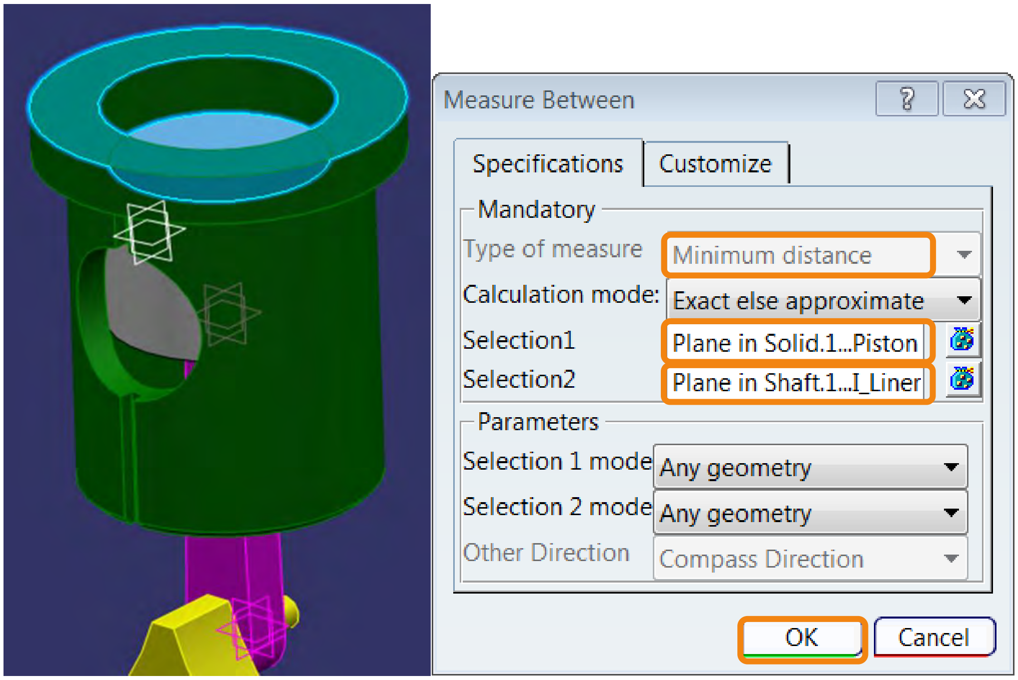
8. Create Measure probes

a. Result is activated, reactivate scenario with a double click.



b. Click on **measure probe** icon 

c. In the specifications tab, select **Minimum distance**
d. Select the face at the **top of the piston**
e. Select the face at the **top of the Liner**
f. Click **OK**

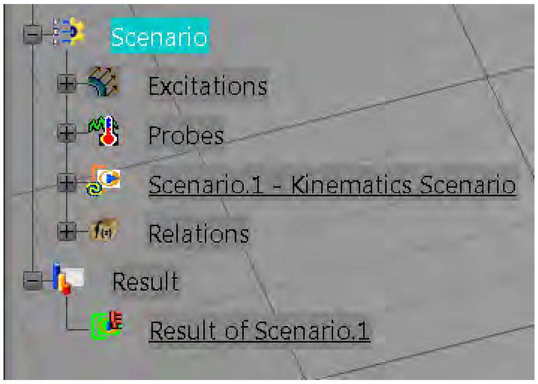


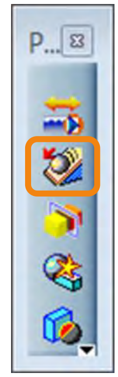
9. Create a point that will be used for Speed and acceleration

   a. Double click on the **piston**, the workshop will change
   b. Select the **point** icon
   c. Create a point anywhere on the **top face** of the piston
   d. Click **OK**

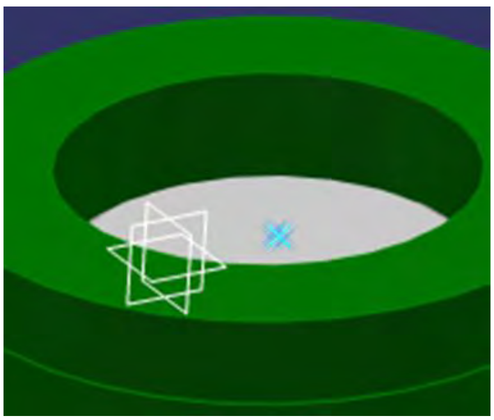
10. Create Speed and acceleration measure

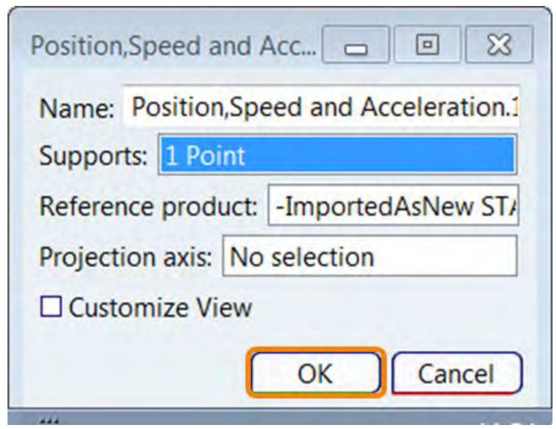
   a. Double click on **Scenario** in the specification tree





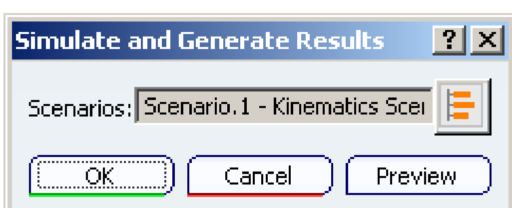
b. Click on the **point, speed and acceleration** icon
c. Select the **point** you have just created
d. Click **OK**



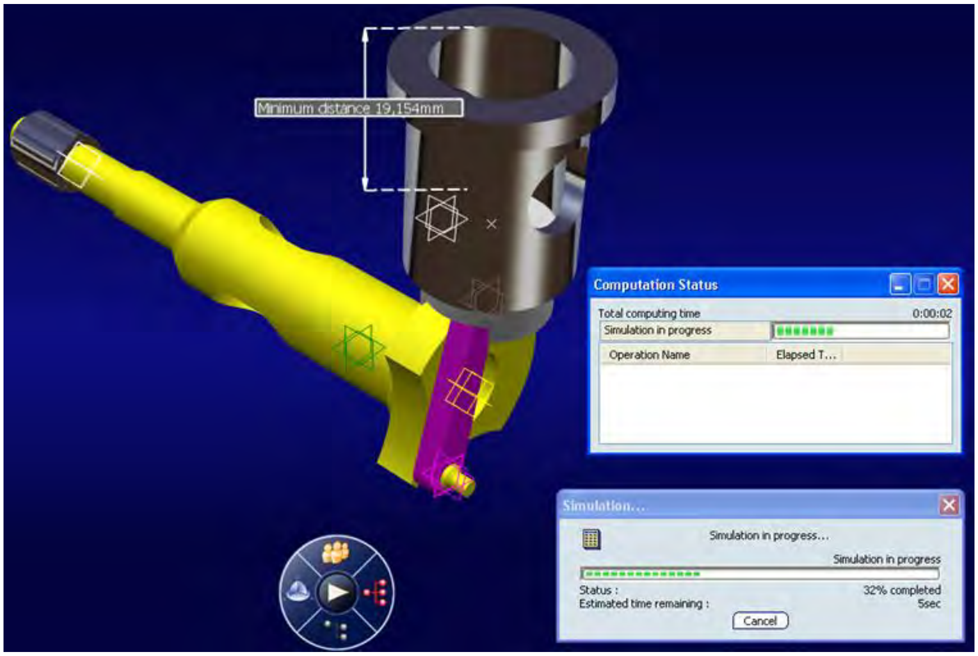


11. Simulate the results

a. Click **simulate and generate results**
b. Click **OK**

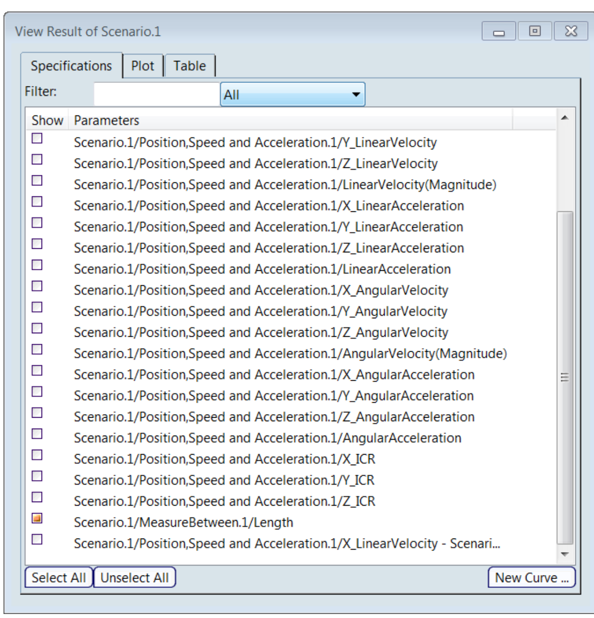


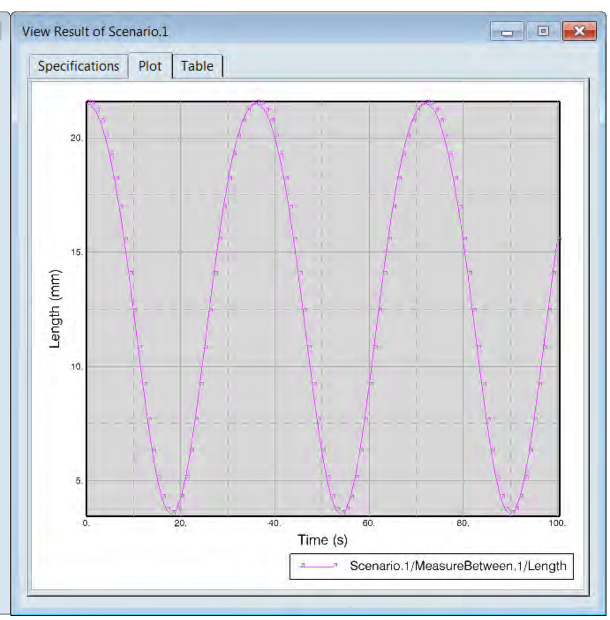
c. The software creates the simulation.



12. View the results:

a. Click on the **export probe results** icon.
b. Click **on a Scenario.1** result e.g. Scenario.1/MeasureBetween.1/Length.
c. Click on the **plot** or **table** tab to view results





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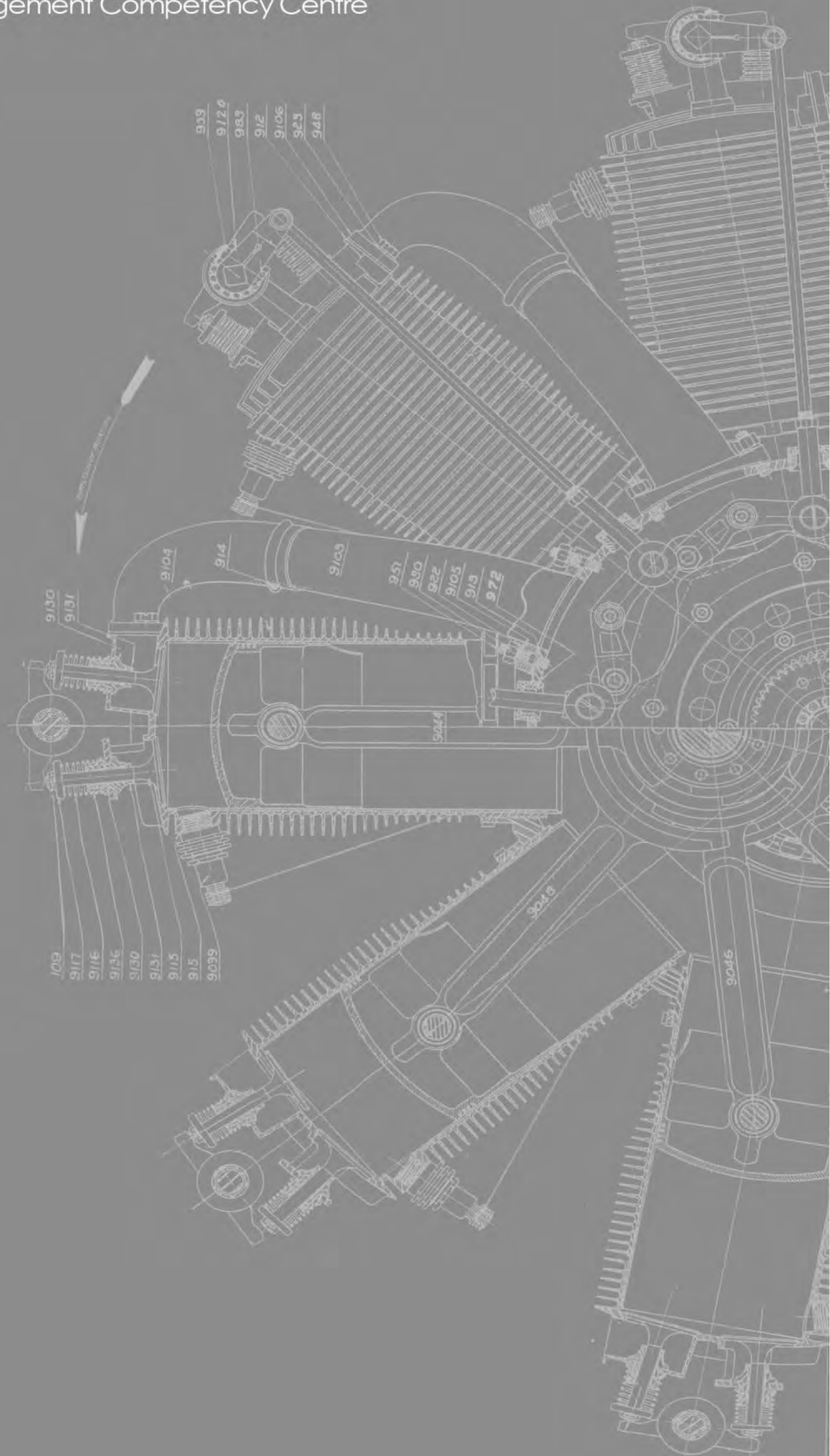

Design Sight

DASSAULT SYSTEMES

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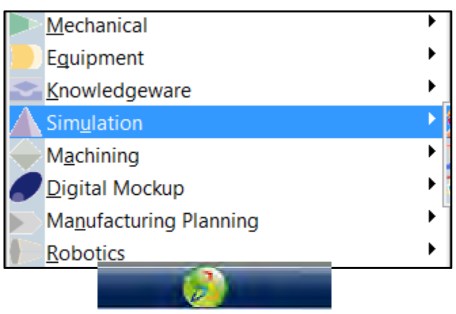
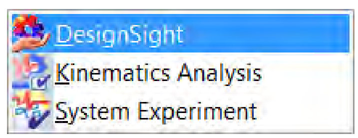
French ministry of education and research

DESIGNSIGHT

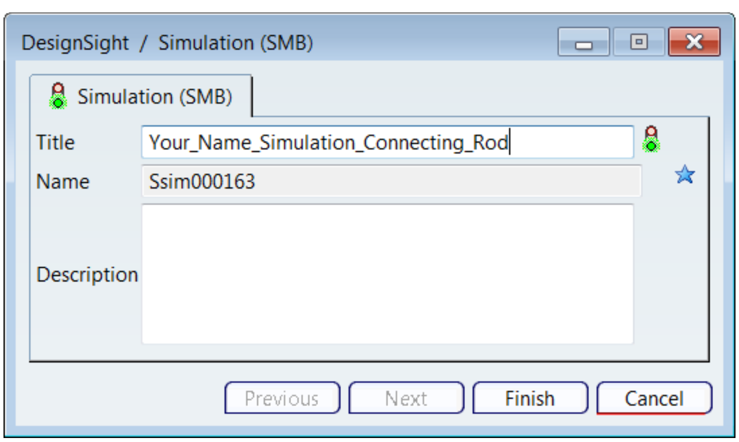
During these steps, we will perform a **stress simulation** and analyse the connecting rod.
You can import the connecting rod provided by the courseware or use your own connecting rod designed previously.
The introduction of SIMULIA products on the V6 platform with **DesignSight** Structure allows designers to run robust **realistic simulations**. The product suite is engineered FOR THE DESIGN COMMUNITY AND DOES NOT REQUIRE EXTENSIVE SIMULATION EXPERTISE.

1. Import the „StressAnalysis_Connecting_Rod" 3DXML file.
2. Import as new and fill „Your_Name_" in the duplicating string.
3. Click Ok
4. Right click on Your_Name_Connecting_Rod on the top of the Specification Tree to open it in an authoring view.
5. Click on Start and find the DesignSight workbench under the Simulation category.



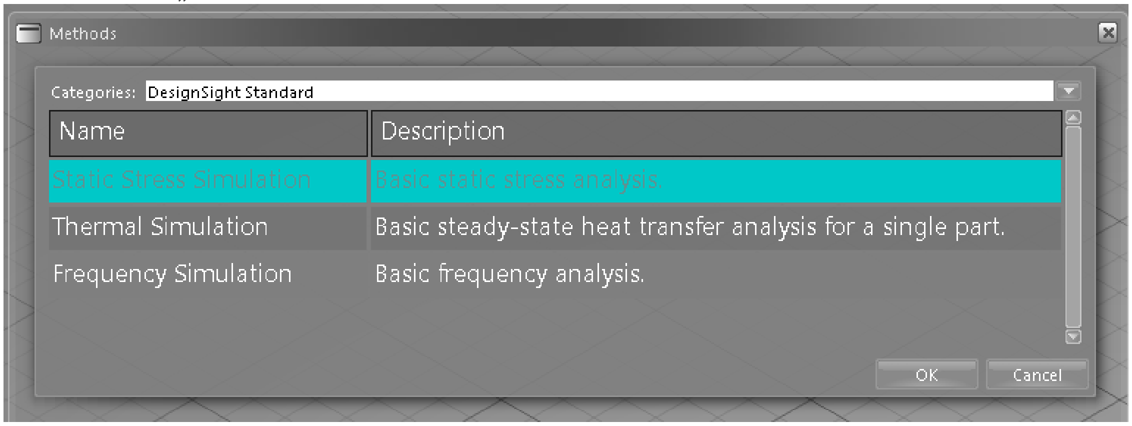


6. Name it as [Your_Name_Simulation_Connecting_Rod]
7. Click „Finish" to validate the name

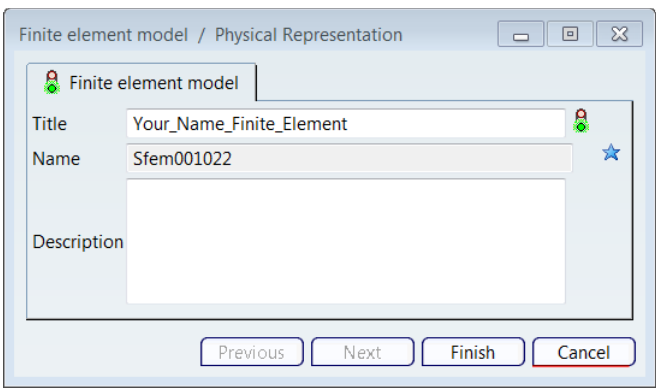


8. Click „Static Stress Simulation"

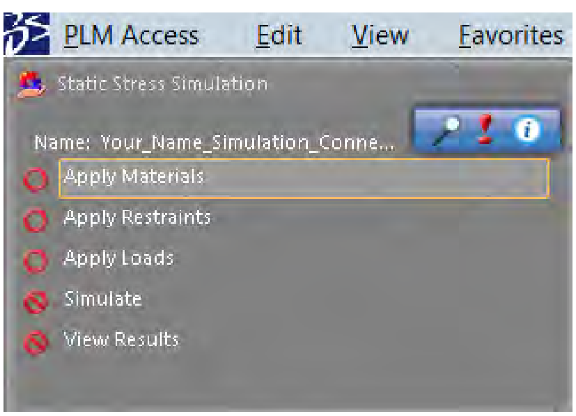
9. Click „OK" to validate



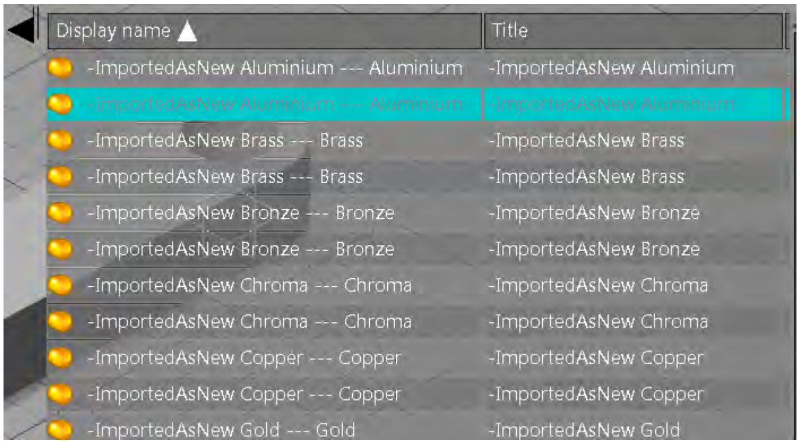
10. Name the Finite Element Model / Physical Representation as „Your_Name_Finite_Element".

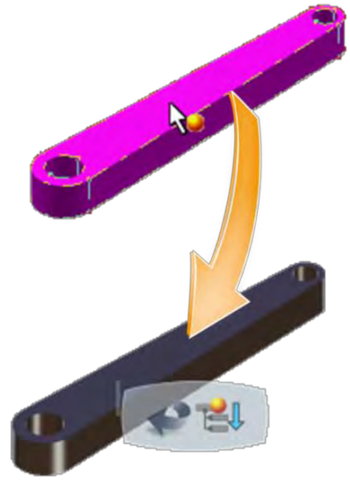


11. Click on „Apply Materials"

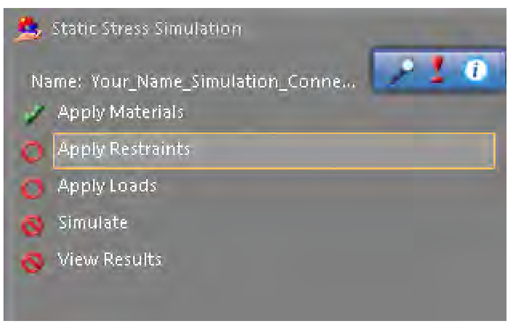


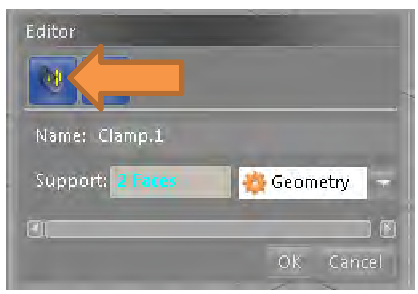
12. A material list pops up. Select Aluminium then click on the Connecting Rod in the graphic area. Close the material list.



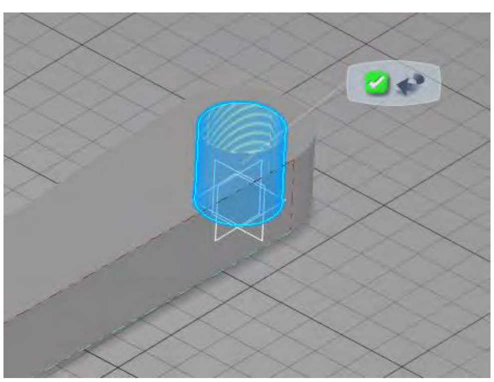


13. Click on „Apply Restraints“
14. Click on the „Clamp“ function in the editor.

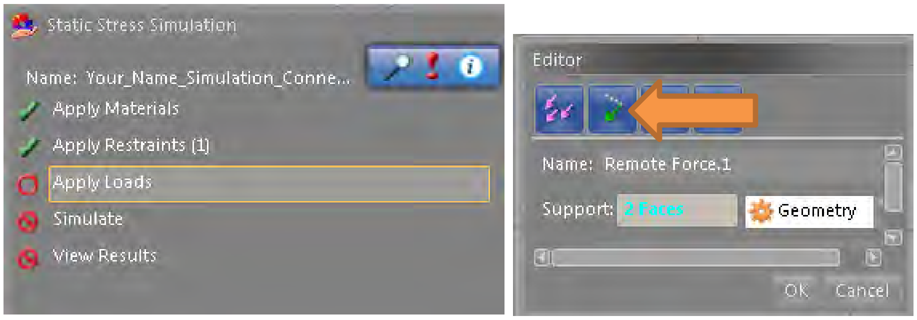




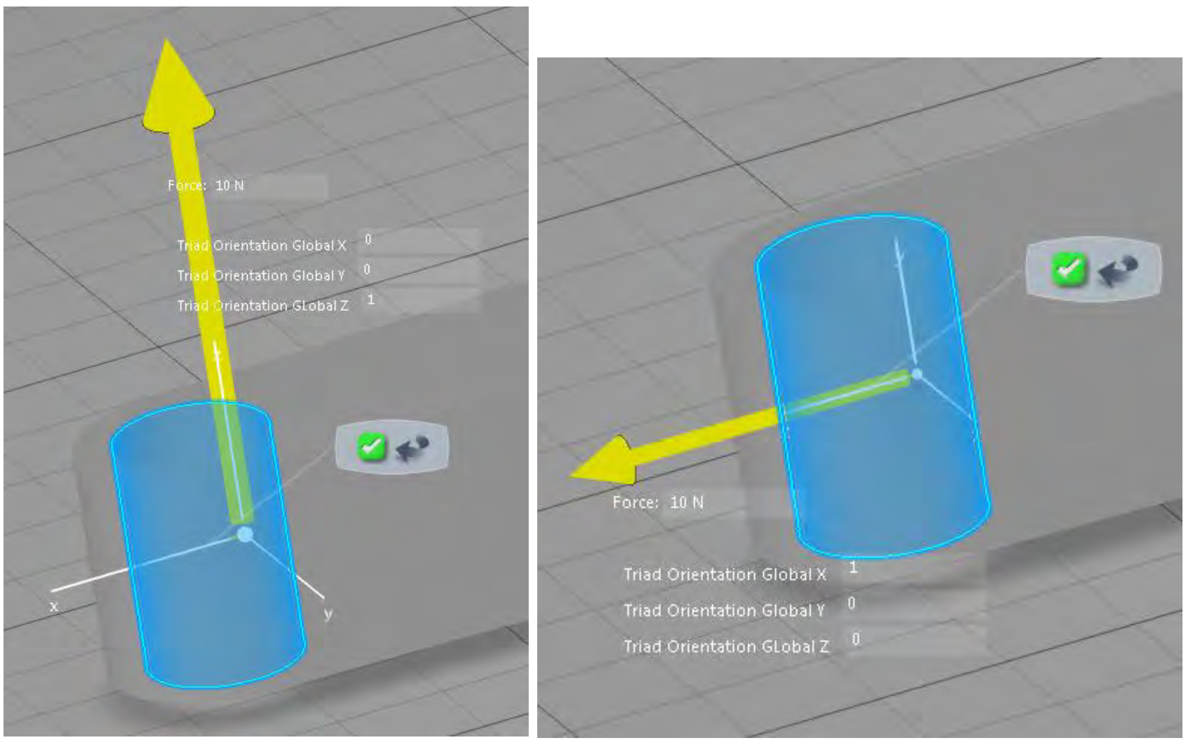
15. Select the inner surface of the bigger hole to be clamped and click on the green tick to confirm.



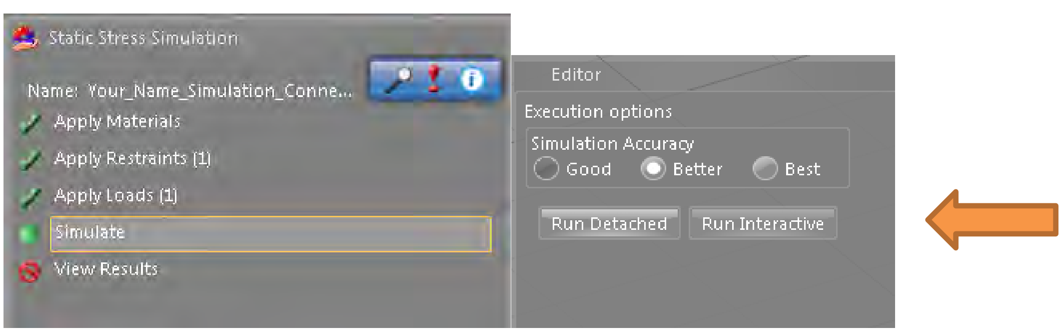
16. Click on „Apply Loads“
17. Apply a remote force on the inner surface of the smaller hole.



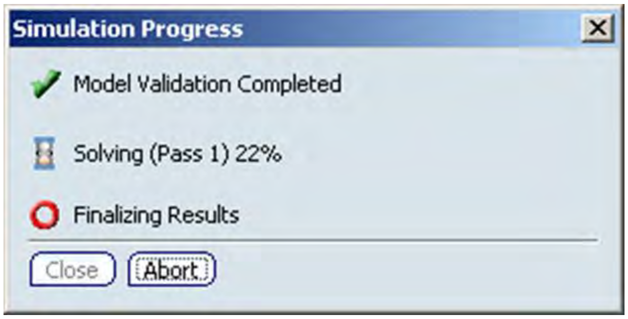
18. Edit the force magnitude to 10N.
19. Change the direction of the force by double clicking on the XYZ axis, and editing the Global orientation as below. Click on the green arrow to confirm.



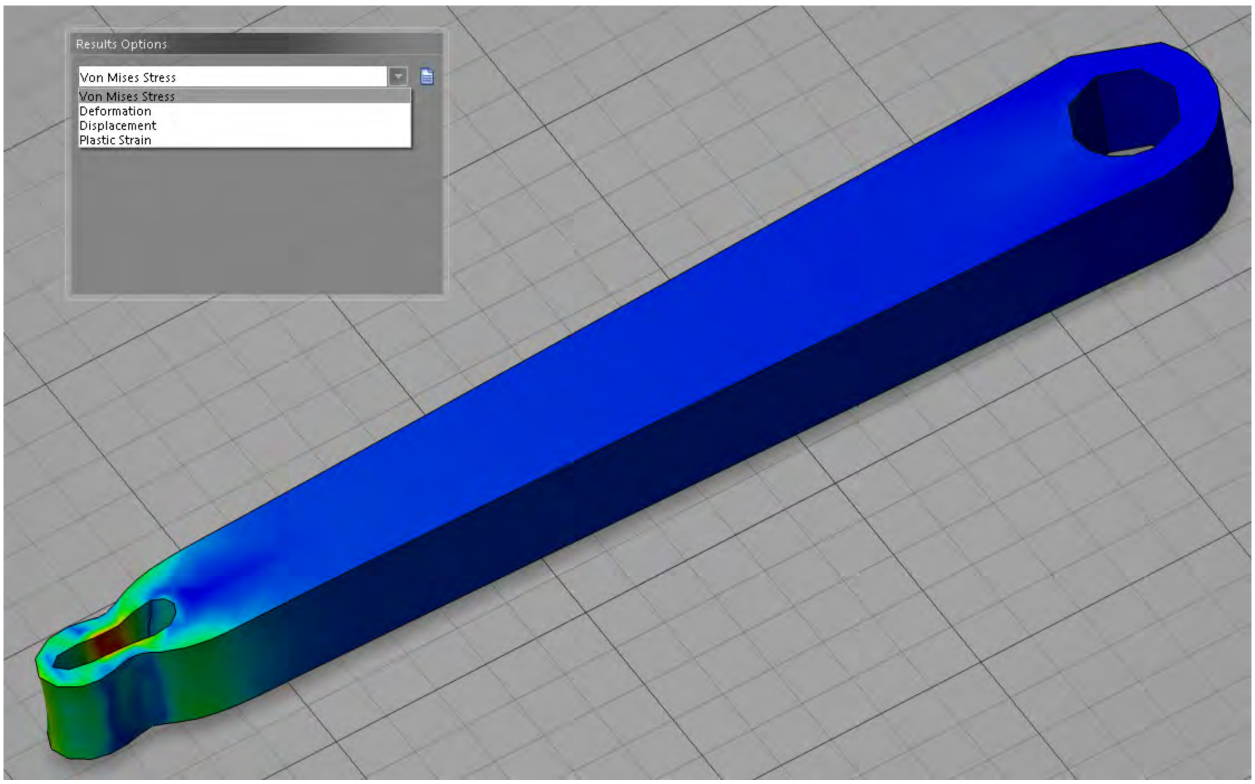
20. Click on „Simulate", choose „Better" simulation accuracy. Select Run Interactive.



21. The window shown below will appear to show you the progress of your simulation



22. You can visualize the result by selecting one of the Results options.



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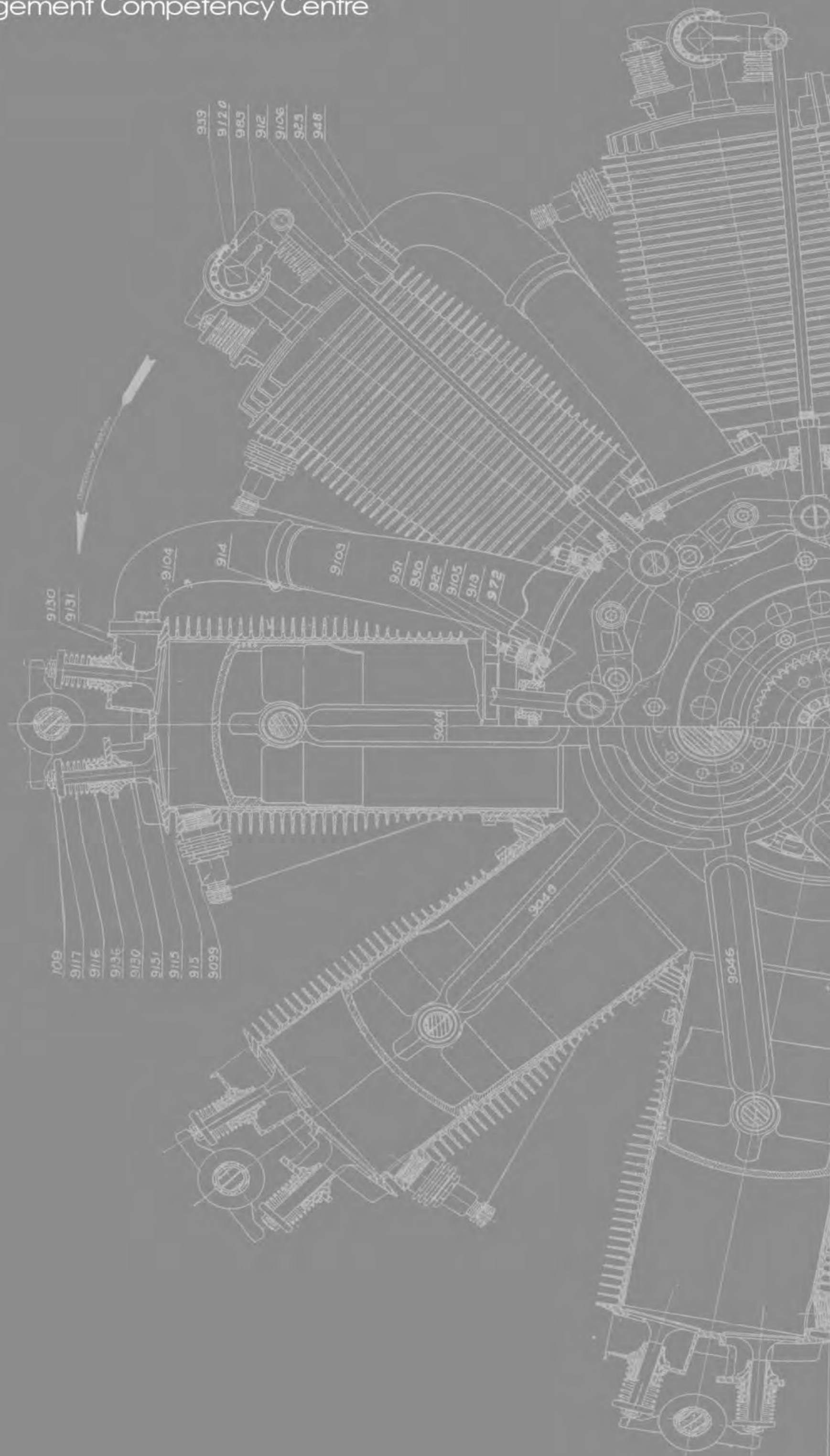

Machine Programming

Dassault Systemes

Cape Peninsula University of Technology

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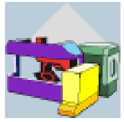
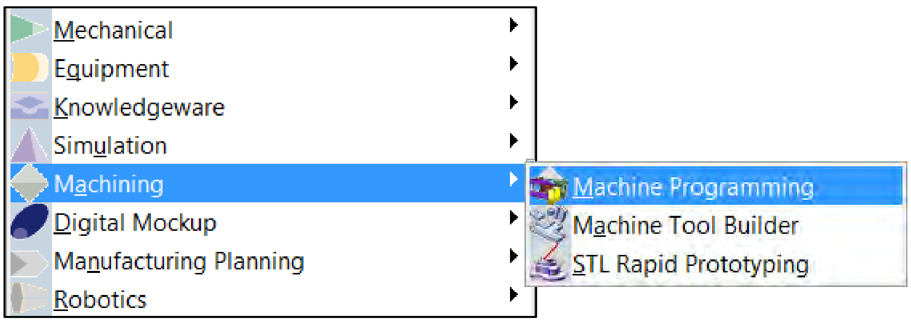
Machine Programming 

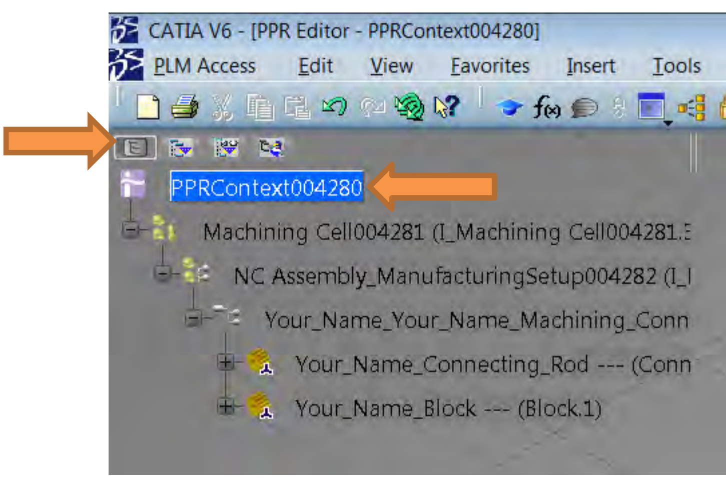
During these steps, we will generate the NC program to machine the connecting rod. **Prismatic Machining** enables you to define and manage NC programs dedicated to machining parts designed in 3D wireframe or solids geometry using 2.5 axis machining techniques.

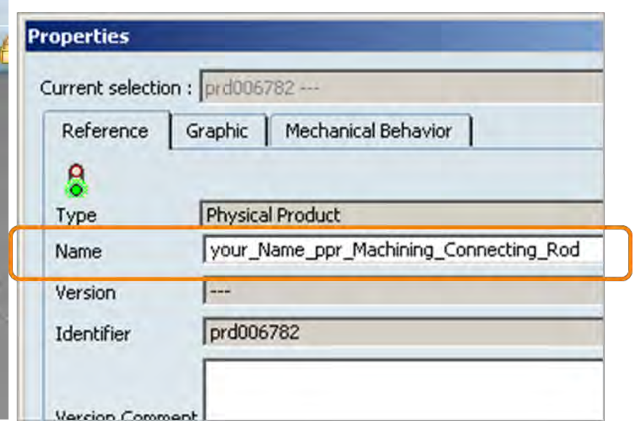
1. **Import the 'Machining_Connecting_Rod.3dxml'**
2. **Open the part in an authoring view.**
3. **Click on 'Start' in the bar > 'Machining' > 'Machine Programming.**



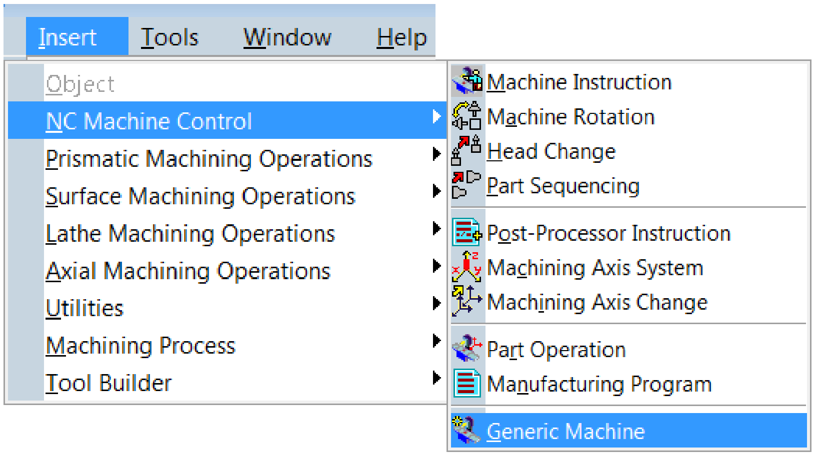
4. **We will rename the current 'ppr file'.**

    a. Select the specification tree tab.
    b. Right click on the name at the top of the „**Specification tree' > „Properties**"
    c. Enter [your_Name_ppr_Machining_Connecting_Rod] as „Name"

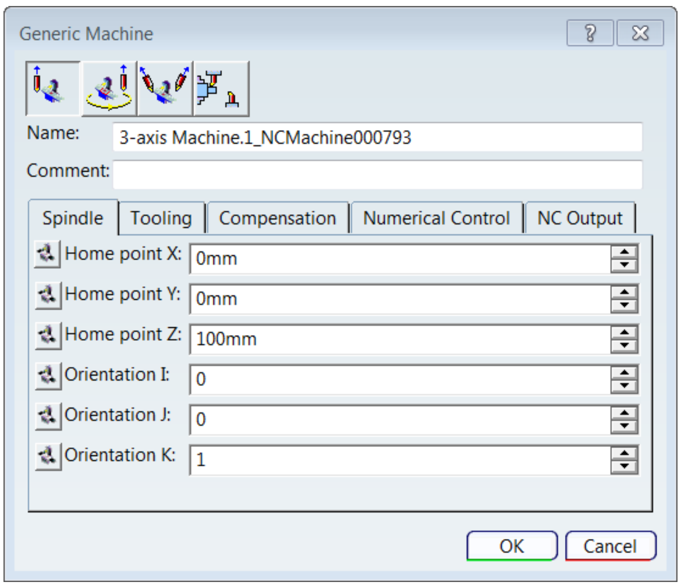




5. **Assign an NC machine to perform the machining.**
6. **Click on Insert: NC Machine Control: Generic Machine.**



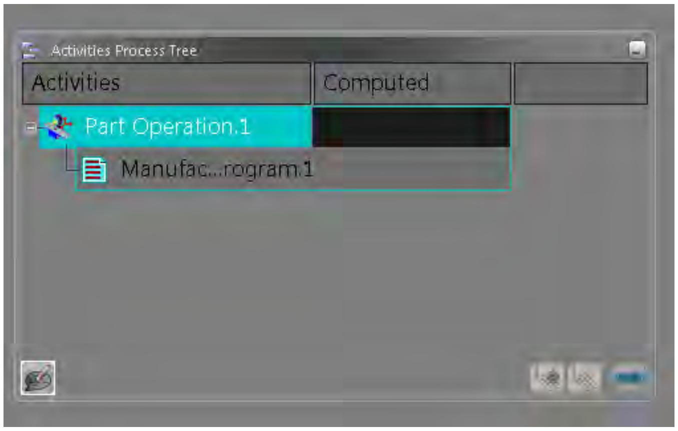
7. **The following dialogue window opens.**



8. **Click ok to accept the Generic Machine.**

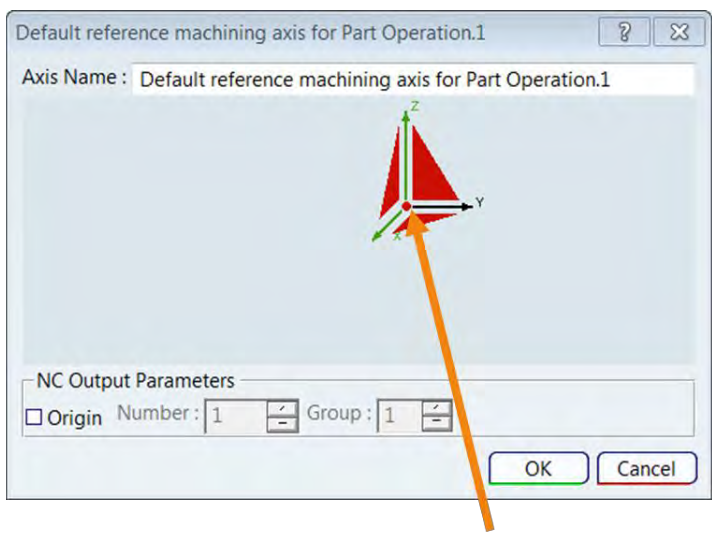
9. **We will now define 'Part Operation.1' in the 'Activities Process Tree' window**

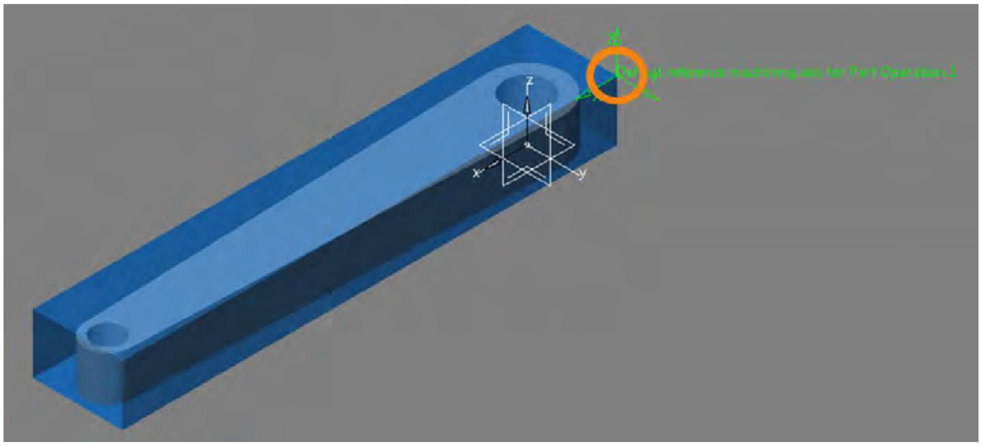
   a. Double click on the „Part Operation.1" activity.
   b. The „Part Operation" dialogue window opens.



10. We will now define the ‘Part Operation.1’

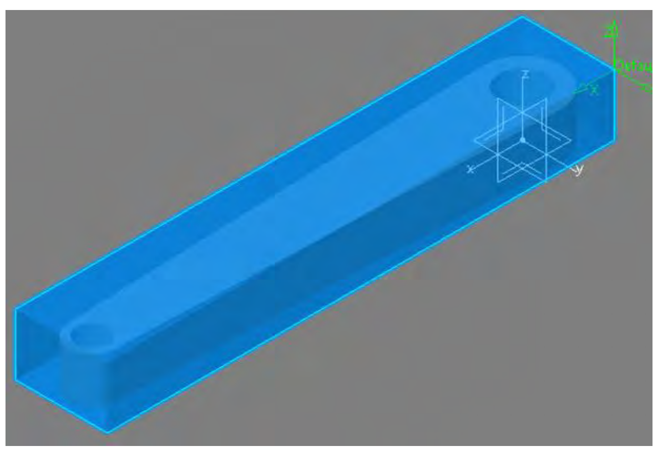
a. Click on the „Default reference machining axis for Part Operation.1“ symbol
b. Click on the center of the axis on the dialogue window, then click on the corner of the part as shown on the figure below.





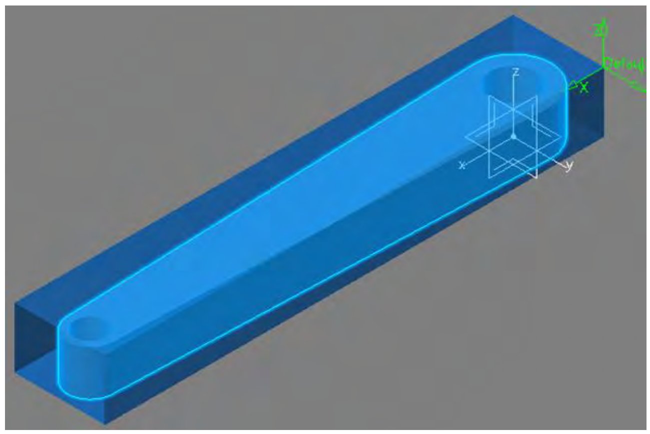
11. We will now define the ‘Rough Stock’

a. Click on the „Rough Stock‟ icon
b. Double click on the „PartBody‟ of the „Block‟ part as shown below.



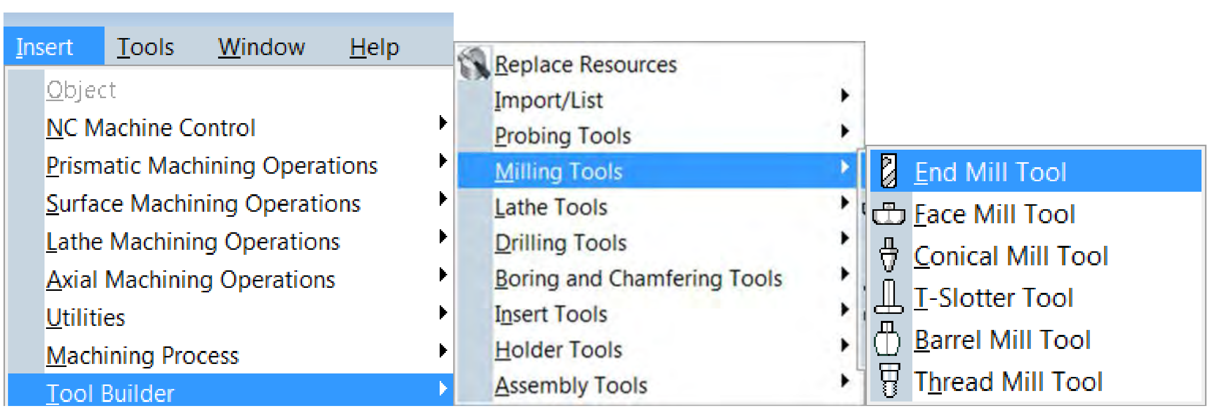
12. We will now define the ‘Design part’

a. Click on the „ Design part“ icon
b. Double click on the „PartBody“ of the „Connecting_Rod“ part as shown below.

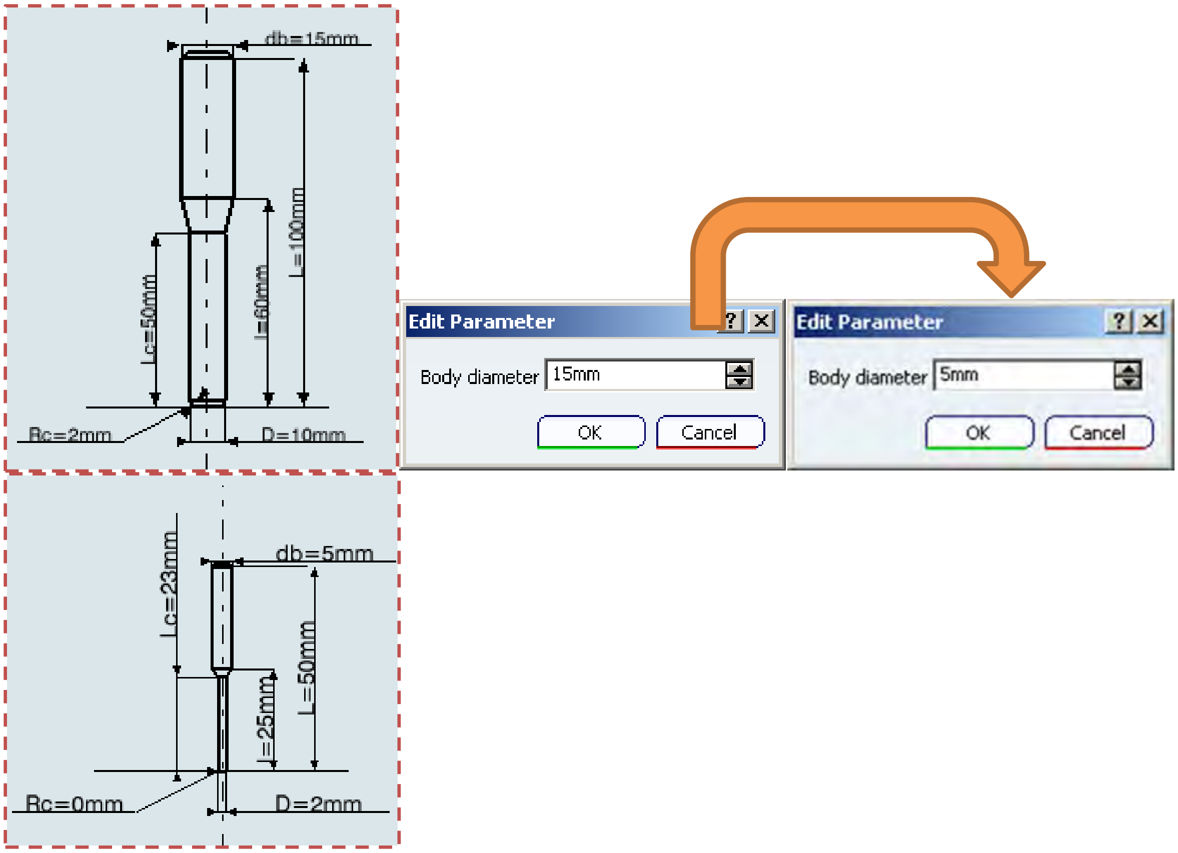


13. We will now face the top surface of the stock. We will start by the tool definition then we’ll define the facing operation

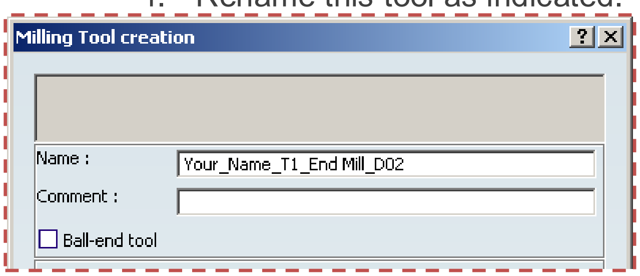
a. Select „**Insert**“ Menu > „**Tool Builder**“ >
b. Then select „**Mill Tool**“ > „**End Mill Tool**“



c. Double click on the dimension
d. Edit the dimension to the desired amount and click „OK“
e. Repeat the previous steps untill you get the correct dimensions

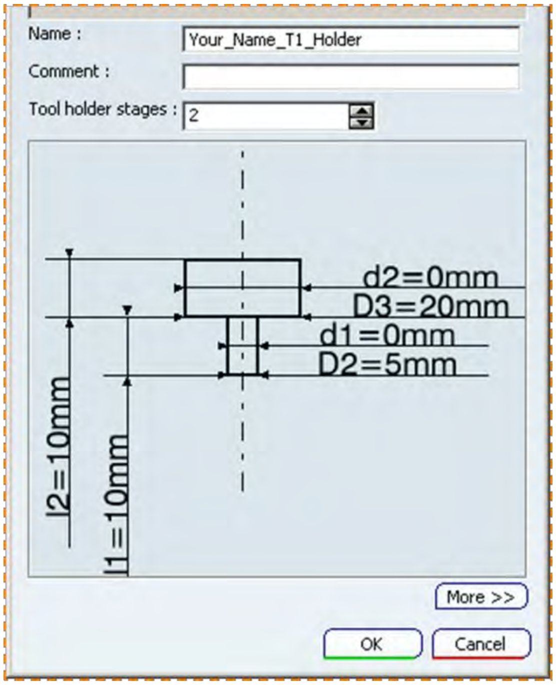


f. Rename this tool as indicated.



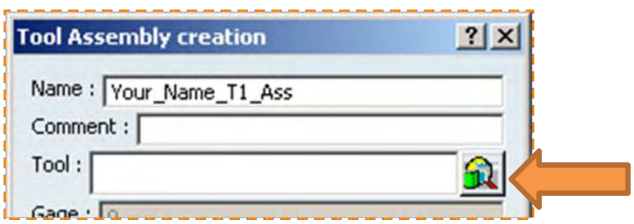
14. We will now define the tool holder for the previous tool.

a. Select „**Insert**" Menu > „**Tool Builder**" > „**Holder Tool**" > „**ConicalTool Holder**"
b. Modify the values as indicated opposite
c. Rename it „Your_Name_T1_Holder"
d. Click „OK" to validate it.

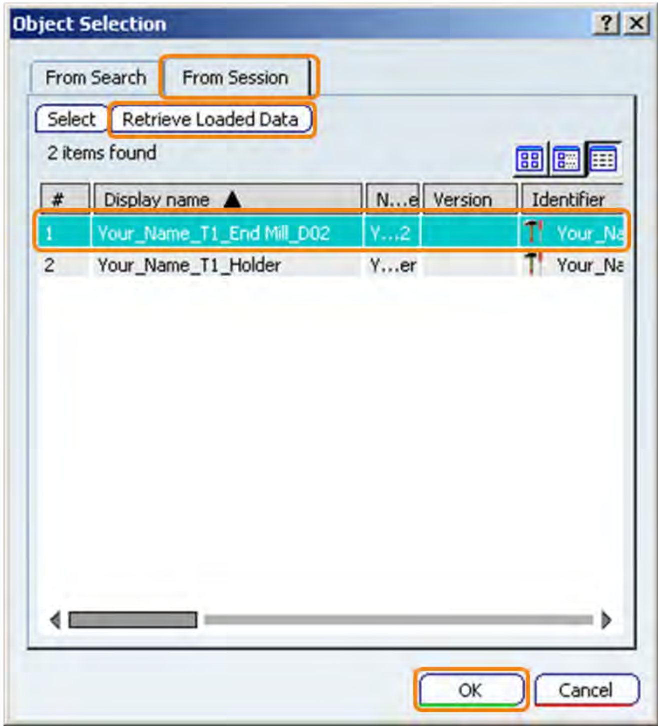


15. We will now assemble the tool holder with the tool.

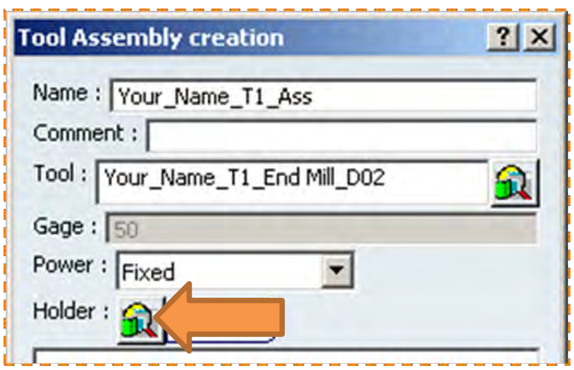
a. Select „**Insert**" Menu > „**Tool Builder**" >"Assembly Tools" > „**Tool Assembly**"
b. Name it [Your_Name_T1_Ass]
c. Click on the „Object selection" icon



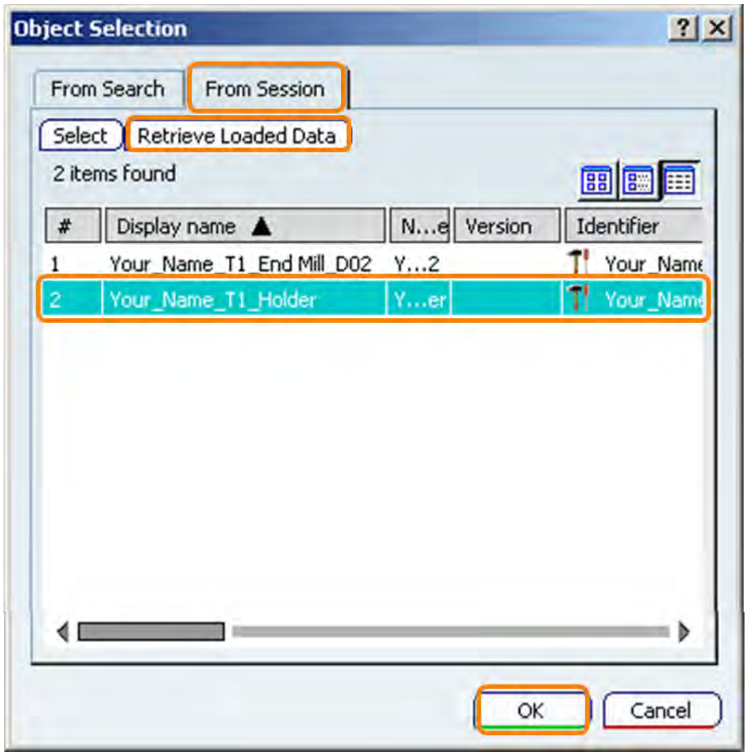
d. Select the „From Session" tab
e. Select then „Retrieve Loaded Data"
f. Select the tool „Your_Name_T1_End Mill_D02"
g. Click „OK" to validate



h. Click on the „Object selection" icon to select the tool holder



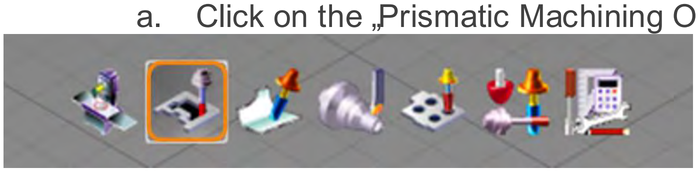
i. Select „From Session" tab
j. Select then „Retrieve Loaded Data"
k. Select the tool „Your_Name_T1_Holder"
l. Click „OK" to validate the tool creation and assembly.



16. Hide the ‘Block’

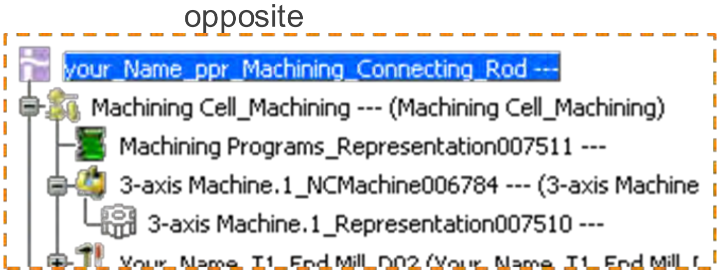
17. We will now define the first facing operation

a. Click on the „Prismatic Machining Operations" icon



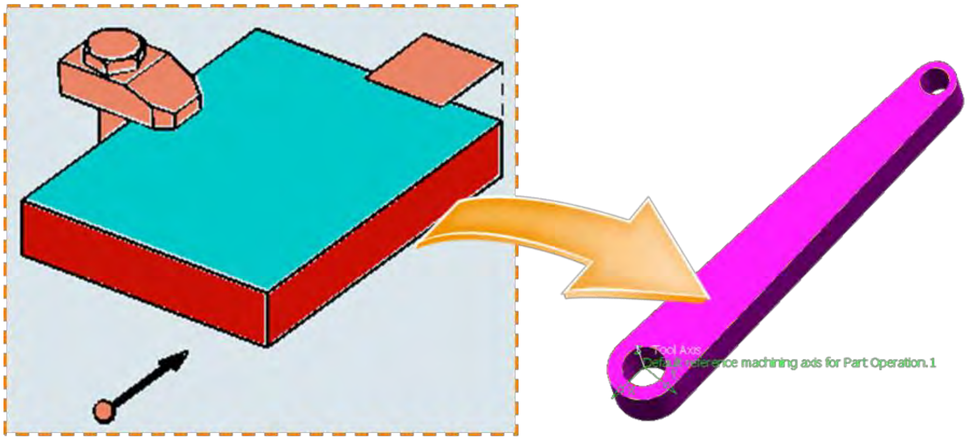
b. Then click on the „facing" icon

c. Select „Your_Name_ppr_Machining_Connecting_Rod" as indicated opposite

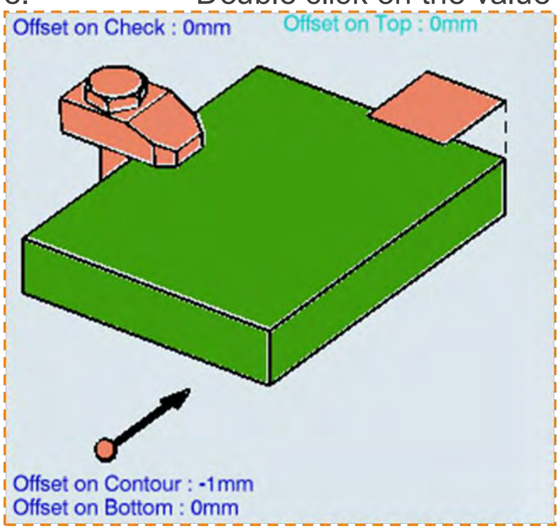


PLMCC

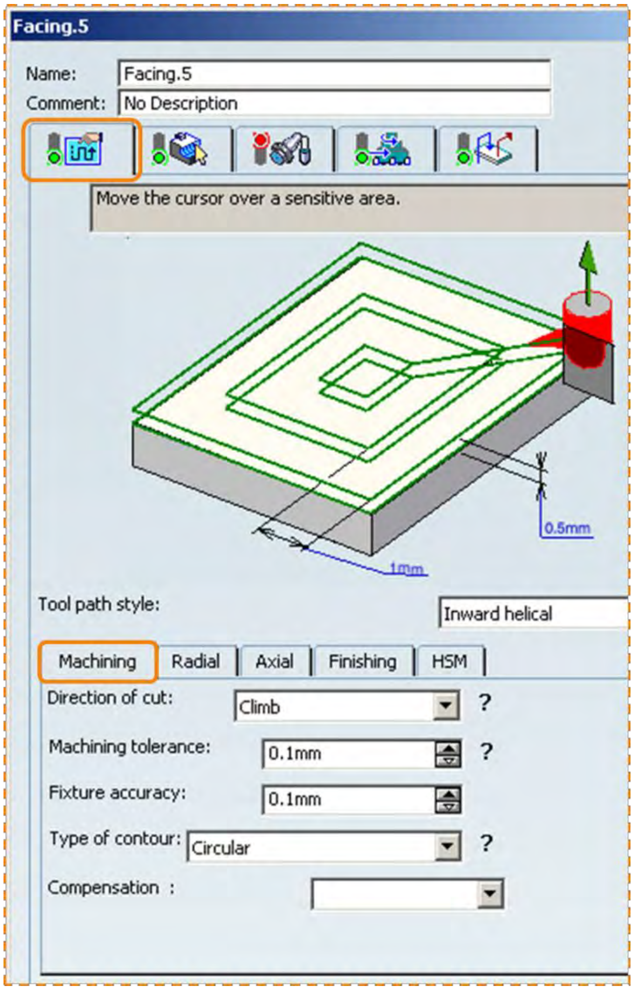
d. Select the red top surface and then select the top surface of the connecting rod as indicated below.

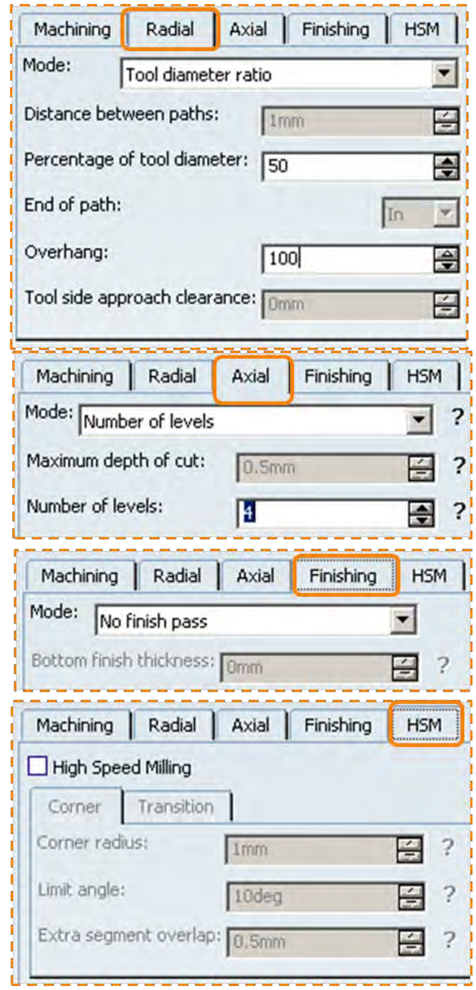


e. Double click on the value of the „Offset on the contour" and enter [-1mm]



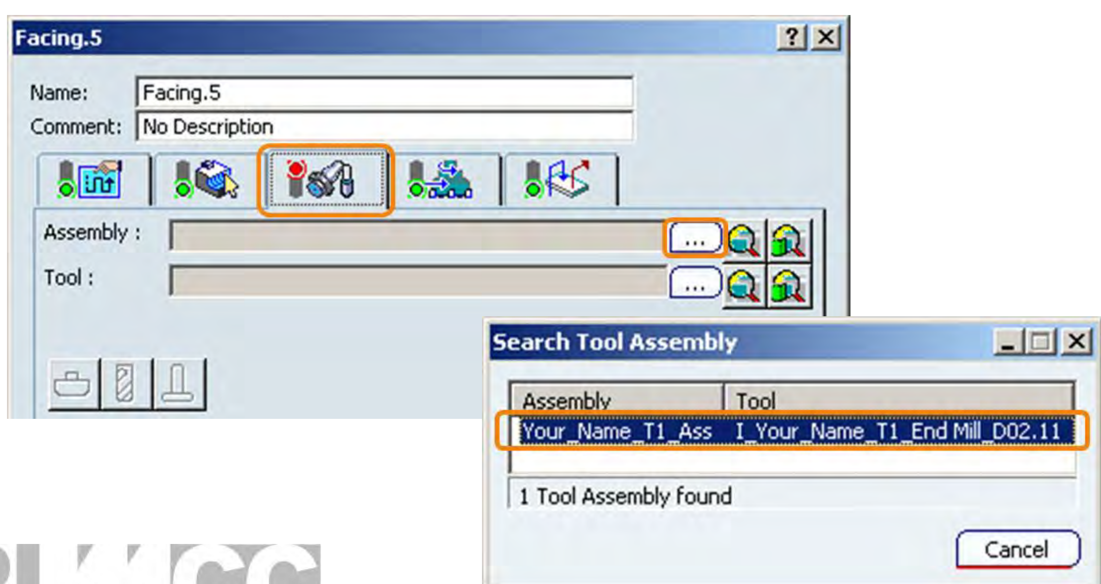
18. We will now define the 'StrategyShortHelp' tab

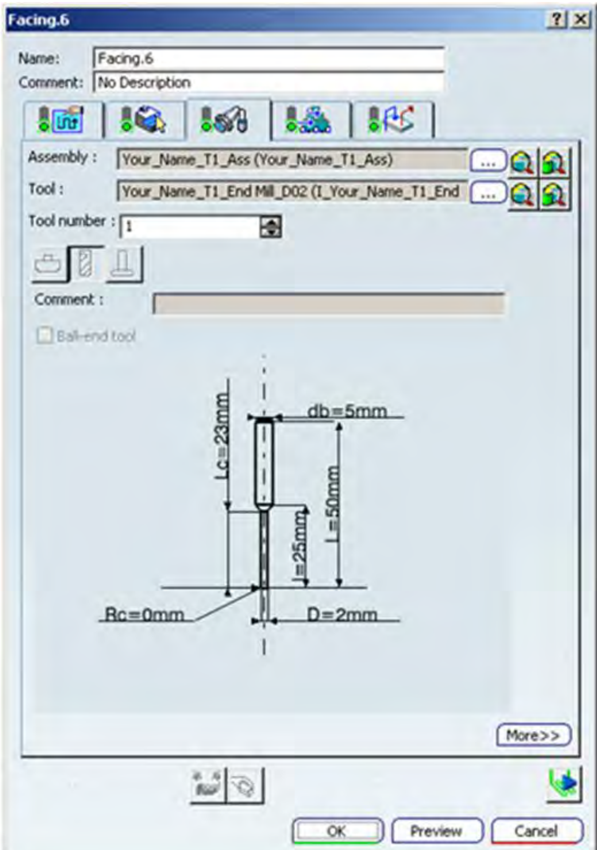




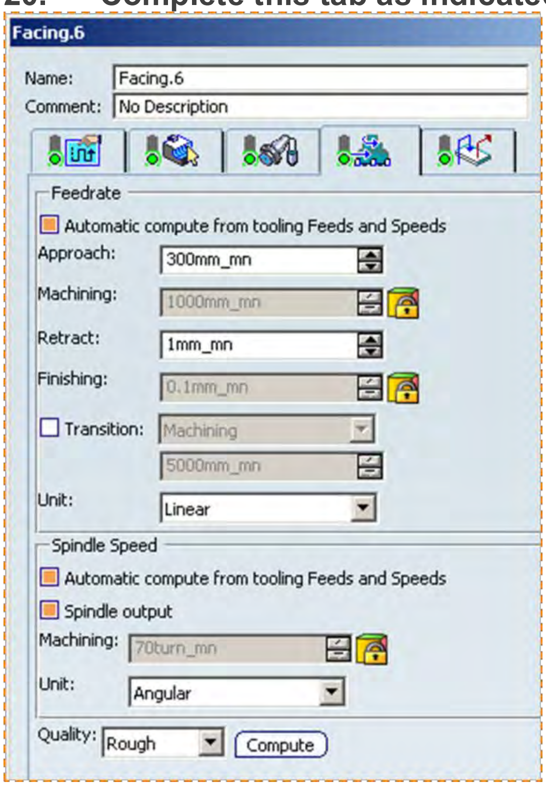
19. We will now define the 'Tool' tab.

a. Click on the search button for
b. Select „Your_Name_T1_Ass"

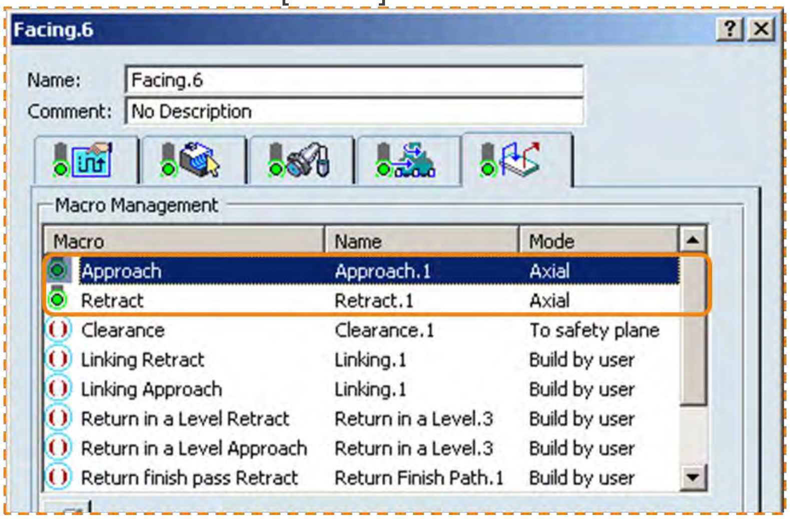


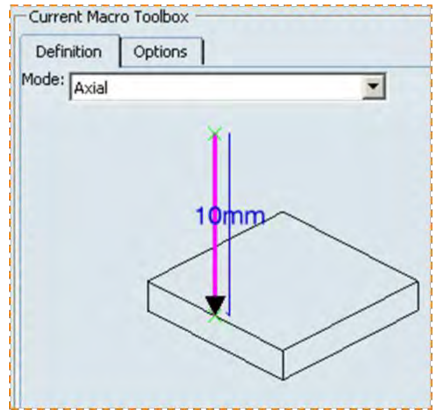


20. Complete this tab as indicated opposite

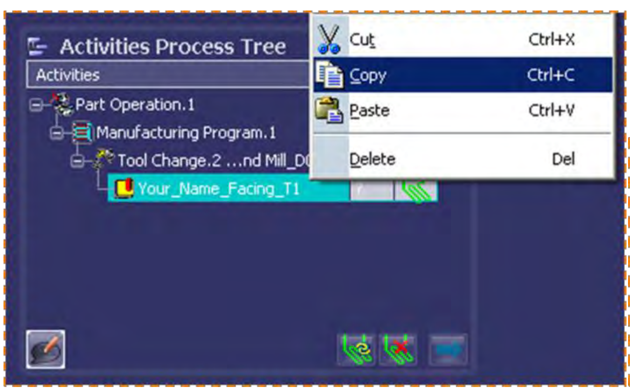


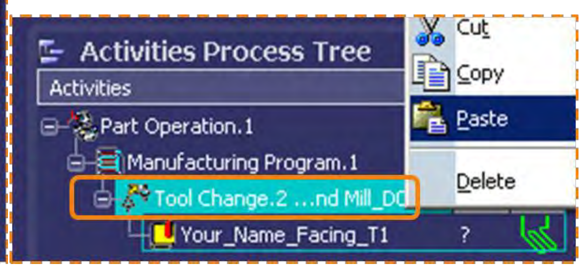
21. **We will now define the 'Macro' tab**
   a. Check that all macro are deactivated except „Approach" and „Retract" Macro
   b. In „Definition" tab select „Axial" as „Mode"
   c. Enter [10mm] as distance for both



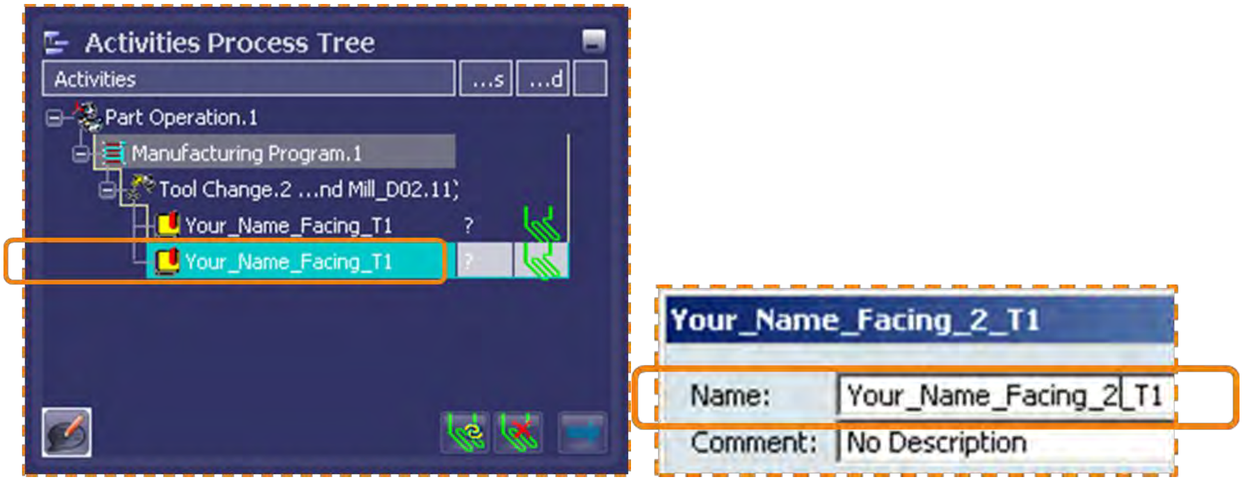


22. **Rename the facing as [Your_Name_Facing_T1]**
23. **Now we will repeat the same operation for the bottom face of the connecting rod.**
   a. Right click on „Your_Name_Facing_T1" > „**Copy**".
   b. Then Right click on „Tool Change.2" > „**Paste**"

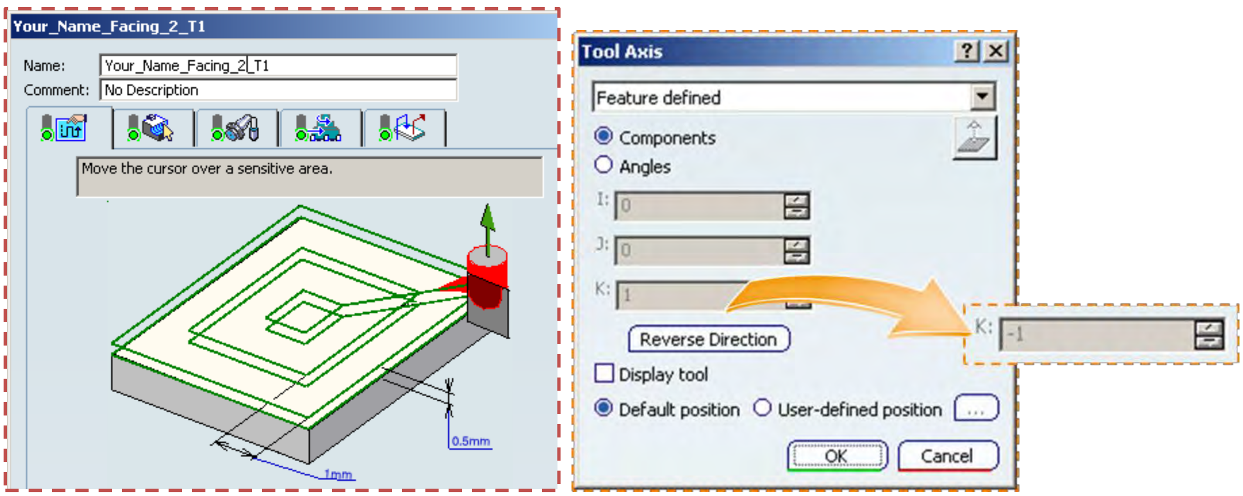




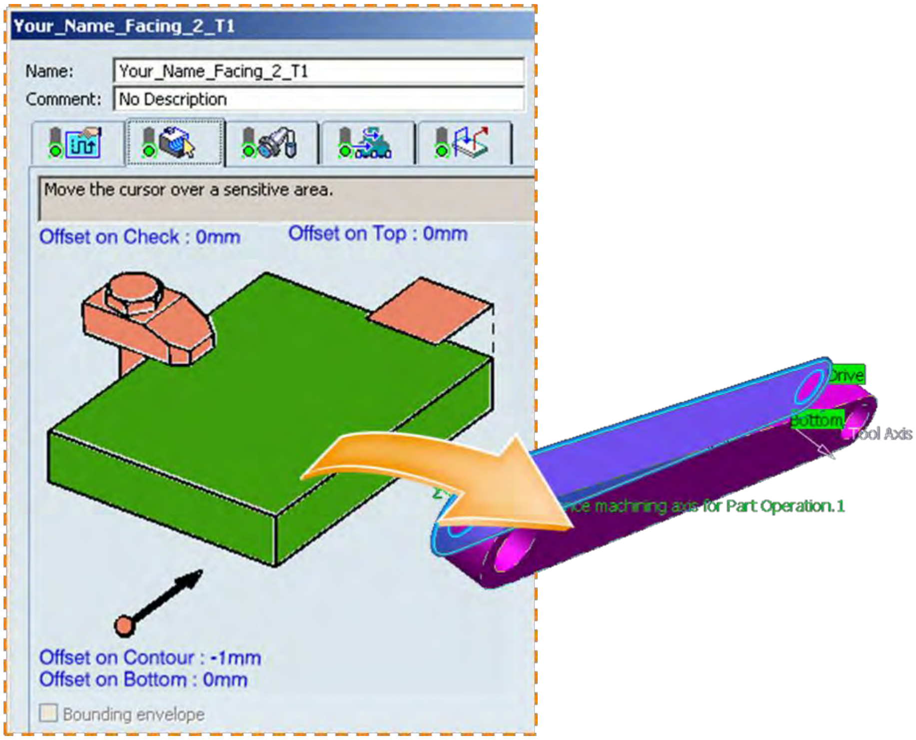
24. **We will now edit and modify the facing operation**
   c. Double click on the previously pasted facing operation
   d. Rename it „Your_Name_Facing_2_T1



c. To change the tool direction, click on the arrow as show opposite
d. Click on the „Reverse Direction" button

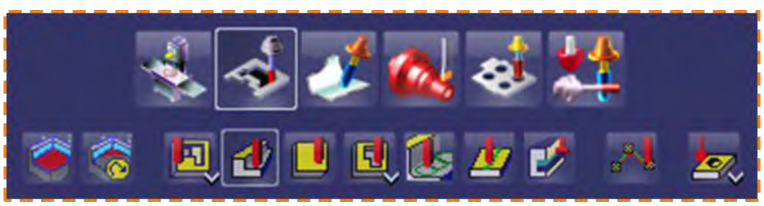


e. Under „Geometry parmeters" tab, click on the top green face
f. Then select the bottom face of the connecting rod
g. Then click „OK" to validate the facing operation

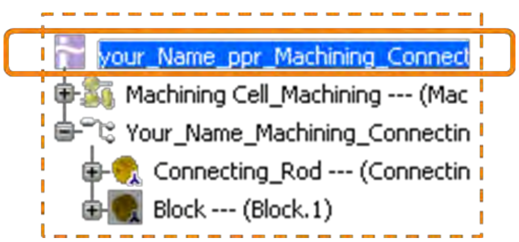


25. Now, we will define the profile contouring operation

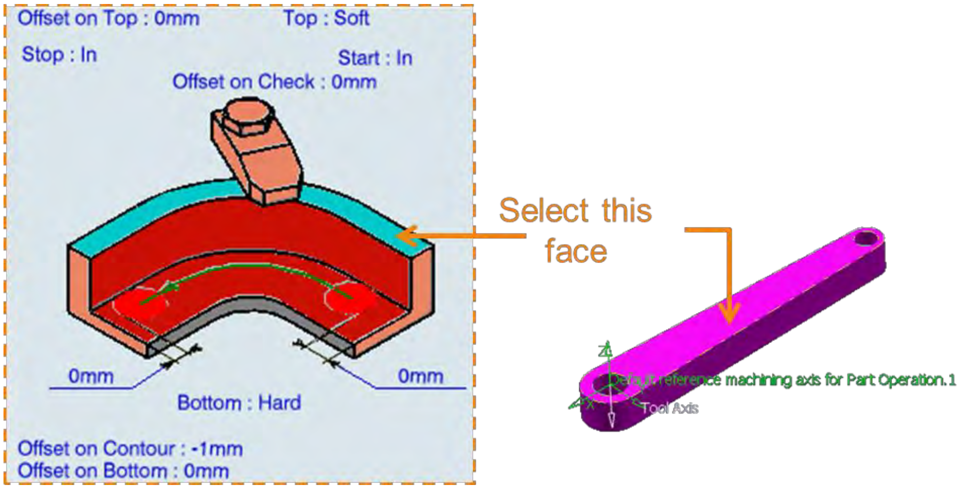
a. Click on the „Profile Contouring" icon



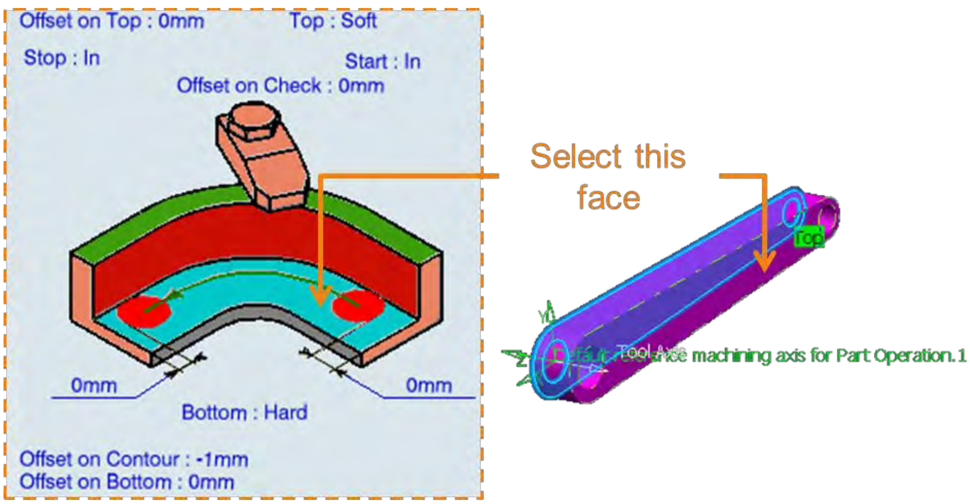
b. Then click on „Your_Name_ppr_Machining_Connecting_Rod"



c. Select the face as shown below
d. Then select the top face of the connecting Rod

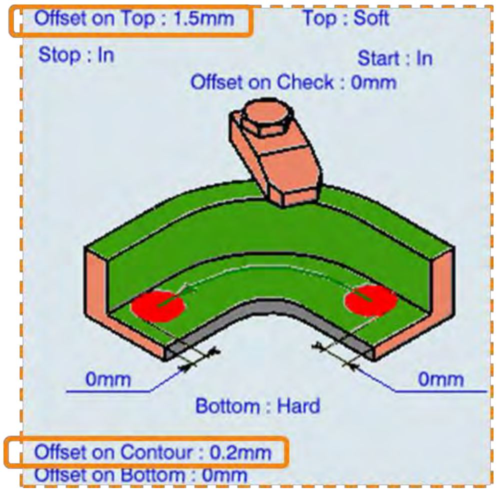


e. Select the bottom face as shown
f. Then select the bottom face of the connecting Rod

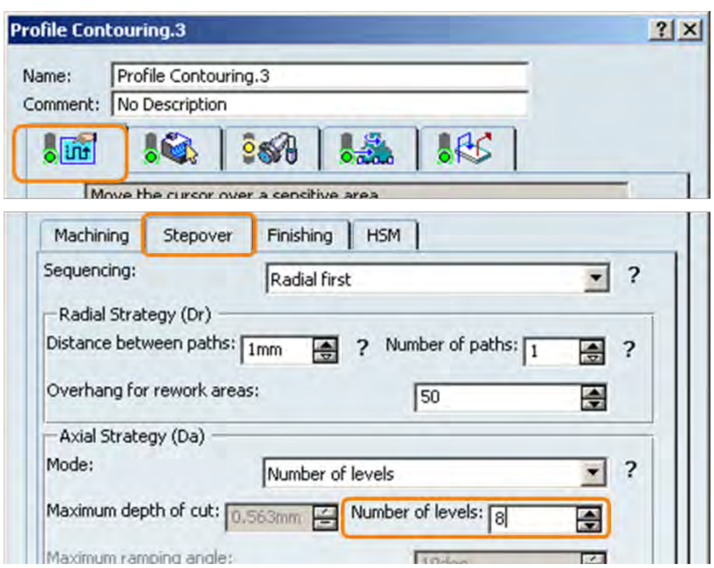


26. We will now modify some offset values

a. Double click on the value of the „Offset on Top" and enter [1.5mm]
b. Double click on the value of the „Offset on Contour" and enter [0.2mm]

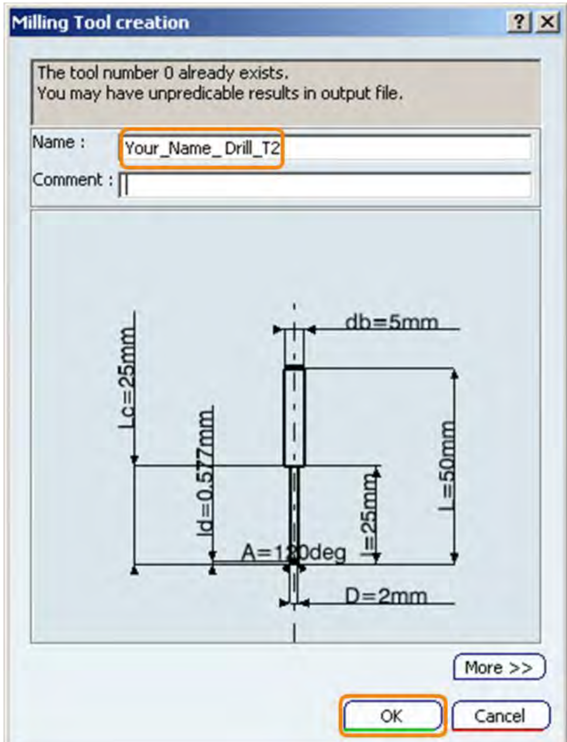


c. Under „StrategyShortHelp" tab select „Stepover" tab then enter [8] as „Number of levels"
d. Rename it „Your_Name_Profile_Contouring_T1"
e. Click „OK" to validate the „Profile Contouring" operation

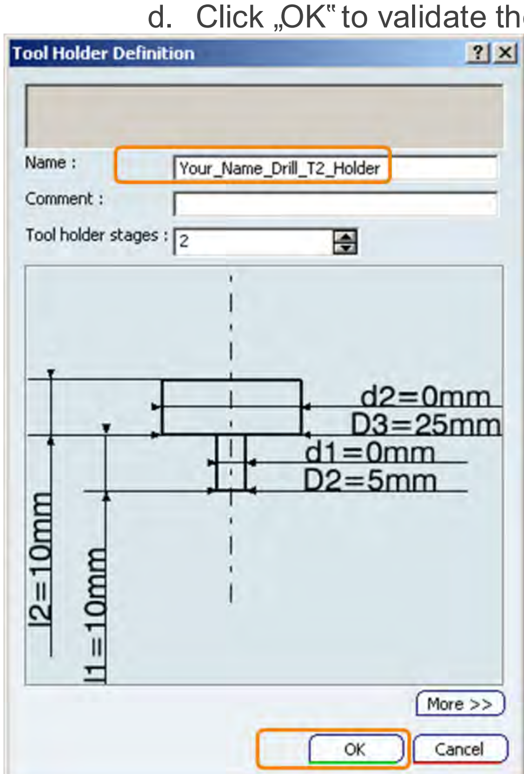


27. Now we will create tool for the first drilling operation

a. Menu „**Insert**" > „**Tool Builder**" > „**Drilling Tools**" > „**Drill Tool**"
b. Rename it as [Your_Name_Drill_T2]
c. Modify values as indicated opposite
d. Click „OK" to validate the drill tool

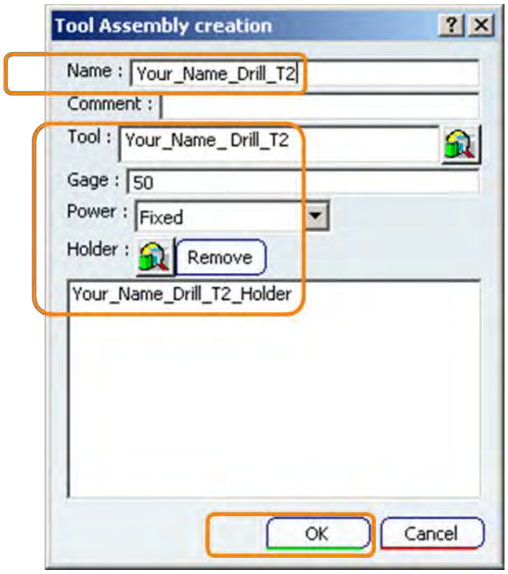


28. **Now we will create tool holder for the first drilling operation**
   a. Menu „**Insert**" > „**Tool Builder**" > „**Holder Tools**" > „**Conical Tool Holder**"
   b. Rename it as [Your_Name_Drill_T2_Holder]
   c. Define the „Tool Holder" as indicated opposite
   d. Click „OK" to validate the drill tool



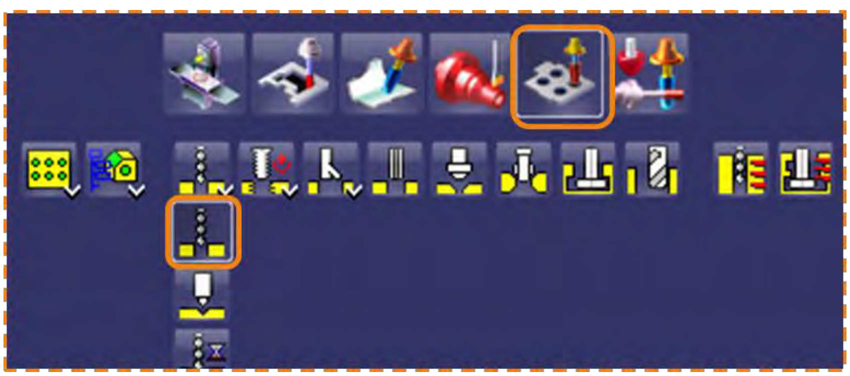
29. **We will now assemble the 'T2'**

a. Menu „**Insert**" > „**Tool Builder**" > „**Assembly Tools**" > „**Tool Assembly**"
b. Name it as [Your_Name_Drill_T2]
c. Select „Tool" and „Holder" as indicated opposite
d. Click „OK" to validate



30. **Now we will drill the smaller Hole.**

e. Click on the „Axial Machining Operations" icon then click on the „Drilling" icon
f. Click on „Your_Name_ppr_Machining_Connecting_Rod on the Specification Tree.



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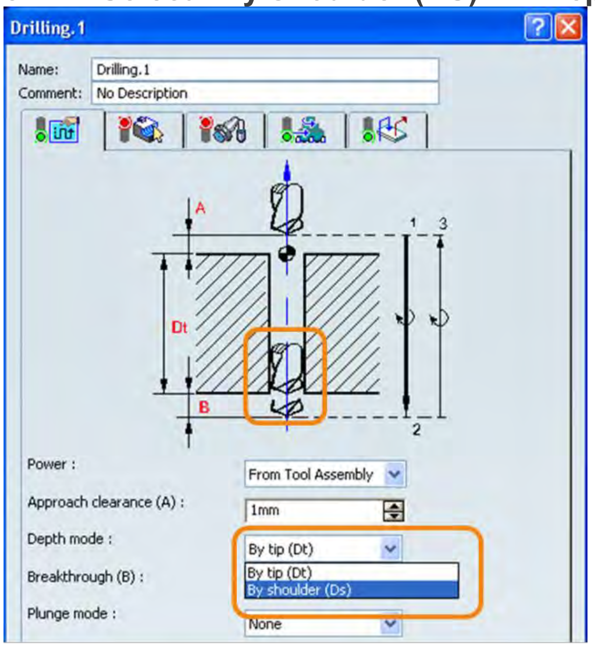
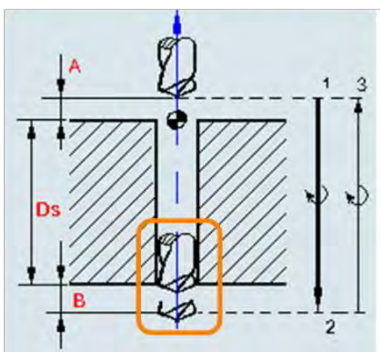
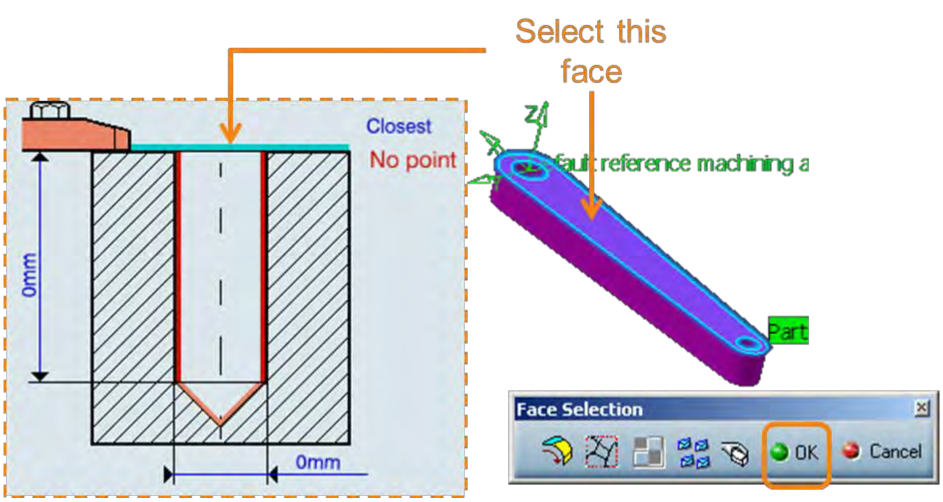
31. Select ‘By shoulder (DS)’ in ‘Depth mode’.



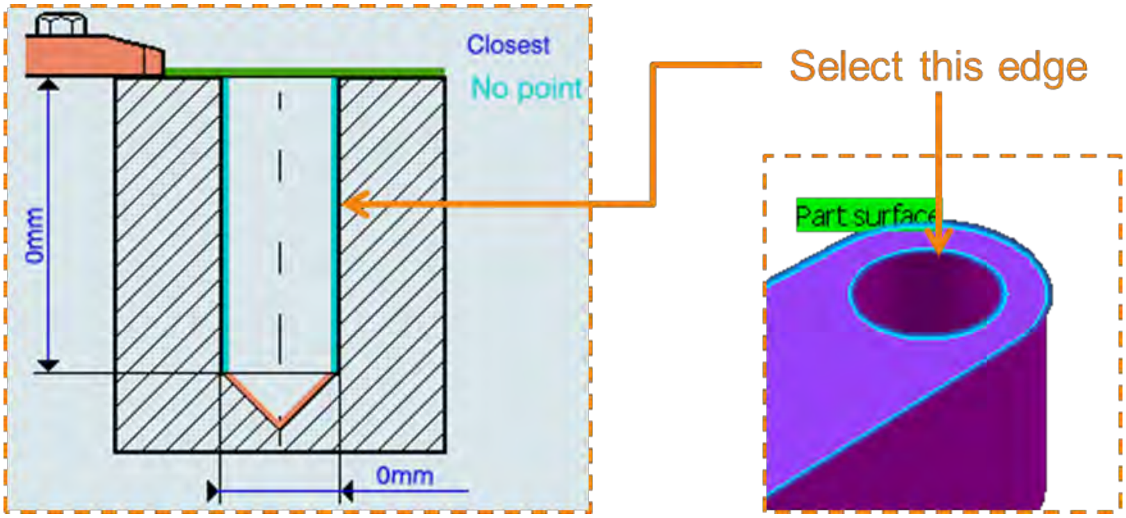


32. We will now define the drilling operation

a. Click on the top face as shown opposite
b. Click on the top face of the connection rod
c. Click „OK“ to validate



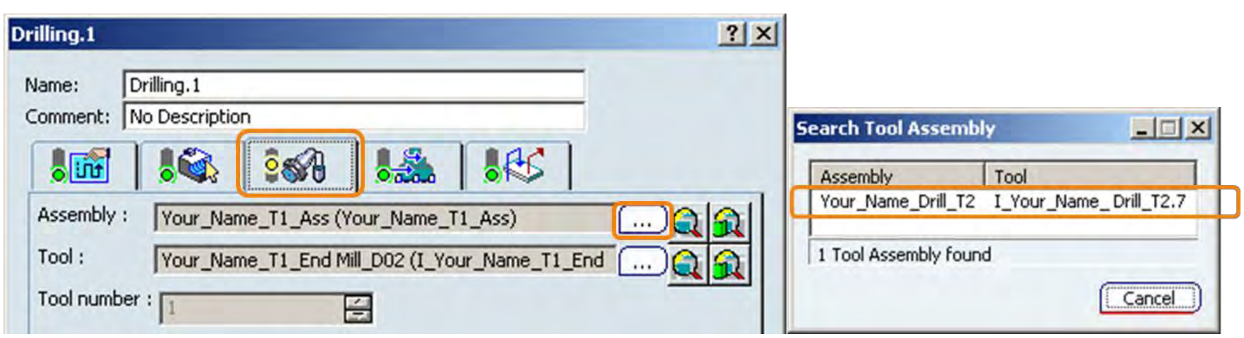
d. Click on the cylindrical face as show opposite
e. Double click on the edge of smaller hole of the connecting rod



f. Click on „Extension : Blind" to replace it with „Extension : Through"

33. We will now define the 'Tool' tab.

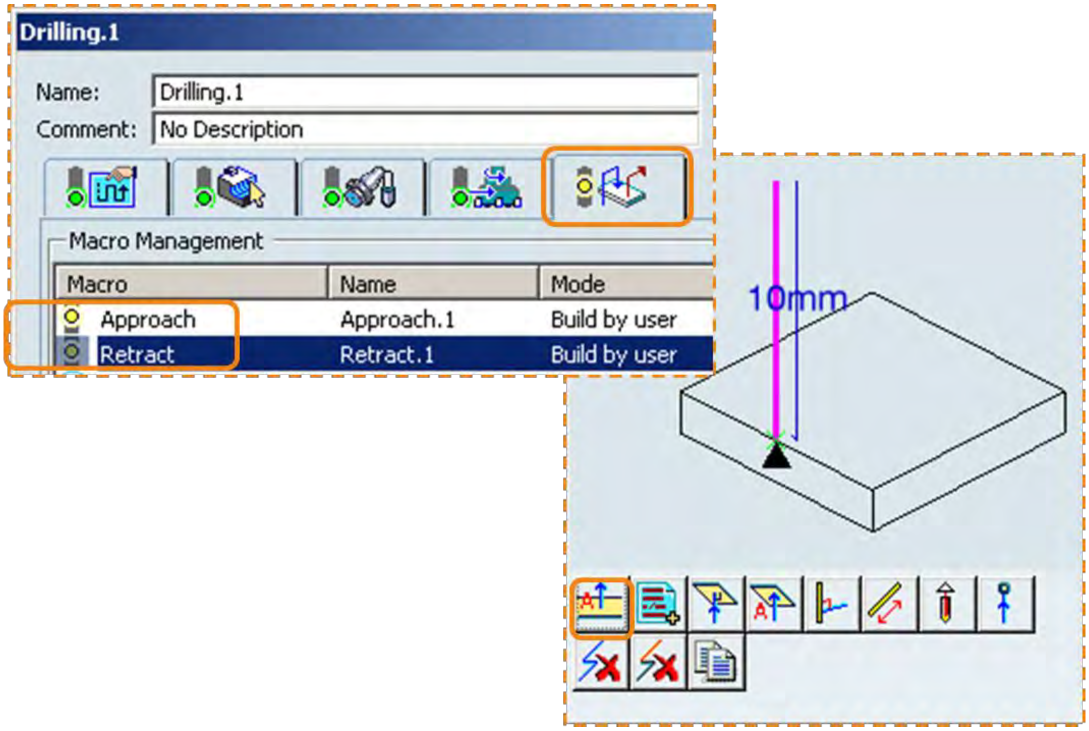
a. Click on the search button to insert the tool assembly.
b. Select „Your_Name_Drill_T2"



34. We will now define the 'Macro' tab

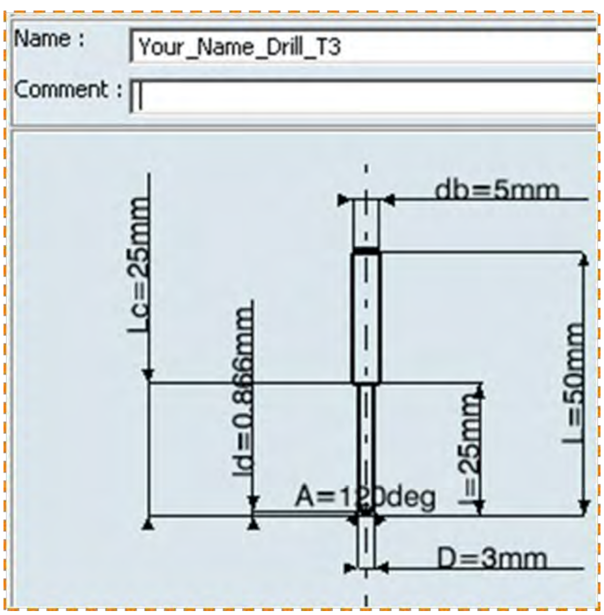
a. Activate „Approach" and „Retract" macro
b. Click on the „Add Axial motion" icon,
c. Modify the value to [10mm]

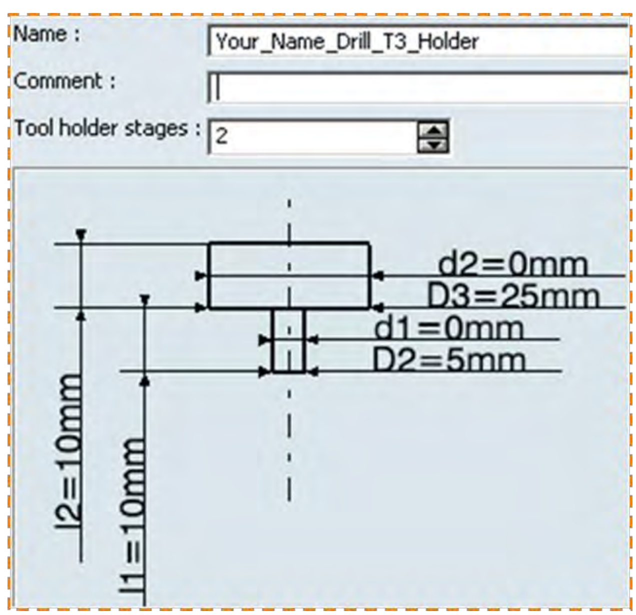
d. Repeat the previous step for „Retract" macro
e. Click „OK" to validate the drilling operation



35. Now we will create the tool for the last drilling operation

a. Create a drilling tool as indicated opposite
b. Then create the tool holder as indicated opposite
c. Assemble and name it as [Your_Name_Drill_T3]



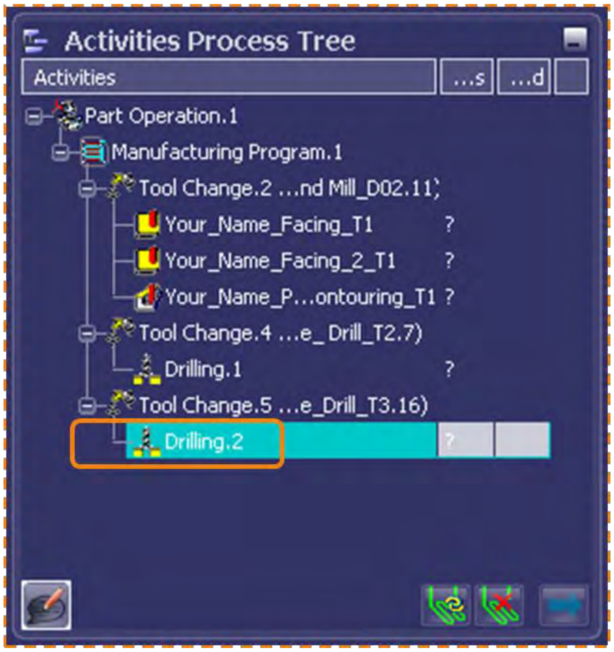


36. Now we will perform the last drilling operation

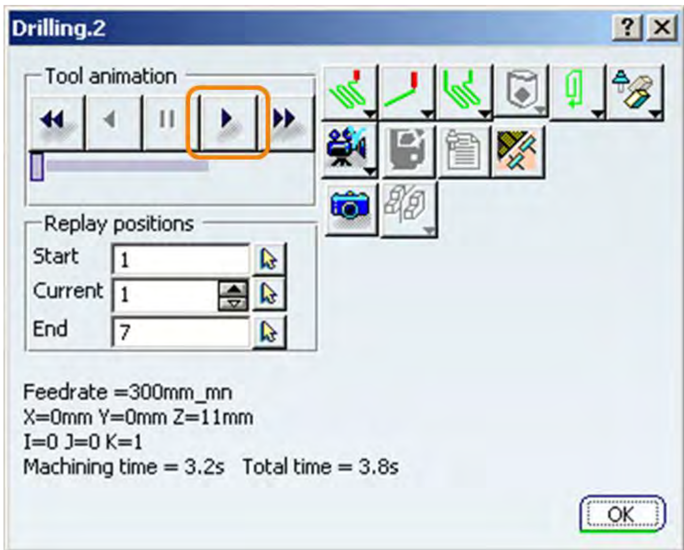
a. Repeat the drill operation previously created for the second hole
b. Keep the same parameters as the previous drilling operation. Replace „Your_Name_Drill_T2" tool with „Your_Name_Drill_T3"
c. Now we will perform the last drilling operation

37. We will now generate a tool path for each machining operation

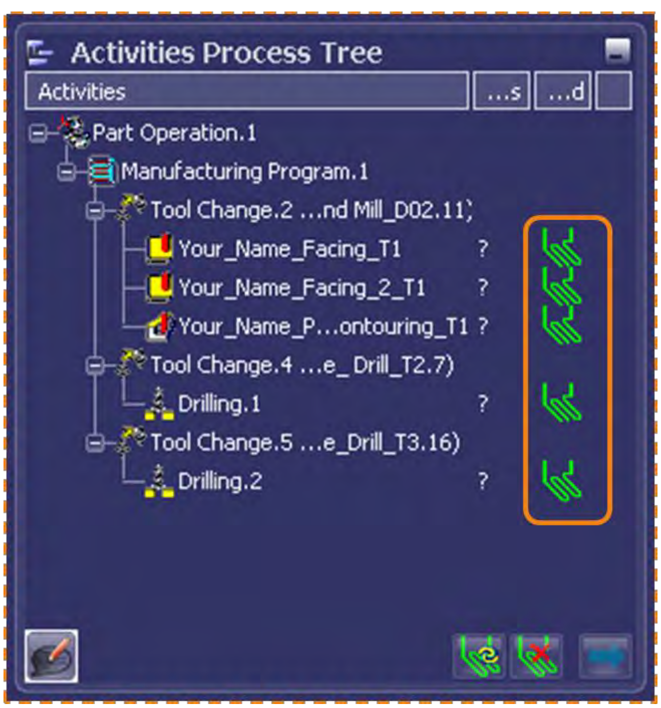
a. Click on the „Drilling.2"
b. Click on the „Replay Tool Path" icon



c. Click on the „Forward replay" icon

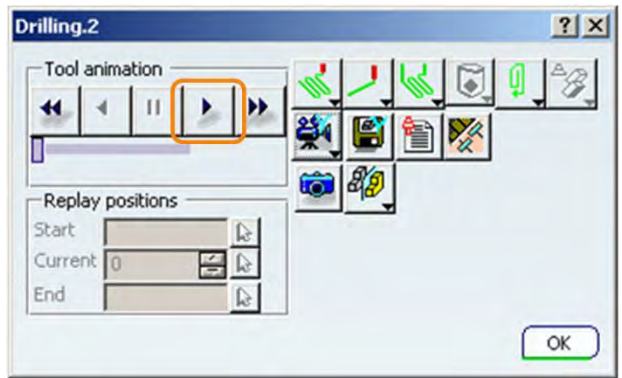


d. Repeat this operation for each operation



38. We will now generate a tool path for each machining operation

a. Click on the „Drilling.2‟
b. Click on the „Replay Tool Path‟ icon
c. Click on the „Video from last saved result‟ icon
d. Click on the „Forward replay‟ icon

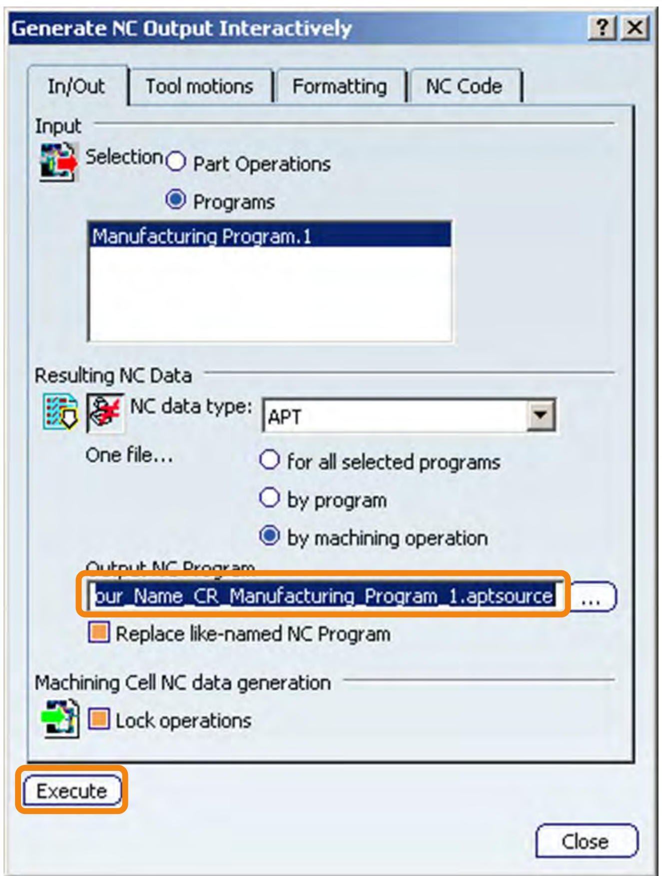


39. We will now generate the manufacturing program of the connecting rod

a. Click on the „Generate NC Code Interactively" icon

b. Rename it as [Your_Name_CR_Manufacturing_Program_1.aptsource]

c. Click on „Execute" to generate the program



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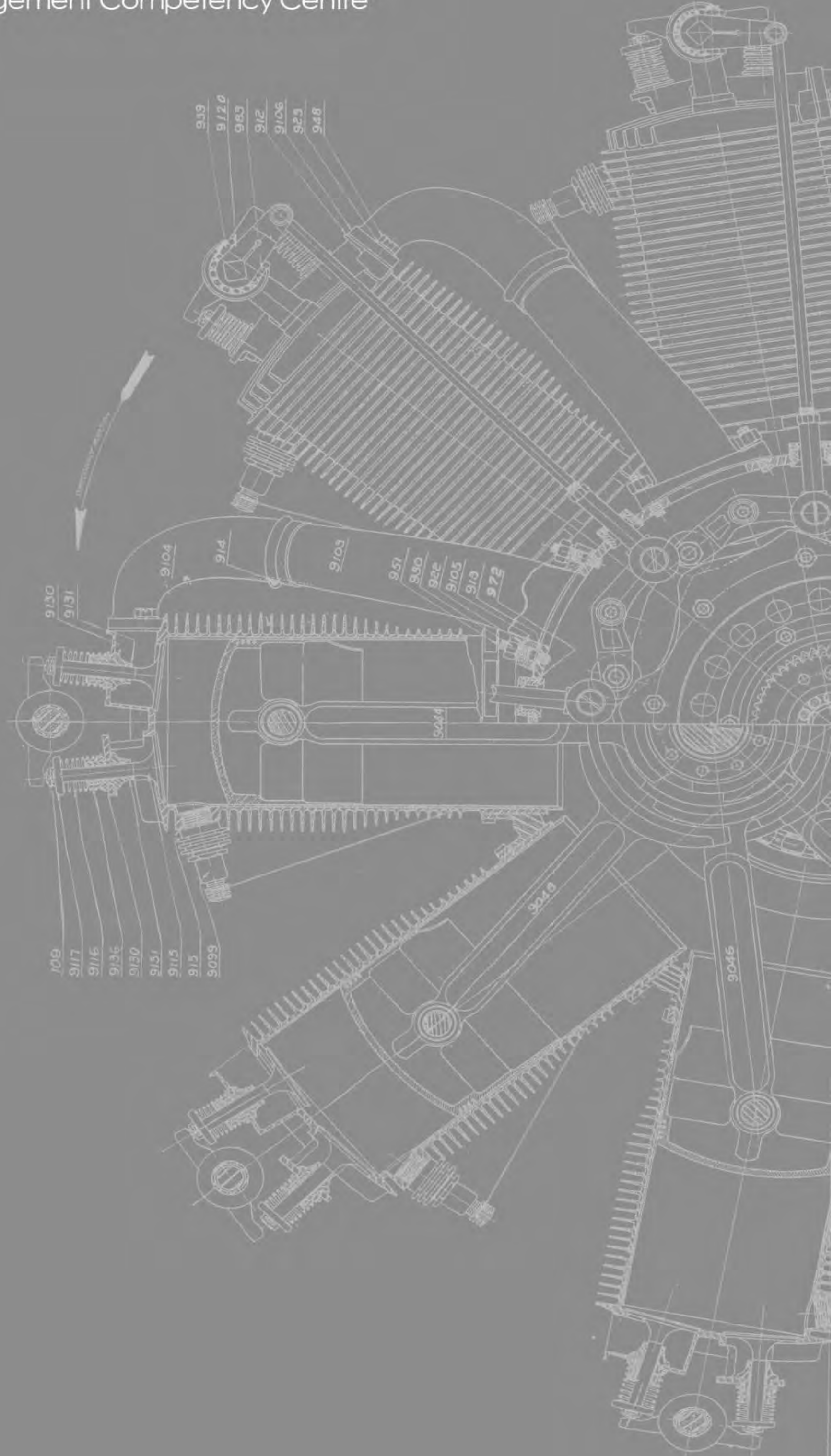
Product Lifecycle Management Competency Centre



Drafting of Part

DASSAULT SYSTEMES

Cape Peninsula University of Technology

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RÉPUBLIQUE FRANÇAISE

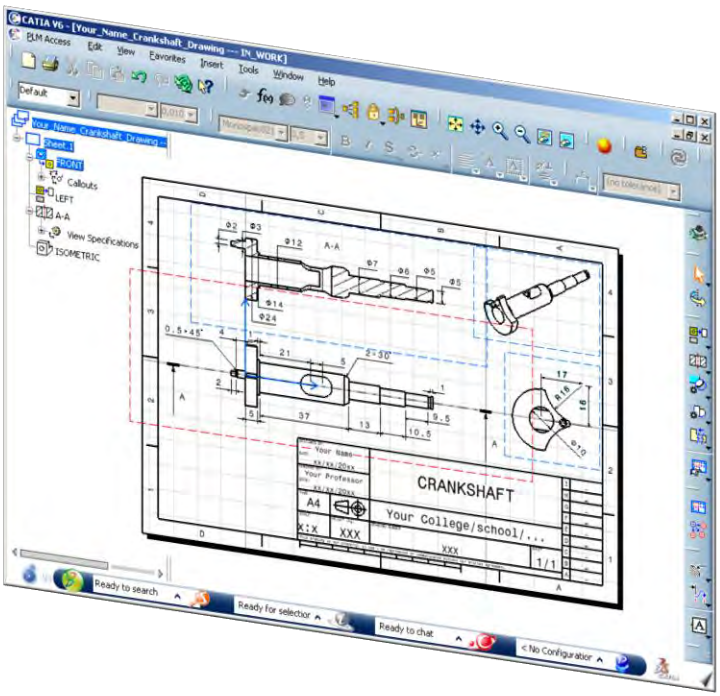
French ministry of education and research

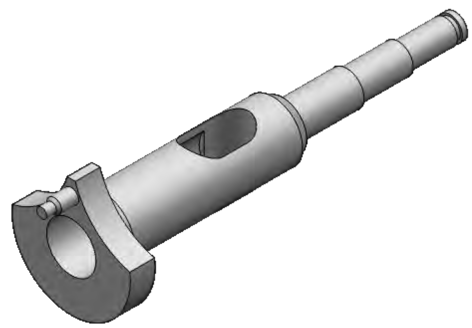
Drafting

During these steps, you will generate the crankshaft drawing.

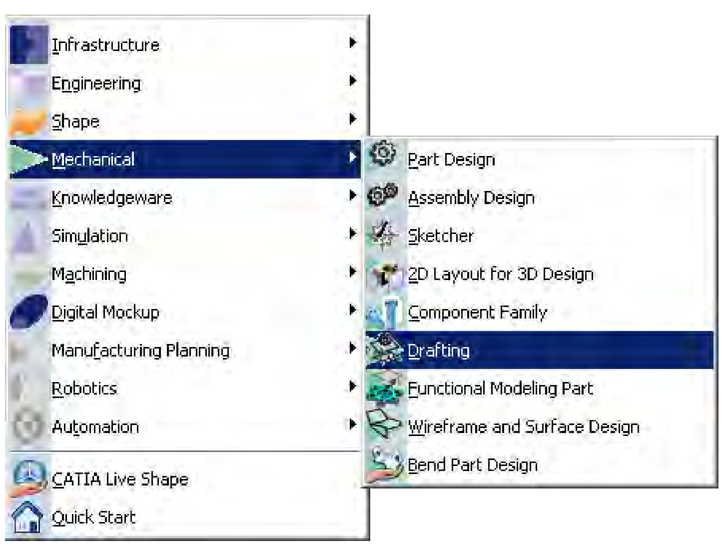
You can import the crankshaft provided by the courseware or use your own crankshaft previously designed

Drafting provides functionalities to generate drawings from 3D parts and assembly definitions

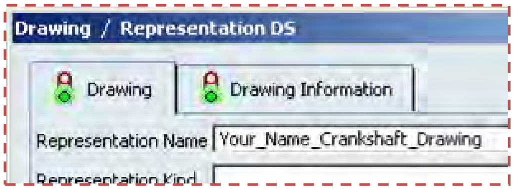


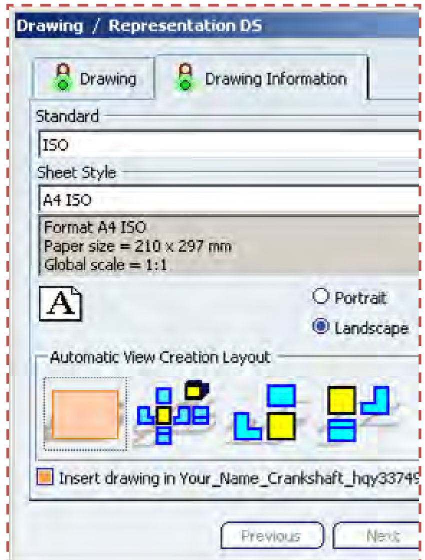


1. If it hasn‟t been done already, download and import the Crankshaft. Then open it
2. To access „**Drafting**‟ click „**Start**‟ in the bar **>** „**Mechanical Design**‟ **>** „**Drafting**‟



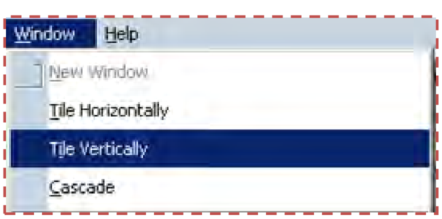
3. Under the „**Drawing**" tab, enter-„Your_Name_Crankshaft_Drawing" as „**Representation Name**"
4. Under the „**Drawing Information**" tab
   a. Select „**ISO**" as „**Standard**"
   b. Select „**A4 ISO**" as „**Sheet Style**"
   c. Check „**Landscape**"
   d. Select „**Empty Sheet**" as „**Automatic View Creation Layout**"

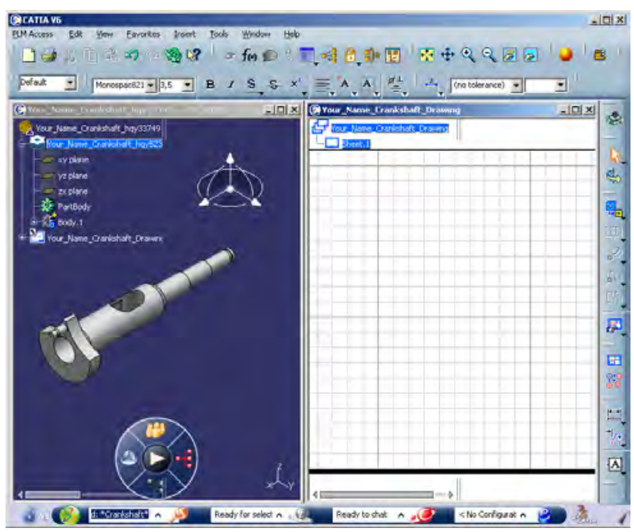




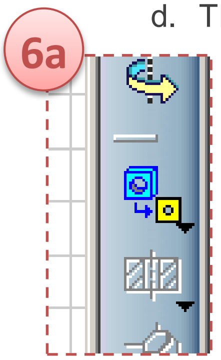
5. We will separate the window into 2 parts

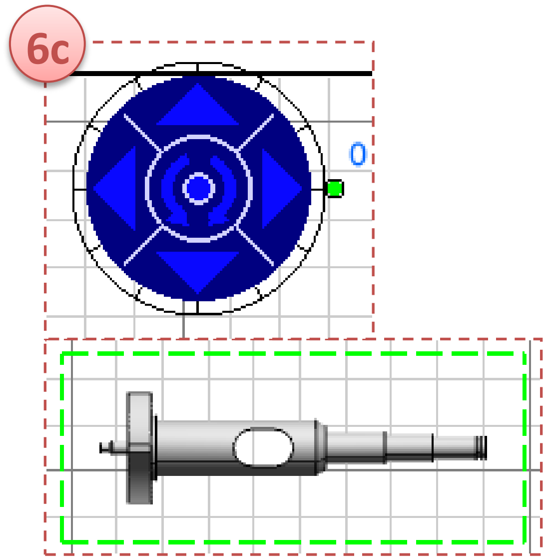
Select „**Windows**" Menu **>** „**Tile Vertically**

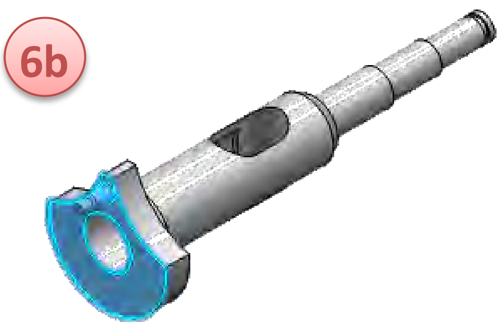


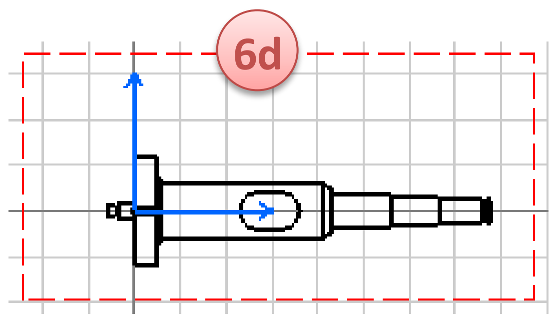


6. Create the front view
    a. Click on the „**Front view**" icon
    b. Select the face of the crankshaft as shown opposite.
    c. Use arrows to orientate the view as indicated opposite
    d. Then click anywhere on the sheet to validate the view



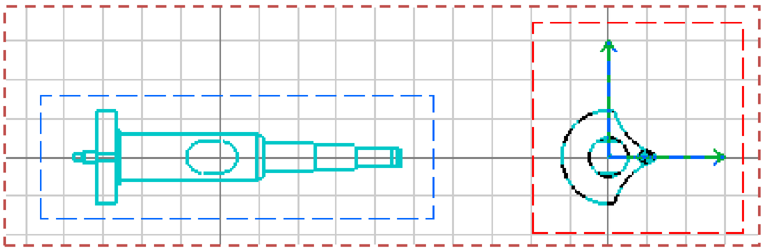






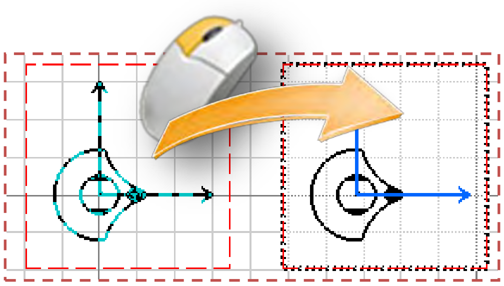
To activate a view:

Double click on the frame of the view you want to activate



To move a view:

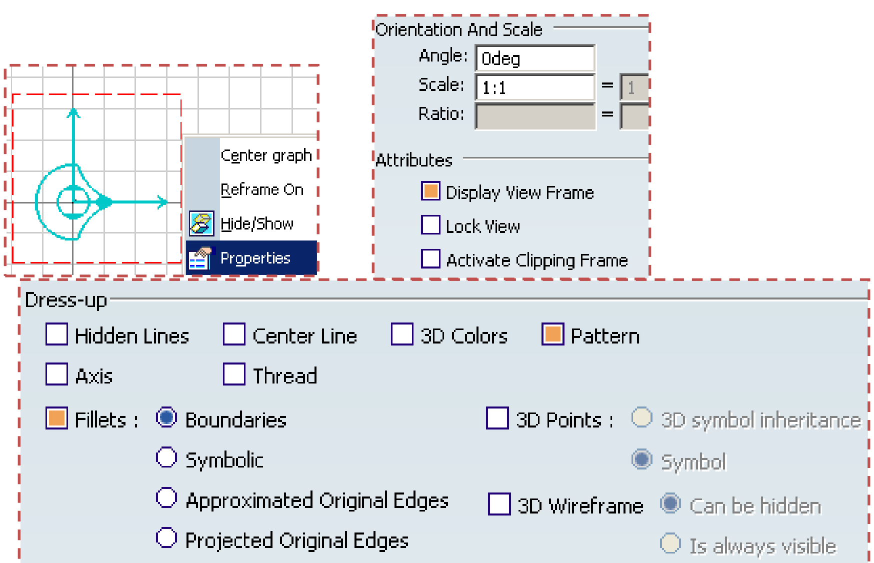
Drag and drop the frame of the view



To modify view properties:

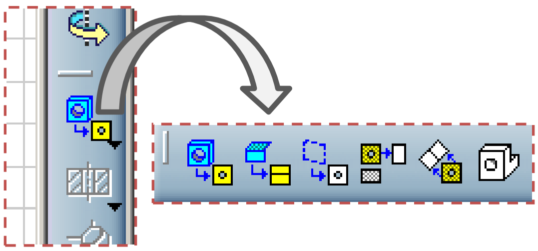
Right click on the frame of the view > „**Properties**"

You can modify the scale and the dress up

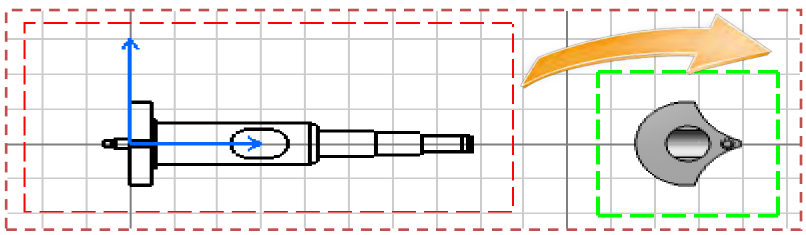


7. We will now create the left view

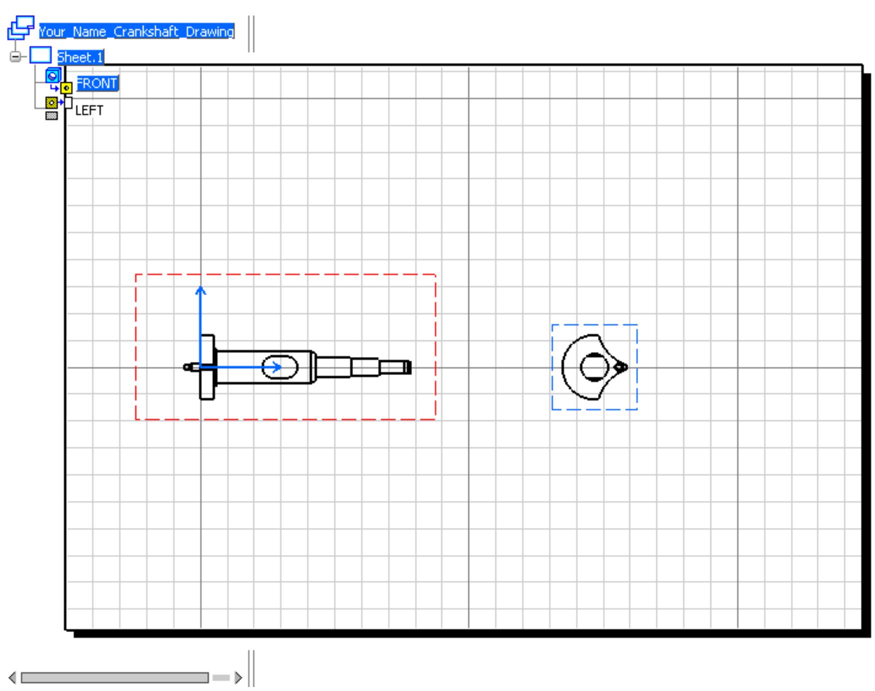
   a. Click on the „**Projection View**“ icon



   b. Move the mouse on the right of the front view

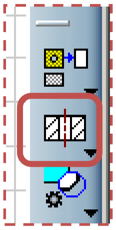


   c. Click anywhere on the sheet to validate the view

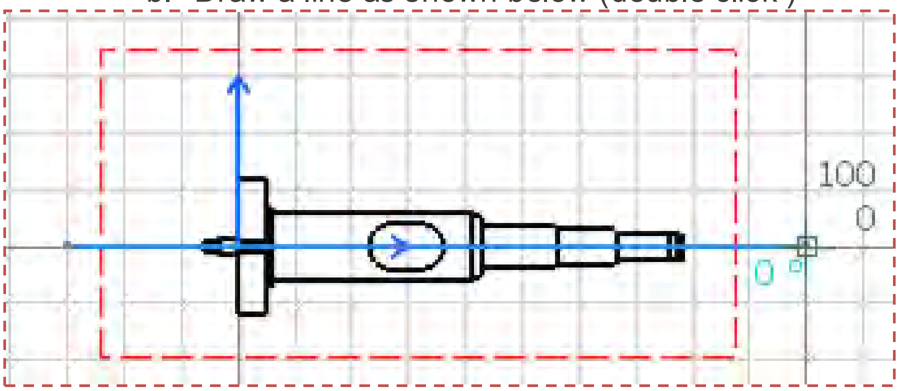


8. We will now create an offset section view

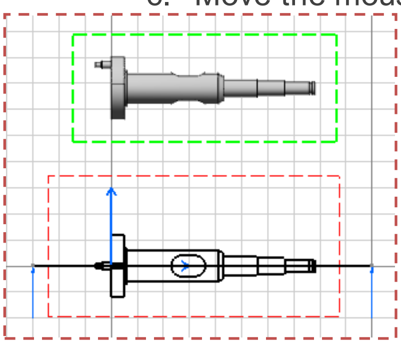
   a. Click on the „**Offset Section View**" icon



   b. Draw a line as shown below (double click)



   c. Move the mouse upper the front view



   d. Click anywhere on the sheet to validate the view

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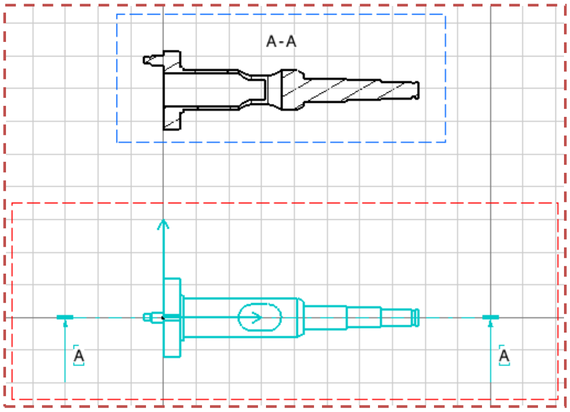
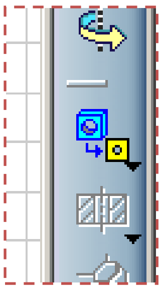
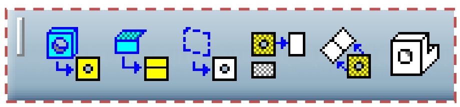
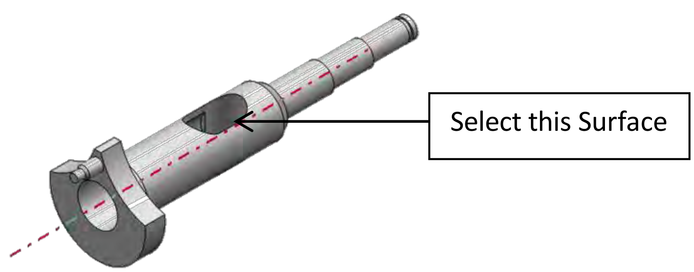


9. We will add an isometric view
   a. Click on the „**Isometric View**" icon

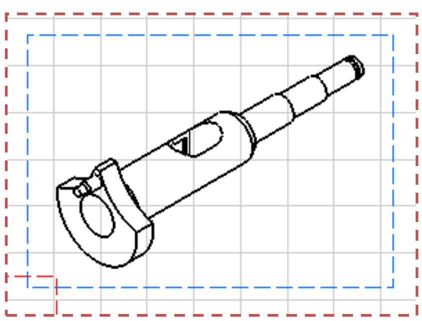




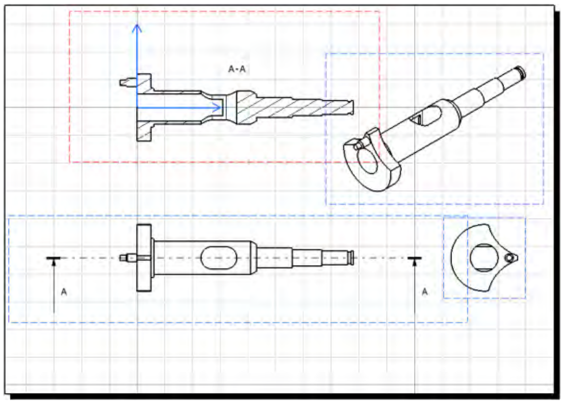
   b. Select a surface of the crankshaft in 3D



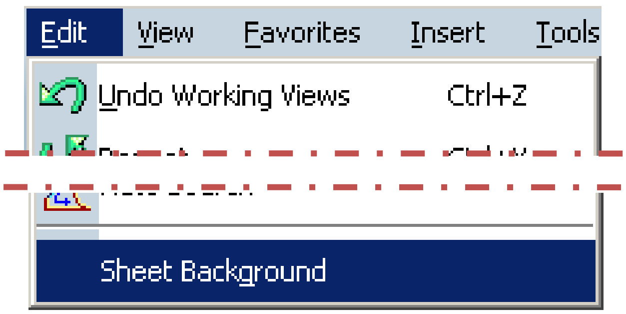
   c. Position the view on the sheet and click anywhere to validate it



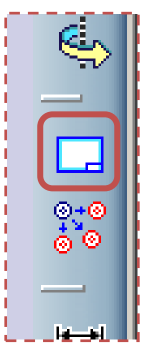
10. Modify the Scale of all the views to [3:2]

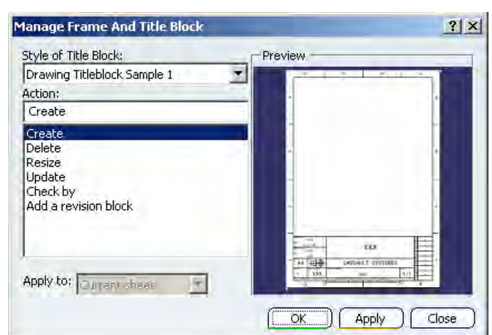


11. We will now insert a background to the drawing sheet
    a. Select „**Edit**" Menu **>** „**Sheet background**"



    b. Click on the „**Frame and Title Block**" icon

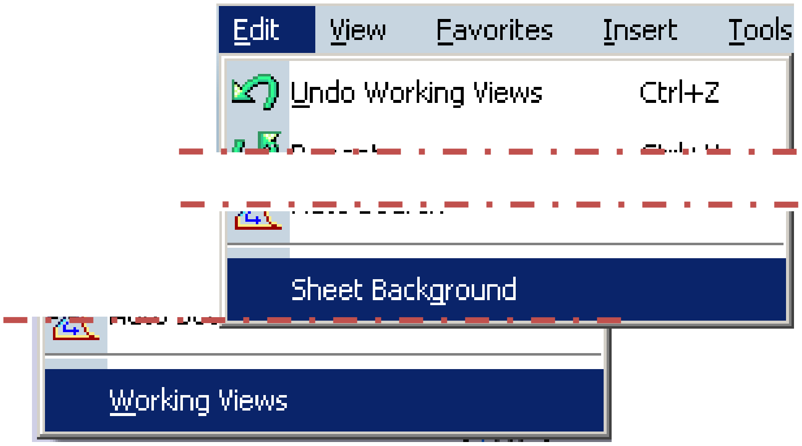




    c. Select „**Drawing Titleblock Sample 1**" as „**Style of Title Block**"

    d. Select „**Create**" as „**Action**"

To activate the sheet background or to work on views:

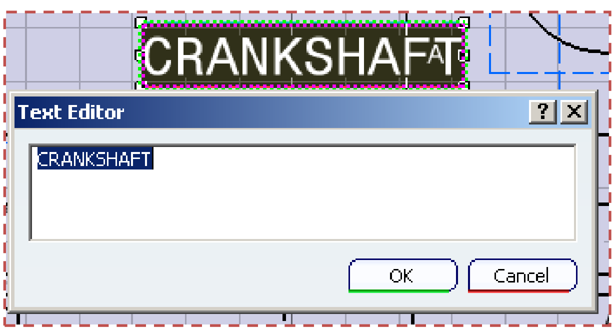


Select „**Edit**" Menu **>** „**Sheet background**" if you are in „**Working view**" mode

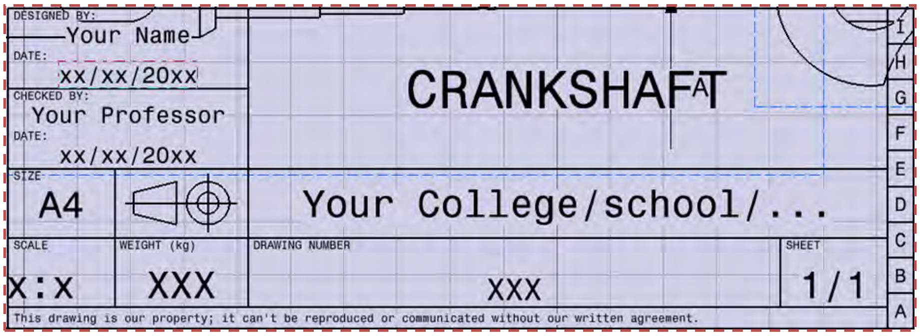
Select „**Edit**" Menu **>** „**Working view**" if you are in „**Sheet background**" mode

To edit the frame:

Double click on the text to modify it

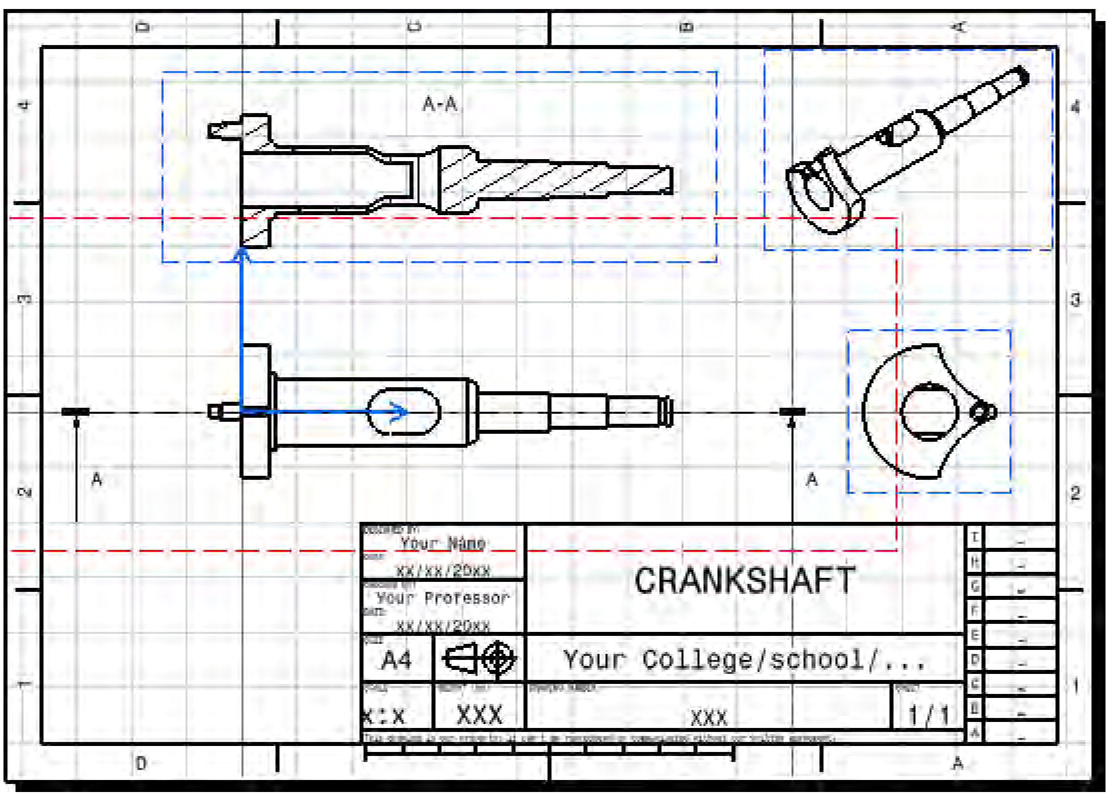


12. Fill the Frame as indicated:



Adjust the view to the new frame

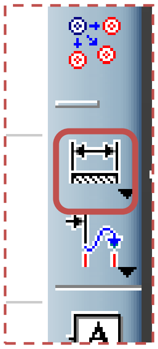
a. Return to the „**Working Views**“ mode
b. Change the scale of the „isometric view“ as [1:1]
c. Move all the views as shown opposite.



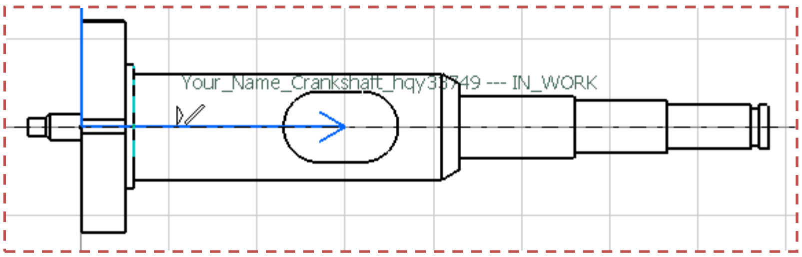
14. We will now add the dimensions to the front view

a. Activate the „FRONT“ view if it has not already done

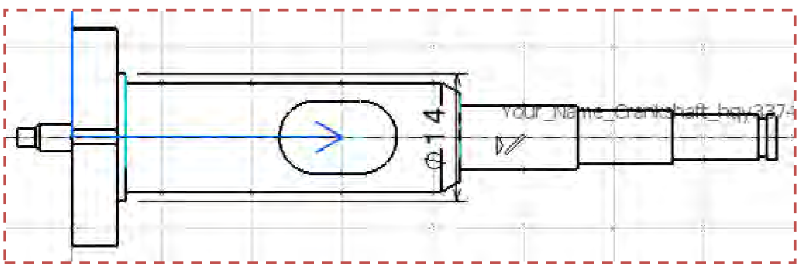
b. Click on the „**Dimensions**“ icon



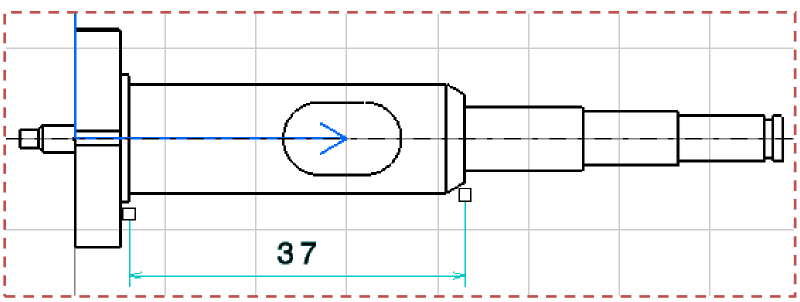
c. Select the first edge



d. Select the second edge



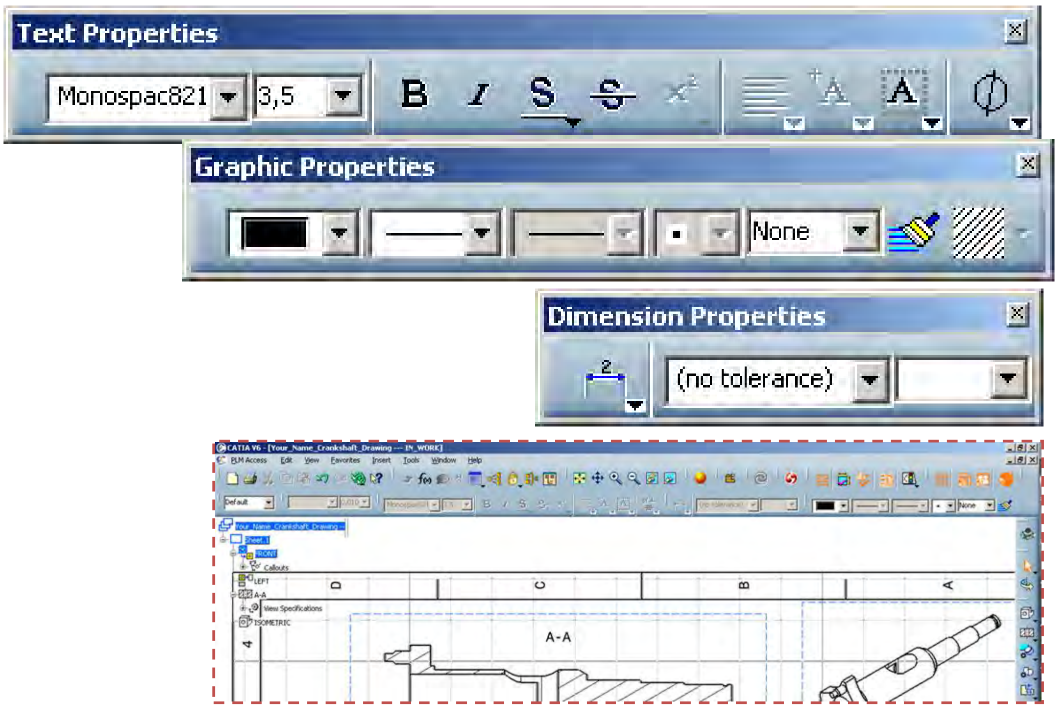
e. Click to validate the position of the dimension



Drafting of a part - Dimensions

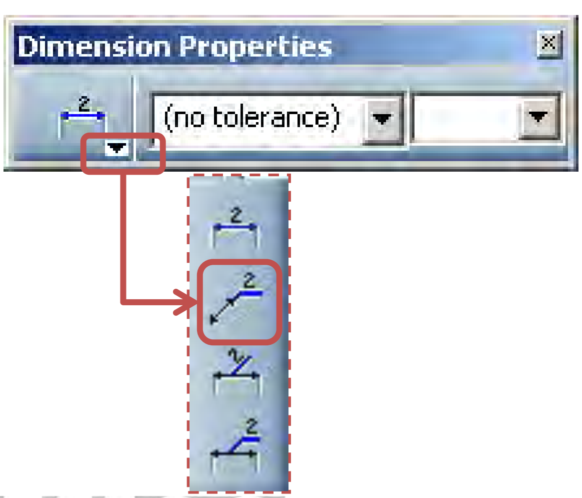
On the top of the window, you can find some tools to customize the dimensions

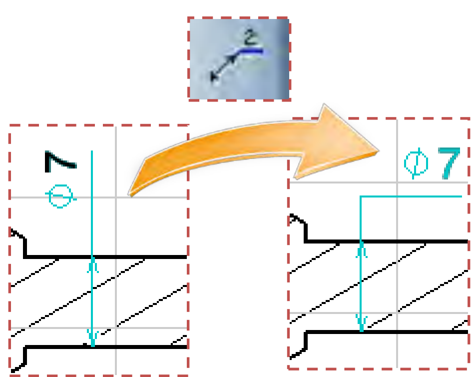
- Text Properties" tools
- Graphic Properties" tools
- Dimension Properties" tools



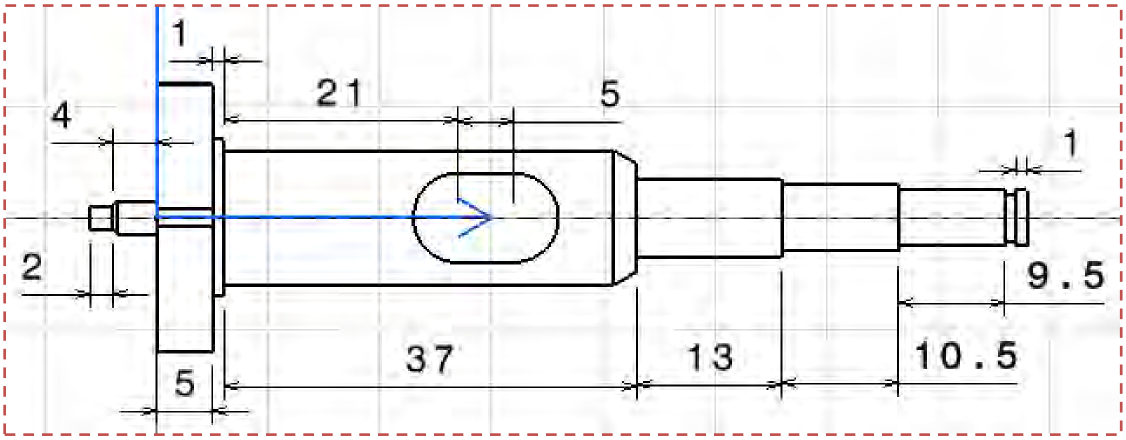
You can modify the dimension line during or after the dimension creation

- Click on the „**Dimension Line**" icon
- Select a type of line

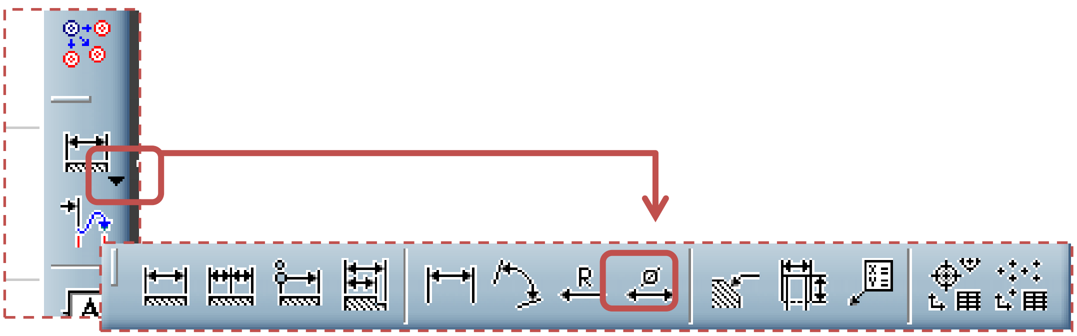




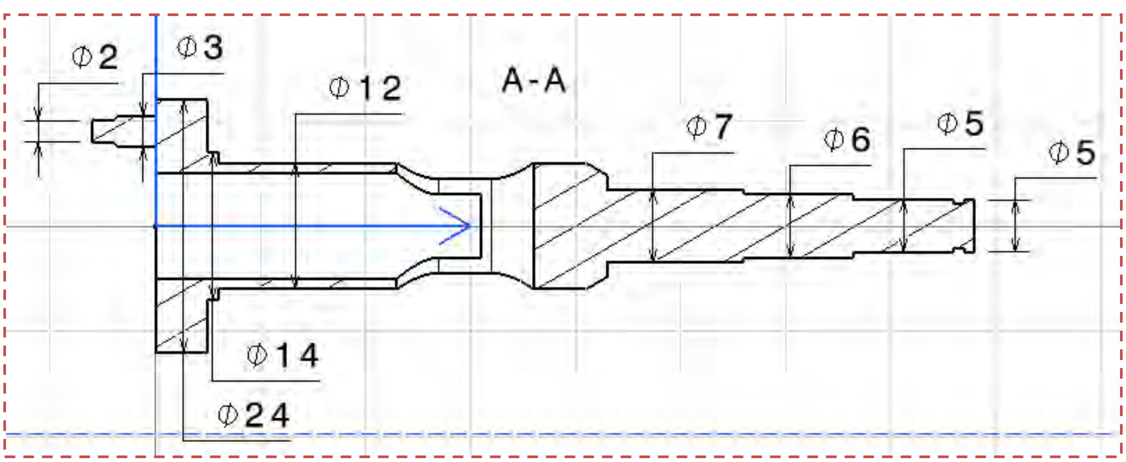
15. Repeat the previous steps to add the dimensions to the „Front" view as indicated opposite



16. We will add the dimensions to the „A-A" view
    a. Click on the „**Diameter Dimensions**" icon

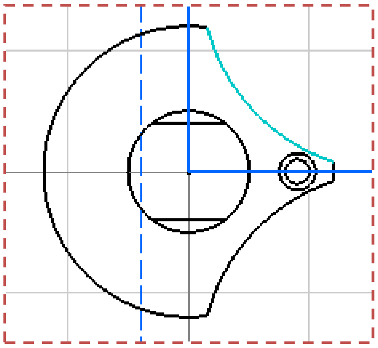


    b. Select an edge of a cylindrical surface
    c. Click to validate the position of the dimension

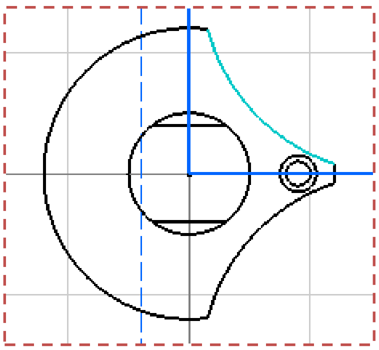


We will add the dimensions to the „LEFT" view

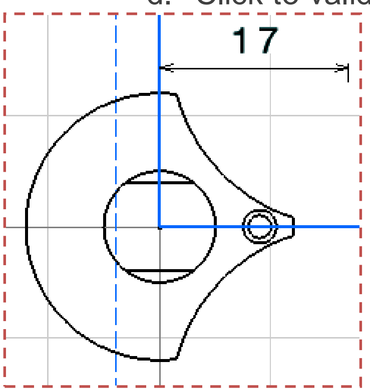
17. Horizontal dimensions
    a. Click on the „**Dimensions**" icon
    b. Select this arc



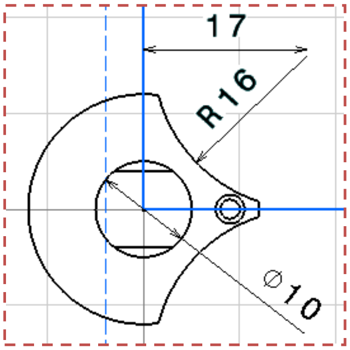
    c. Select the center of the hole



    d. Click to validate the position of the dimension

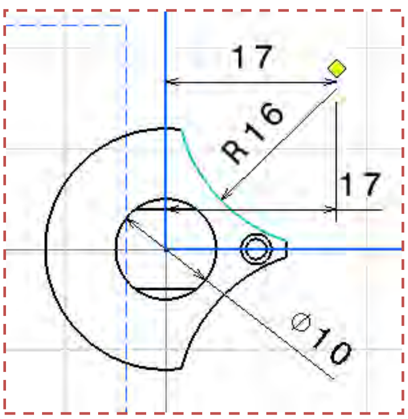


18. Add radius and diameter dimensions as shown opposite

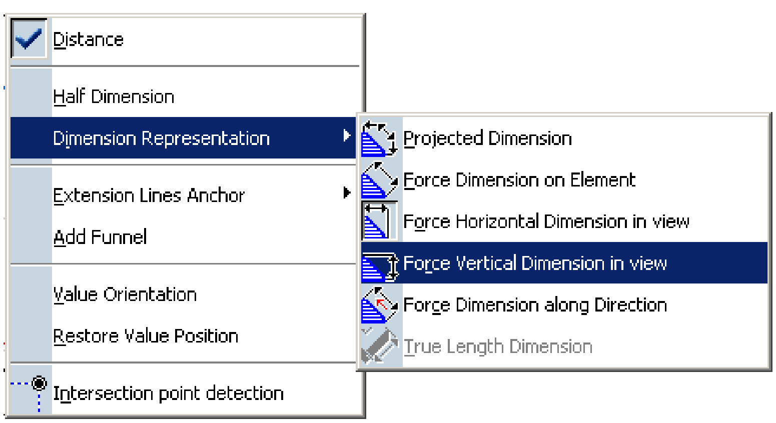


19. Vertical dimension

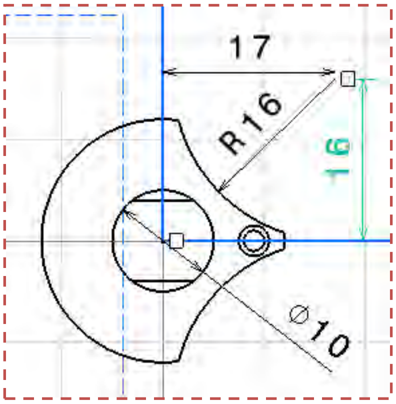
a. Click on the „**Dimensions**" icon
b. Select this arc
c. Select the center of the hole
d. Right click on the dimension



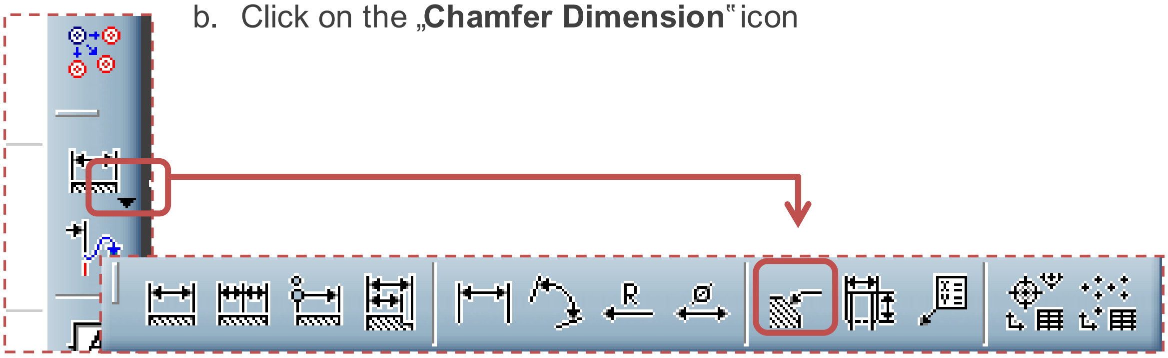
e. Select „**Dimension Representation**" > „**Force Vertical Dimension in view**"



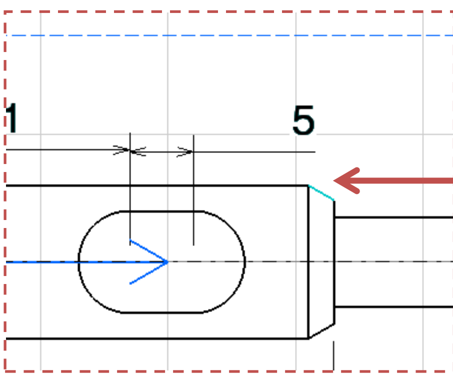
f. Click to validate the position of the dimension



20. To finish we will add the dimensions to the 2 chamfer on the front view
    a. Activate the „FRONT" view
    b. Click on the „**Chamfer Dimension**" icon



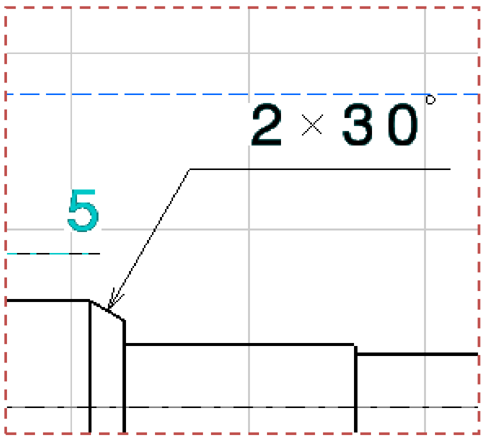
    c. Select the edge of the chamfer



Select this edge

    d. Select „**Length x Angle**"

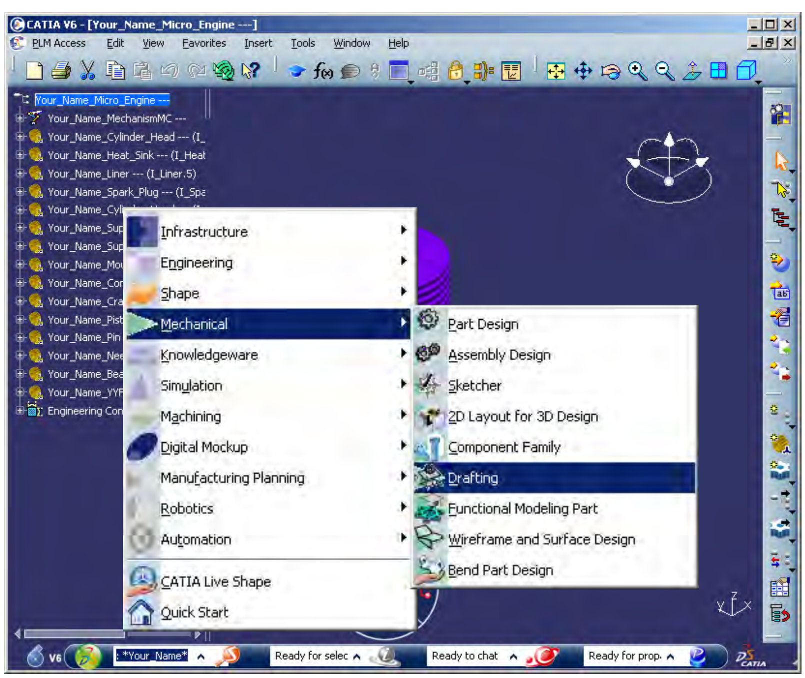
e. Click to validate the position of the dimension



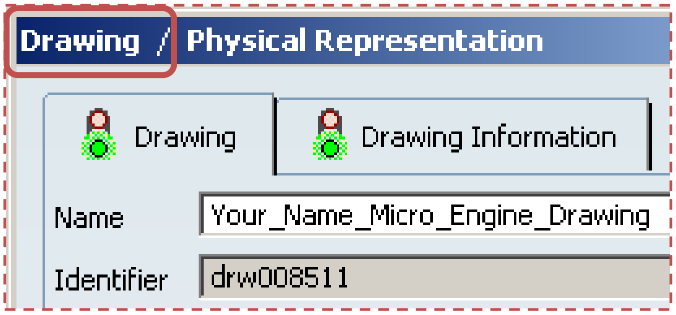
21. Don‟t forget to „Propagate" your work to save it in the data base

Drafting of a product

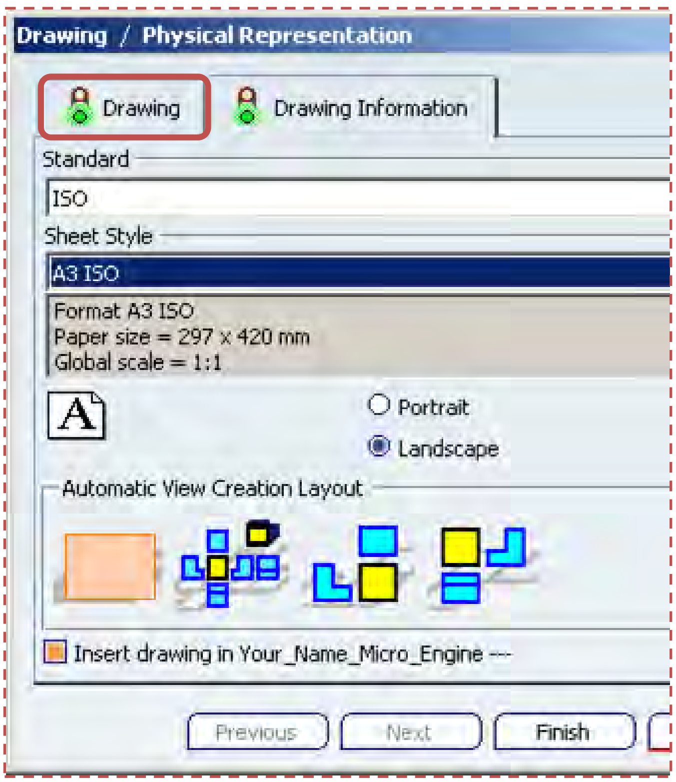
1. If it hasn‟t been already done, download and import the Micro Engine. Then open it
2. To access „**Drafting**" click „**Start**" in the bar **>** „**Mechanical Design**" **>** „**Drafting**"



3. Under the „**Drawing**" tab, enter „Your_Name_Micro_Engine_Drawing" as „**Representation Name**"

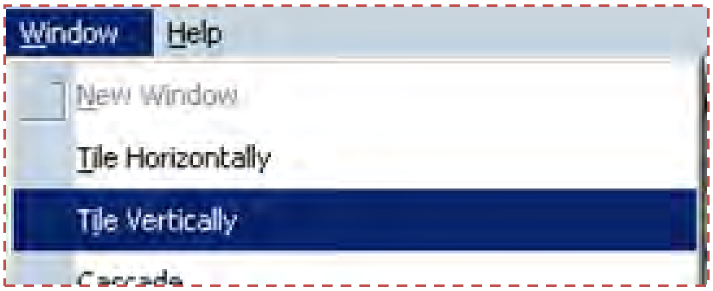


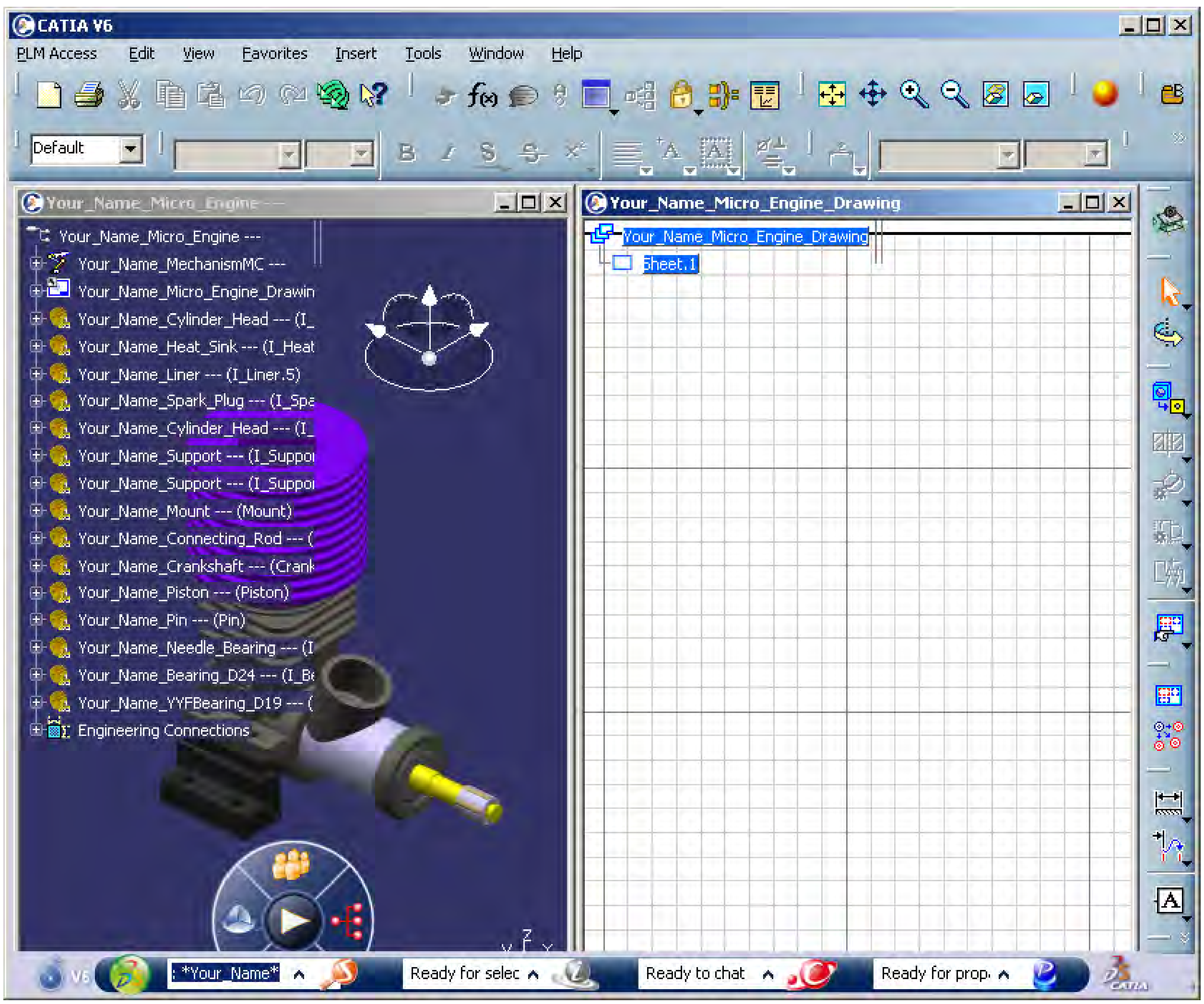
4. Under the „**Drawing Information**" tab
   a. Select „**ISO**" as „**Standard**"
   b. Select „**A3 ISO**" as „**Sheet Style**"
   c. Check „**Landscape**"
   d. Select „**Empty Sheet**" as „**Automatic View Creation Layout**"
   e. Click „Finish" to validate the drawing informations



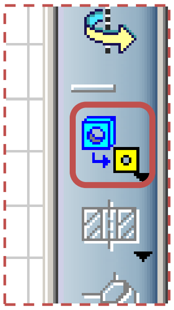
5. We will separate the window into 2 parts

    a. Select „**Windows**“ Menu **>** „**Tile Vertically**“

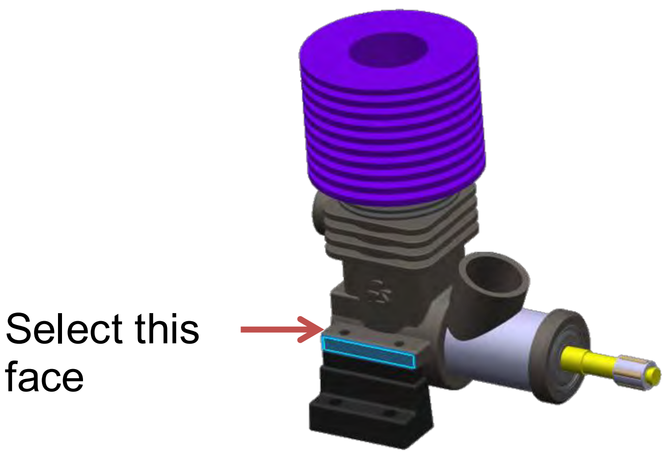




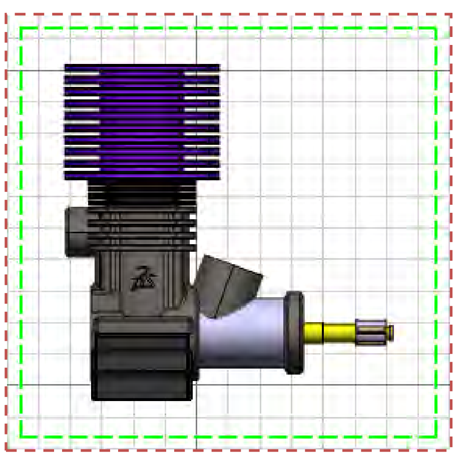
6. Create the front view
    a. Click on the „**Front view**" icon

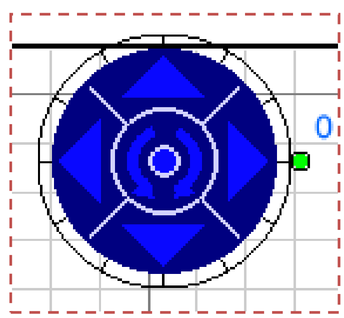


    b. Select the face of the crankshaft as shown opposite.

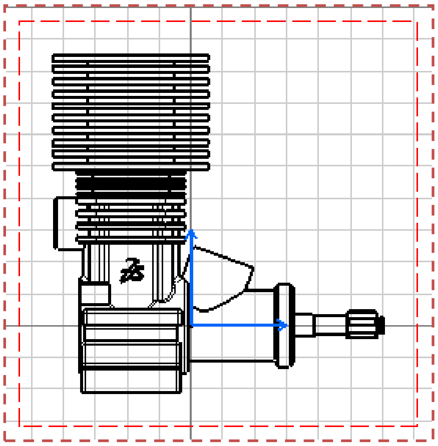


    c. use the arrows to orientate the view as indicated opposite, if you need to.





d. Then click anywhere on the sheet to validate the view

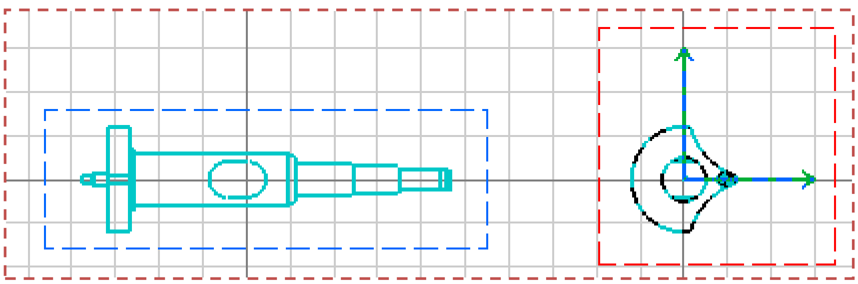


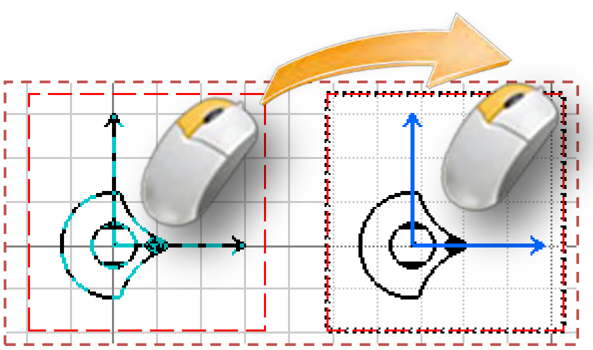
To activate a view

Double click on the frame of the view you want to activate

To move a view

Drag and drop the frame of the view

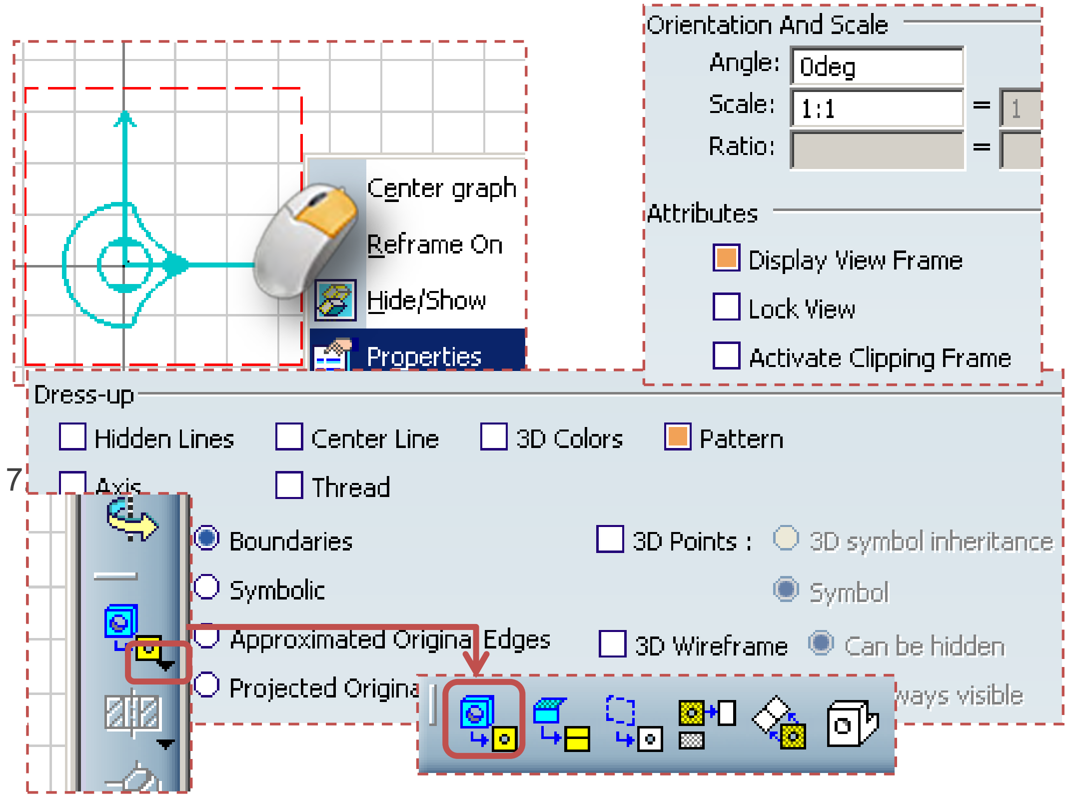




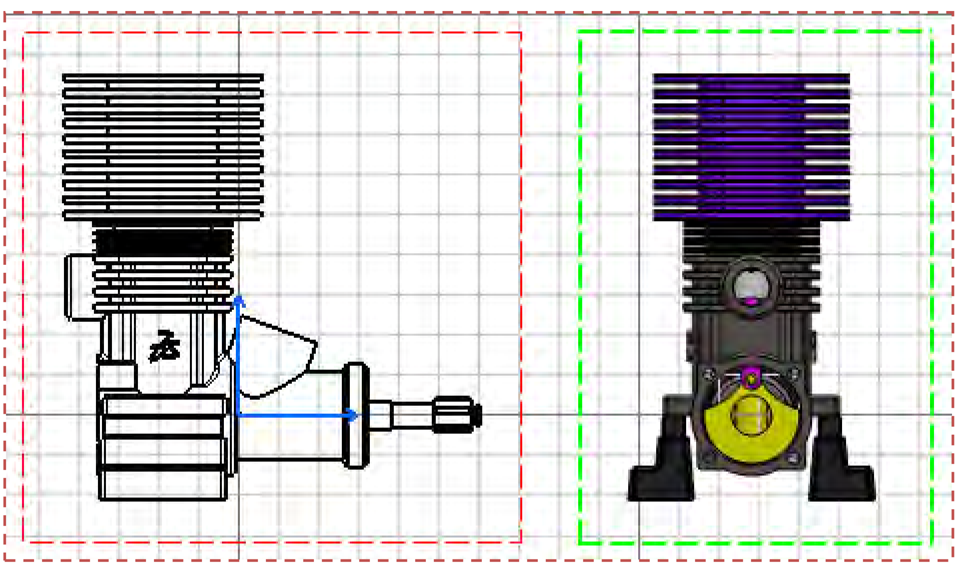
To modify the view properties

Right click on the frame of the view > „**Properties**"

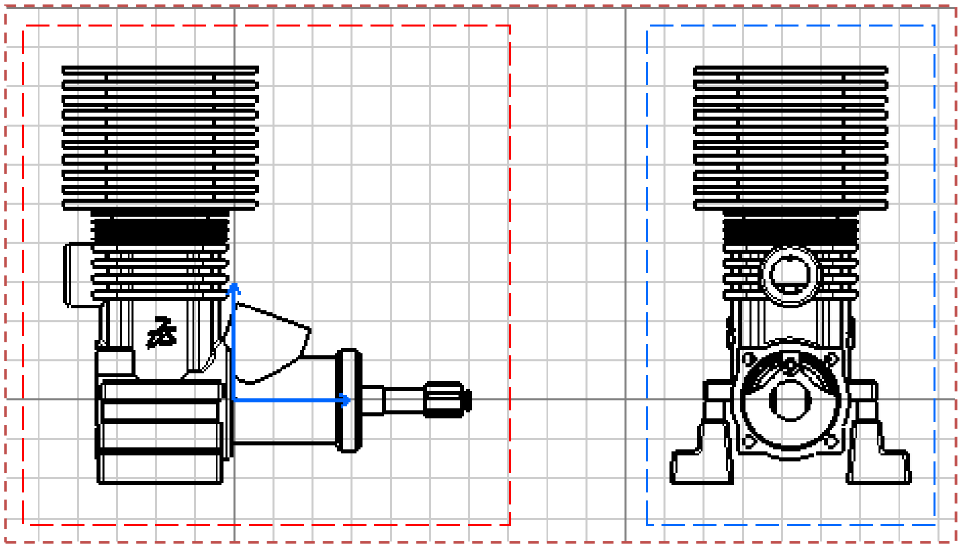
You can modify the scale, the dress up



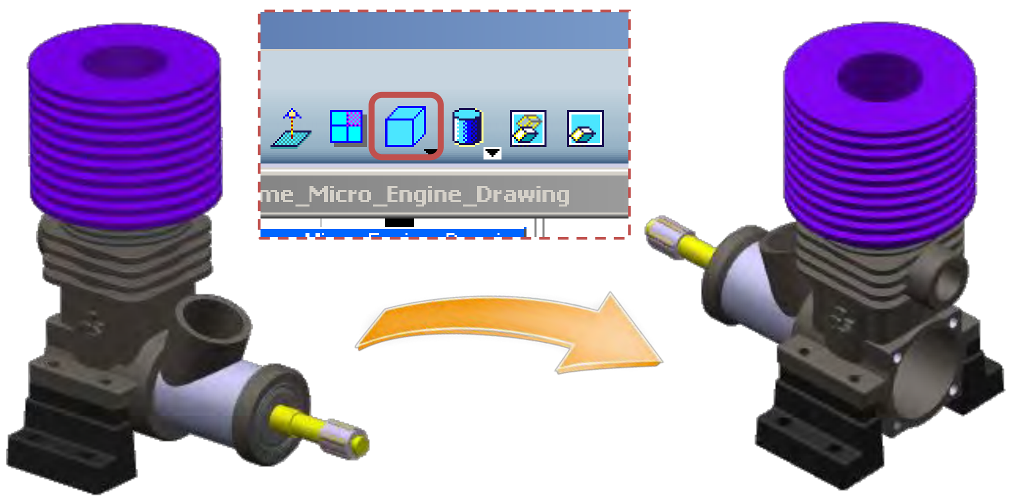
b. Move the mouse on the right of the front view



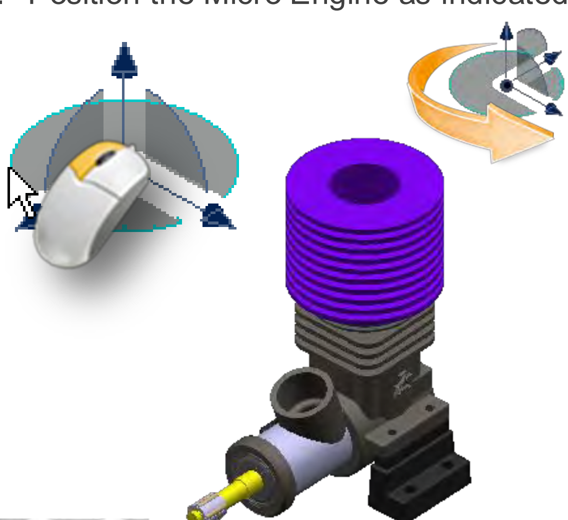
c. Click anywhere on the sheet to validate the view



9. We will position the Micro Engine for the Isometric View
   a. Click on the „Isometric View“ icon

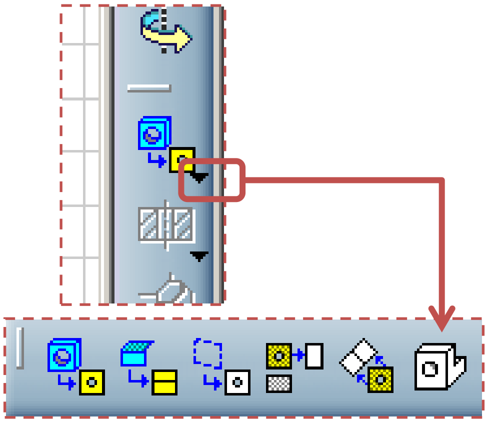


   b. Click on the bassement of the robot
   c. Position the Micro Engine as indicated opposite

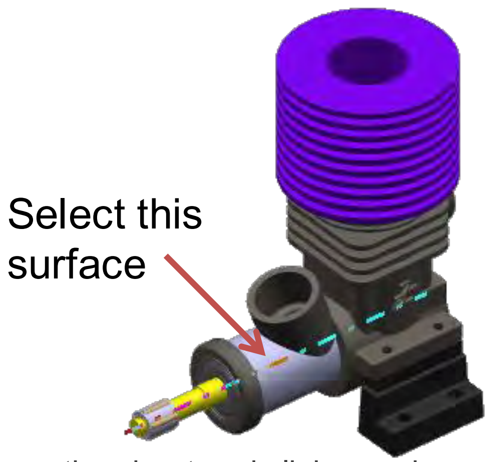


10. We will add an isometric view

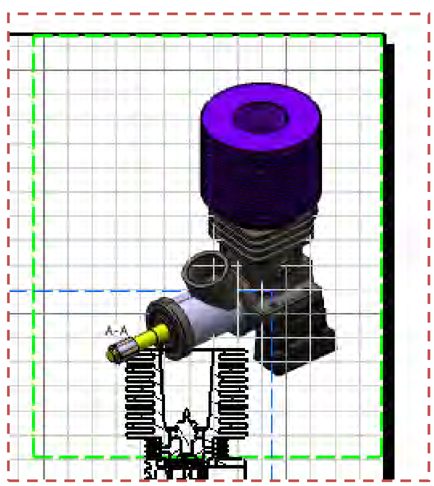
a. Click on the „**Isometric View**" icon



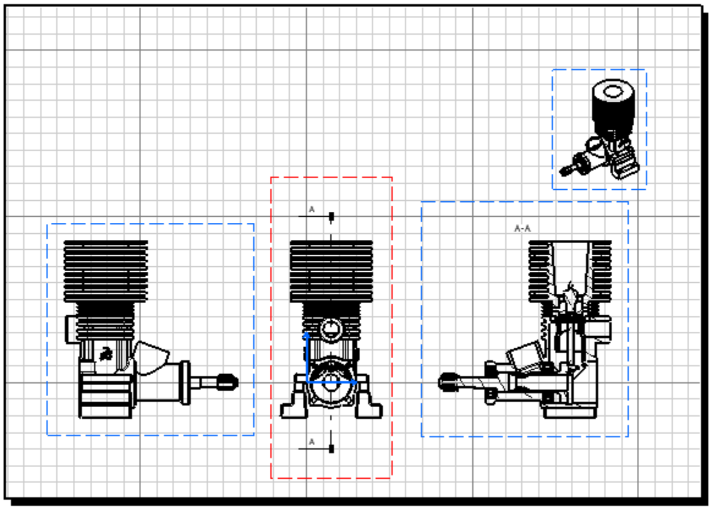
b. Select the surface in 3D as indicated opposite



c. Position the view on the sheet and click anywhere to validate it

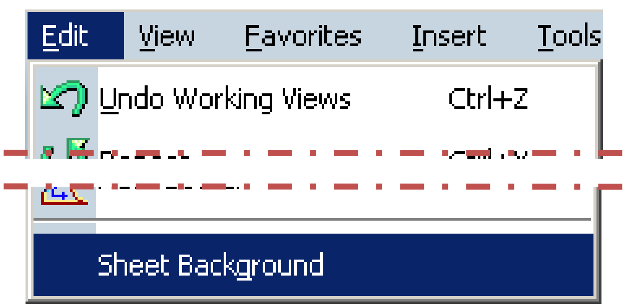


11. Modify the Scale of all the views to [1:2]

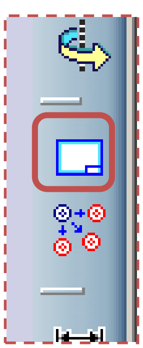


12. We will now insert a background to the drawing sheet

a. Select „**Edit**" Menu **>** „**Sheet background**"

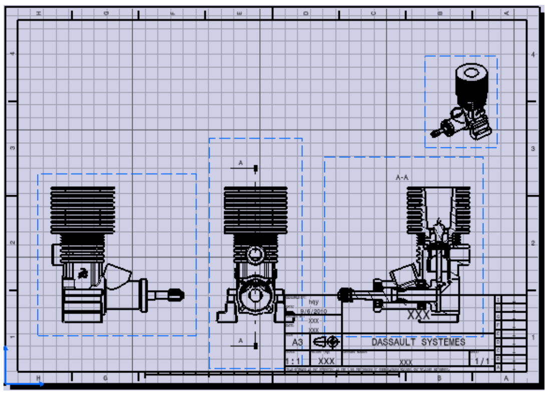


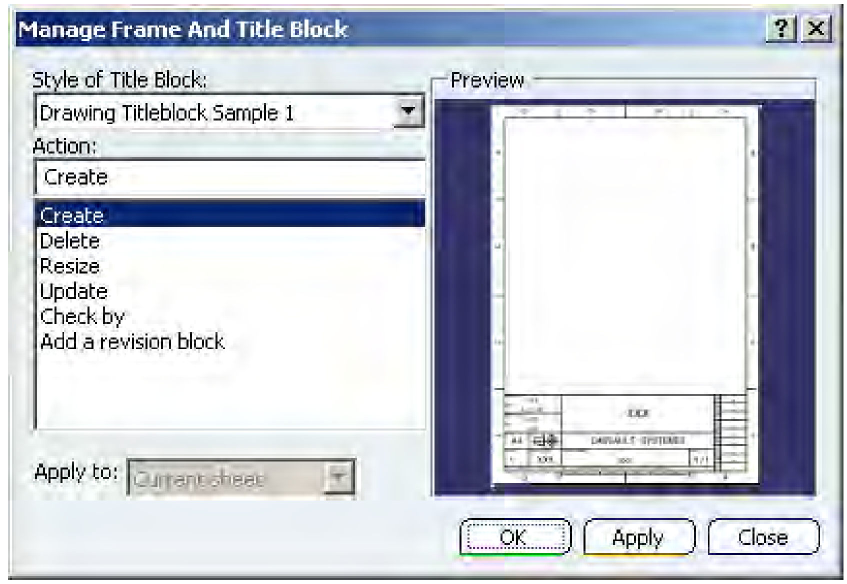
b. Click on the „**Frame and Title Block**" icon



c. Select „**Drawing Titleblock Sample 1**" as „**Style of Title Block**"

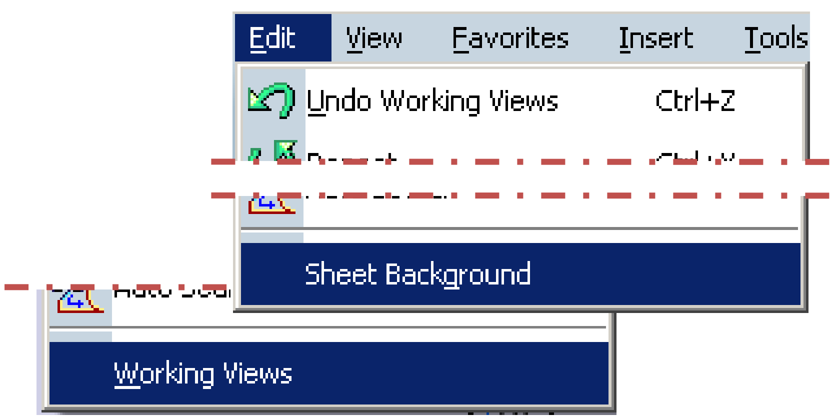
d. Select „**Create**“ as „**Action**“



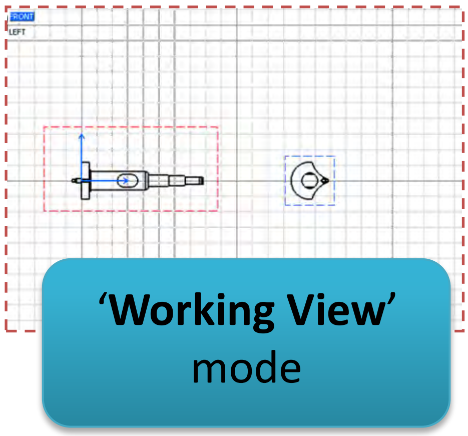


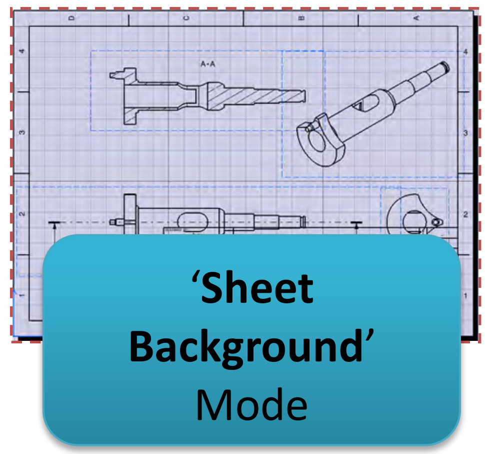
To activate the sheet background mode or to work on views*

Select „**Edit**“ Menu **>** „**Sheet background**“ if you are in „**Working view**“ mode



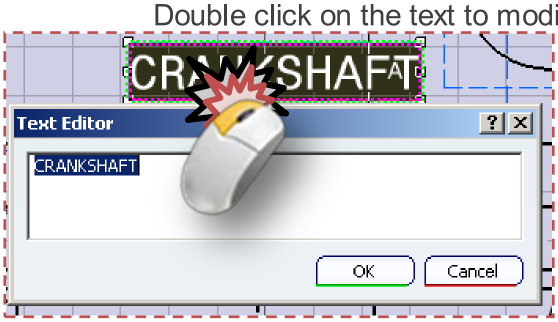
Select „**Edit**" Menu **>** „**Working view**" if you are in „**Sheet background**" mode



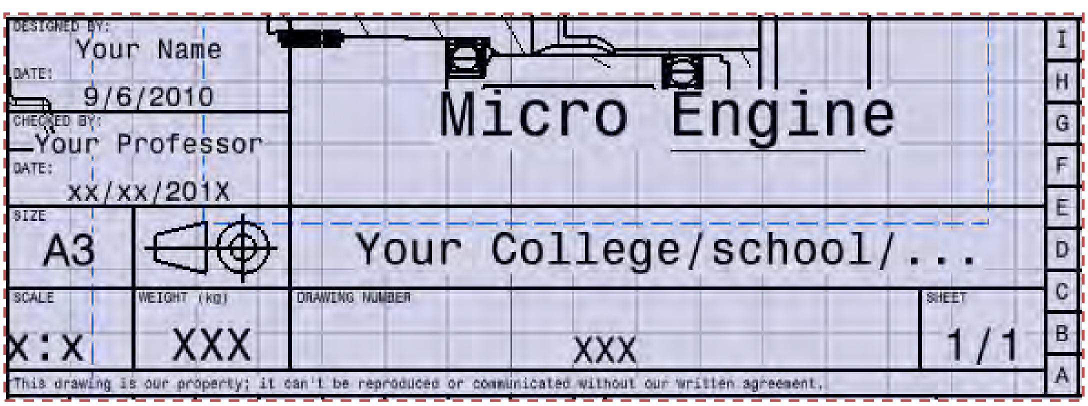


To edit the frame

Double click on the text to modify it



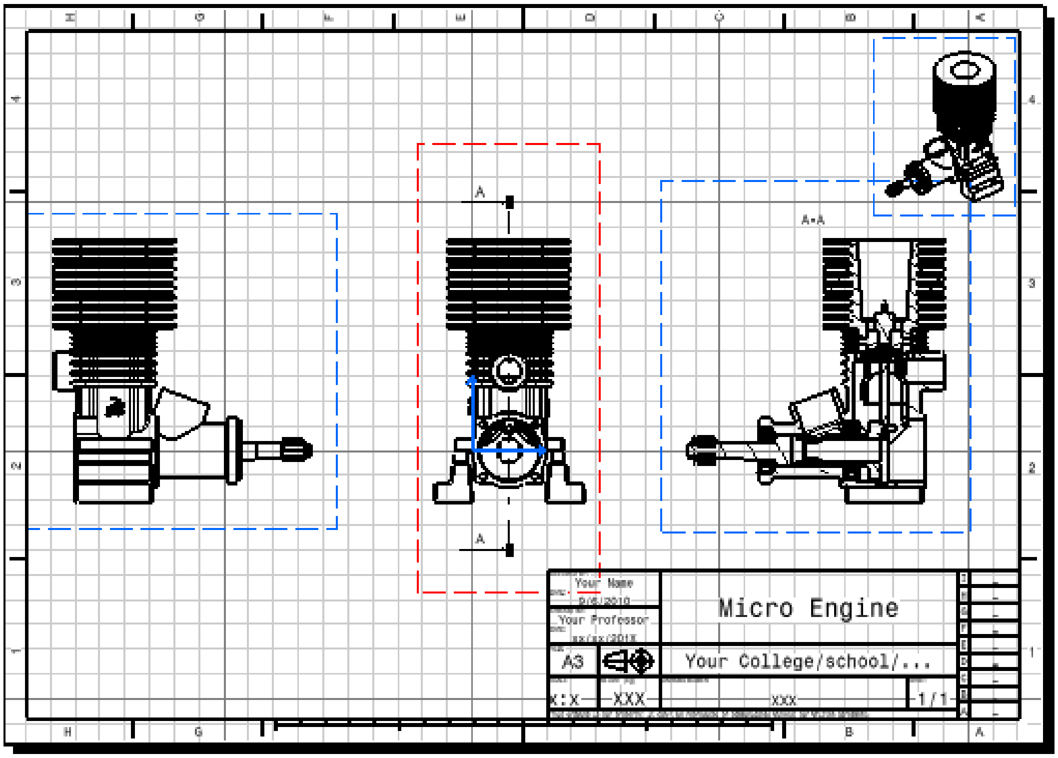
13. Fill the Frame as indicated opposite



14. Adjust the view to the new frame

   a. Return to the „**Working Views**“ mode

   b. Move all the views as shown below.



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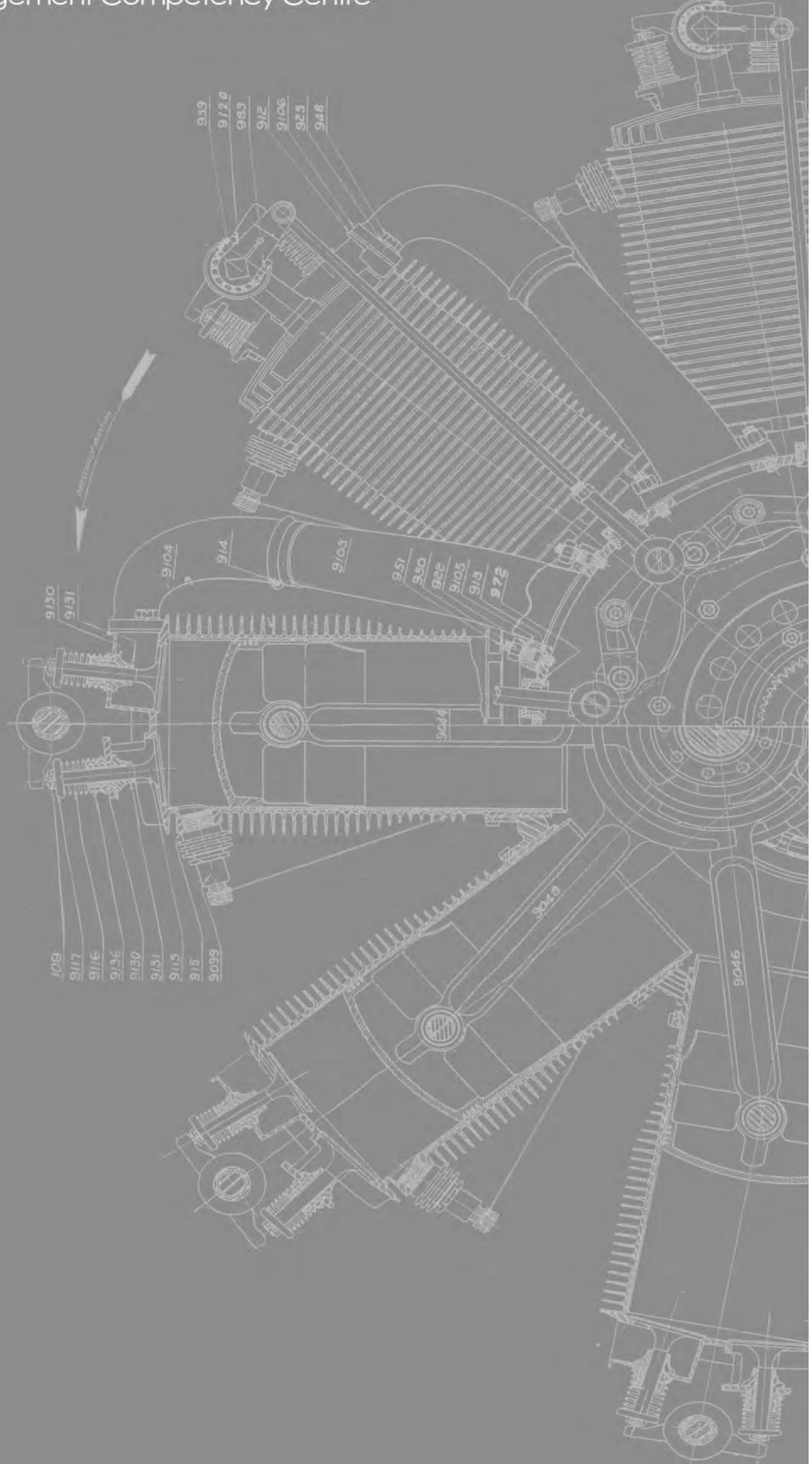

Drafting

Exercises

DASSAULT SYSTEMES
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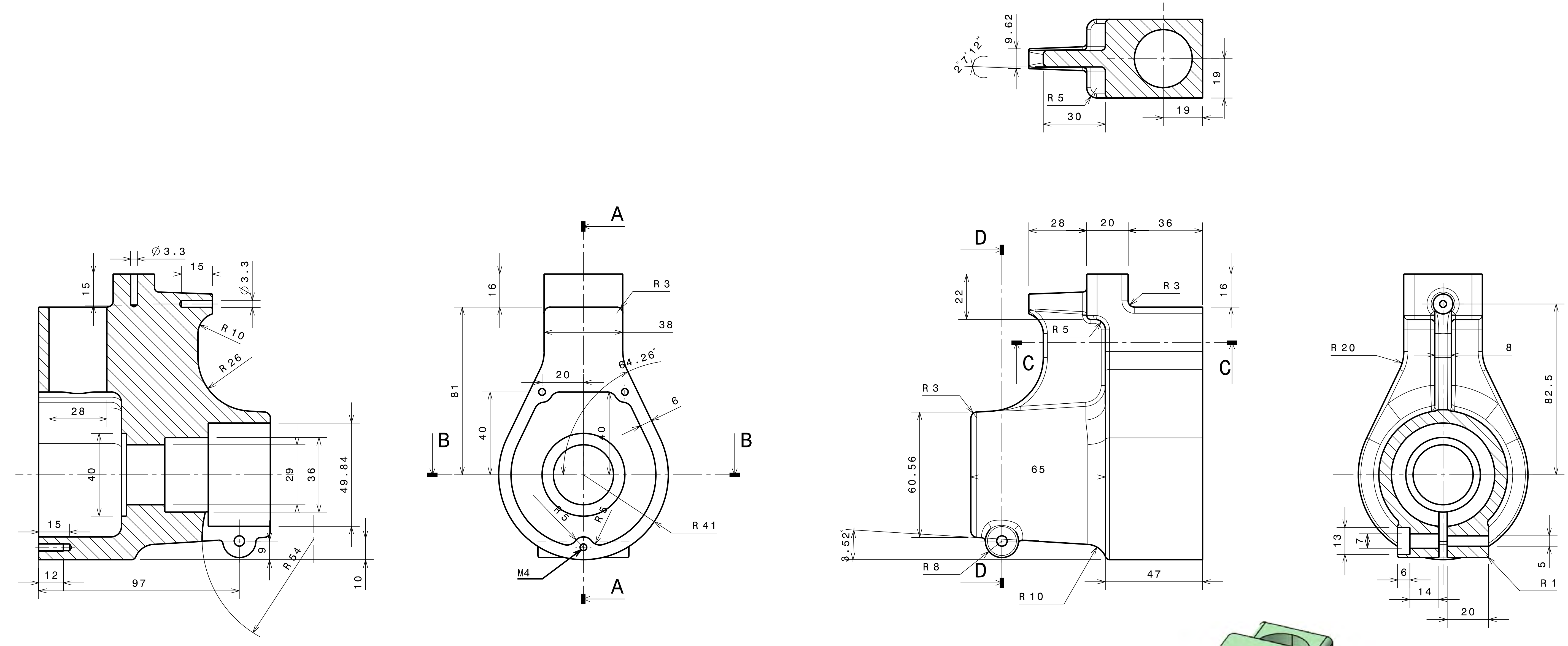
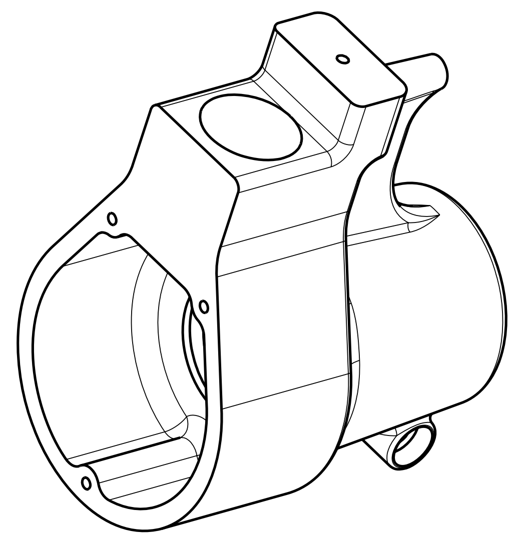
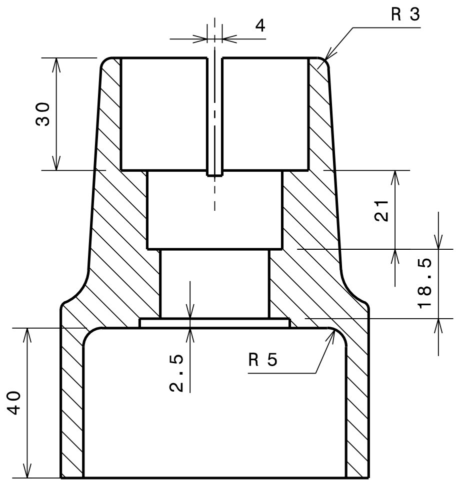
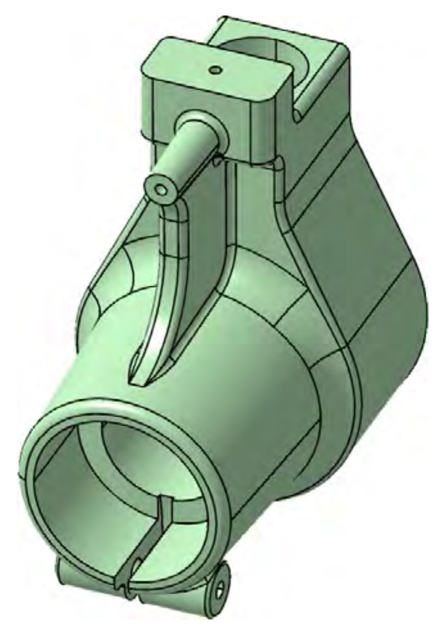
French ministry of education and research







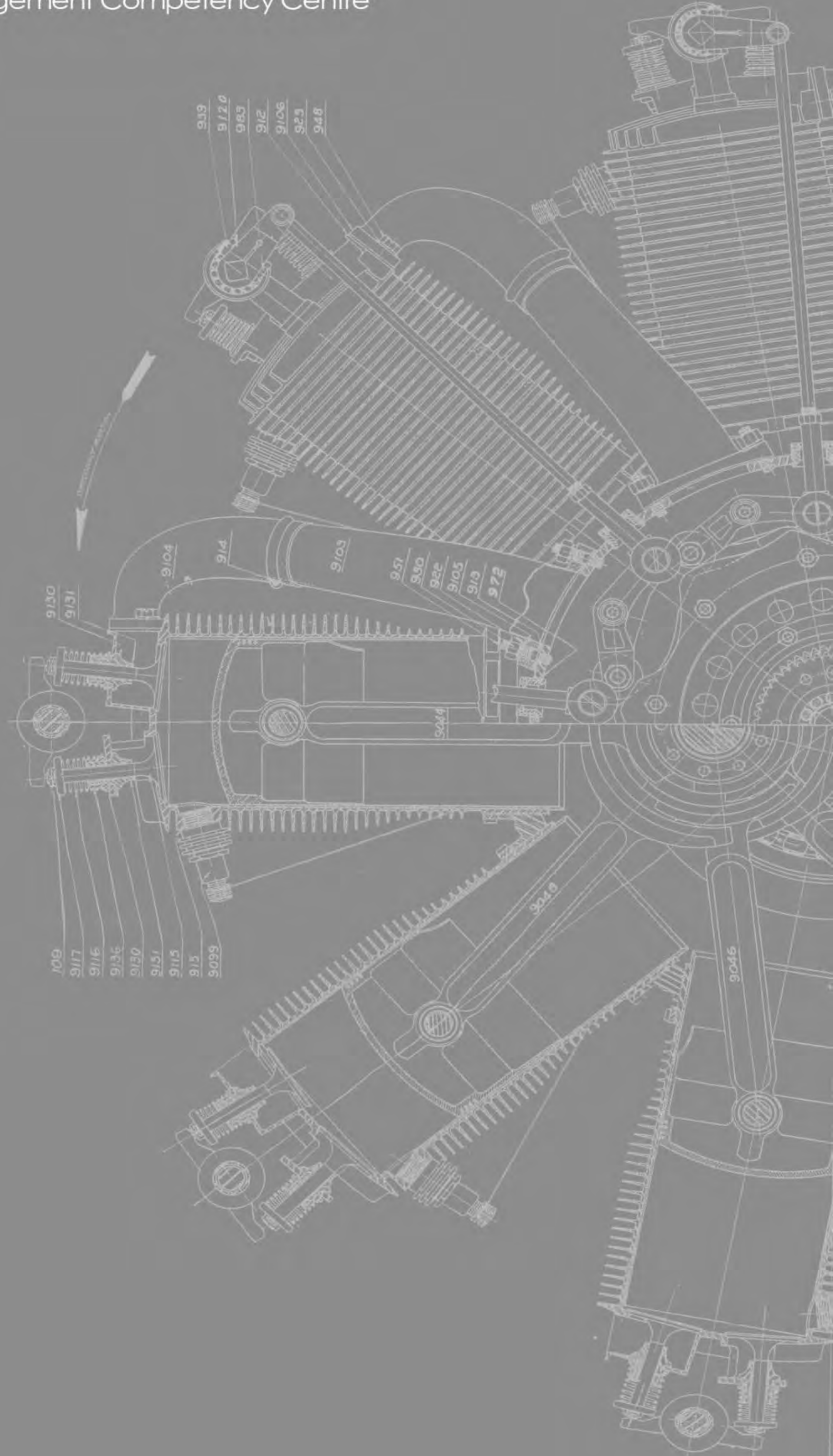


Default Radius = 2 mm

1:2	SAW CASING
	PLMCC
A3	Drawing by: S. Bouye
	Date : 20/04/07

PLMCC

Product Lifecycle Management Competency Centre



Surface Design

DASSAULT SYSTEMES

Cape Peninsula University of Technology

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RÉPUBLIQUE FRANÇAISE

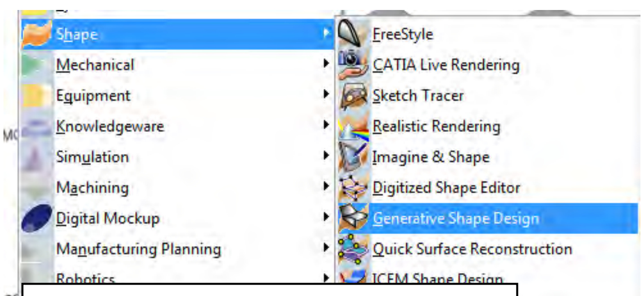
French ministry of education and research

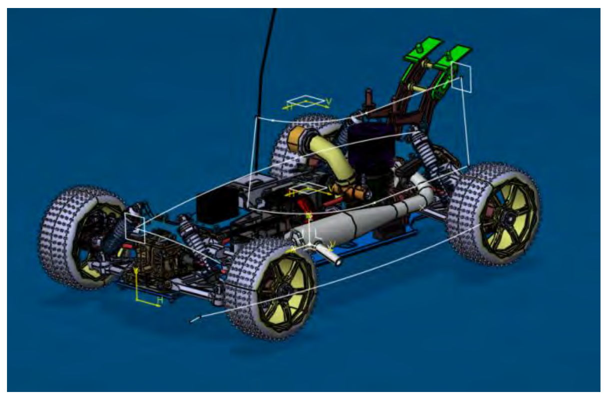
Basic Surface Design

The purpose of this exercise is to create surfaces for the buggy from a wireframe that has already been set up. You will get to know the basics of surface creation and convert the final surface design into a solid part

Import the micro motor 3DXML

Click PLM Access > Import > 3D XML….
You will find it in the Buggy_3Dxml> Starting_Data>surface folder.
Select the file named: -your-Name-buggy.3dxml
Check Import As New.
As duplication string enter „*your name_'*
Click OK.

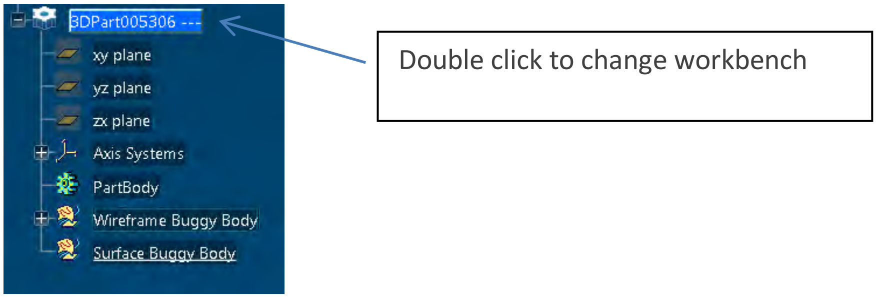




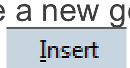
Use the Generative Shape Design (under shape) Workbench to create the surface.

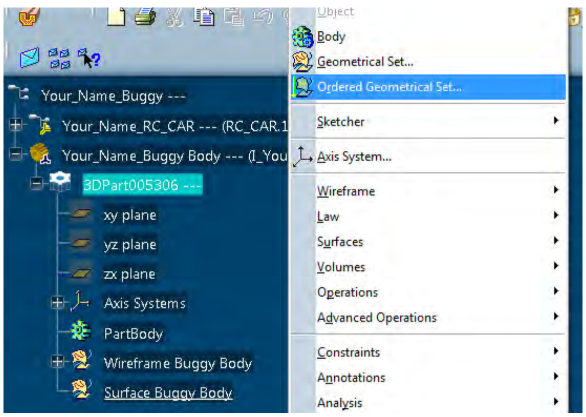
The Image on your screen should relate to the image above. Notice that the Wireframe geometrical set has already been created for this as well as the Surface geometrical set.

It is advantageous to create two separate geometrical sets for the creation of a surface. The wireframe serves as a referencing element to all the surfaces that will be created. This is done to easily hide all the wireframes and sketches created in this specific geometrical set.



Notice that the geometrical set “Surface Buggy Body” is active, so any creation, be it surfaces or sketches, will take place under the Surface geometrical set.

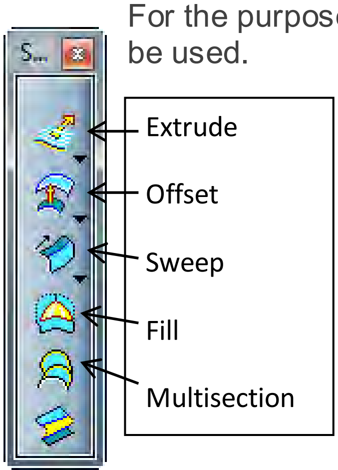
To create a new geometrical set, ensure that the 3D Part is active in the feature tree. By selecting  on the top tool bar, one can insert a new Body, Geometrical set or Ordered geometrical set.



Surface creation

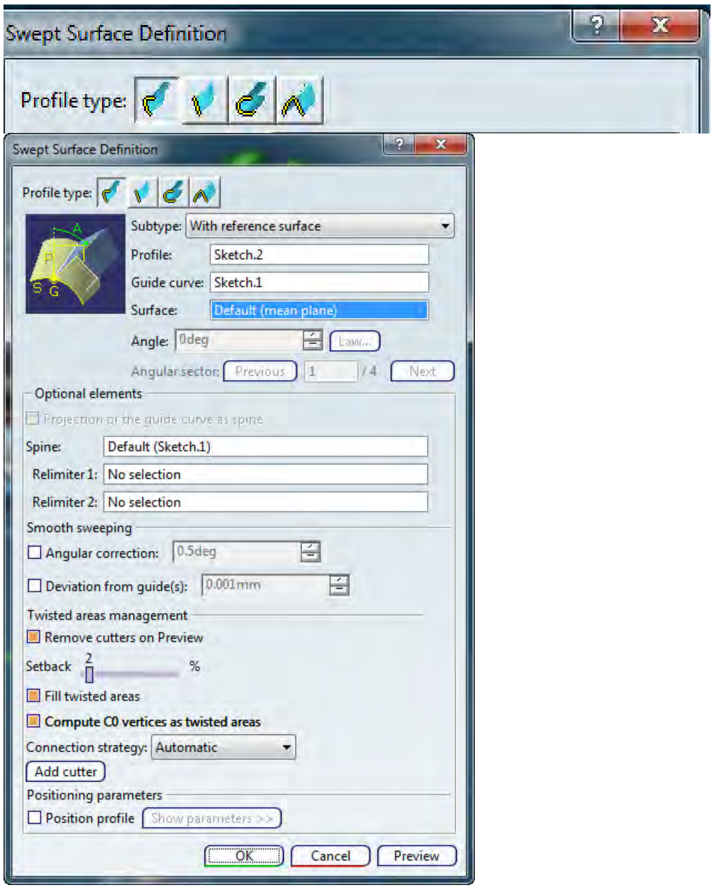
By utilizing the wireframe that has already been created, the surfaces can be created using various tools in the surface creation toolbar.

For the purposes of this exercise, the following tools in the surfaces toolbar will be used.

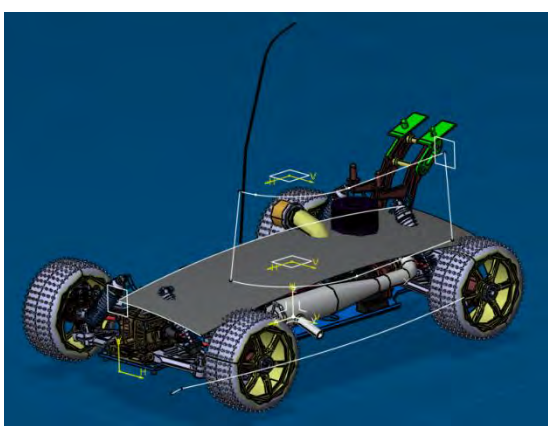


By using the sweep  tool, create a surface using Sketch.2 as your profile and Sketch.1 as the Guide curve. See dialogue box on the next page for instructions.

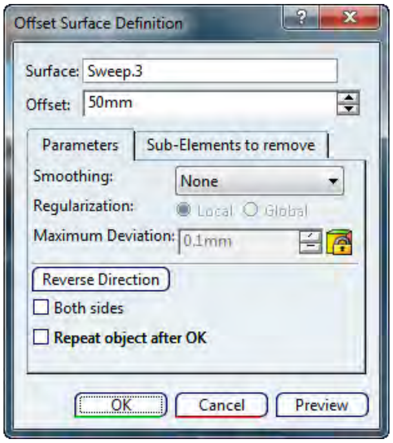
The sweep function has various profile types that can be utilized for different scenarios. For this one you will use the implicit sweep.

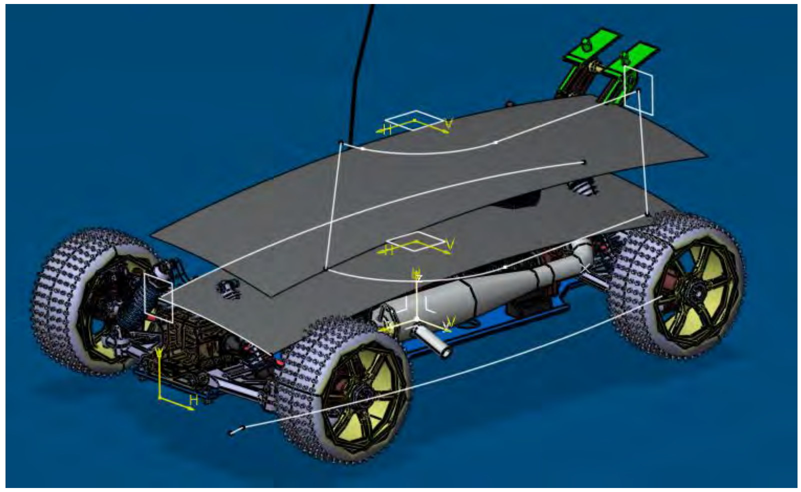


Your surface should be similar to the one that is shown below.



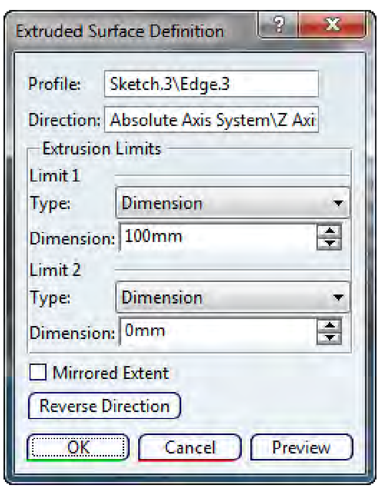
Using the Offset tool  select the swept surface to create a new surface offset 50mm above the original surface.

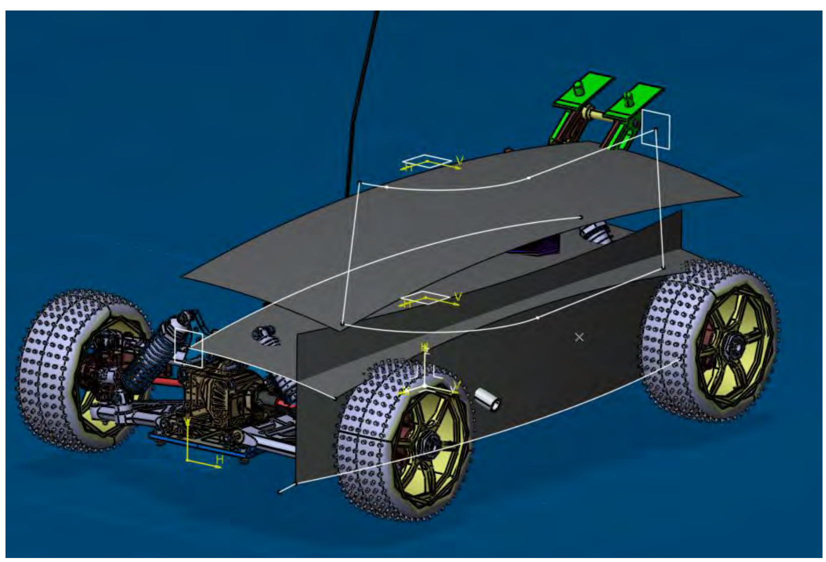




Check that the result is similar to what is shown above.

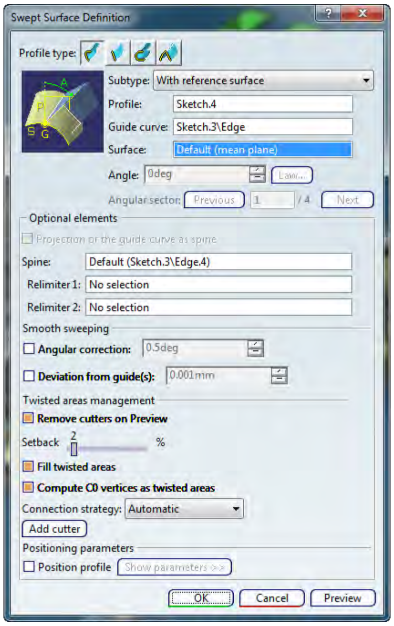
Using the extrude tool  you can create a new surface that runs vertically from Sketch.3

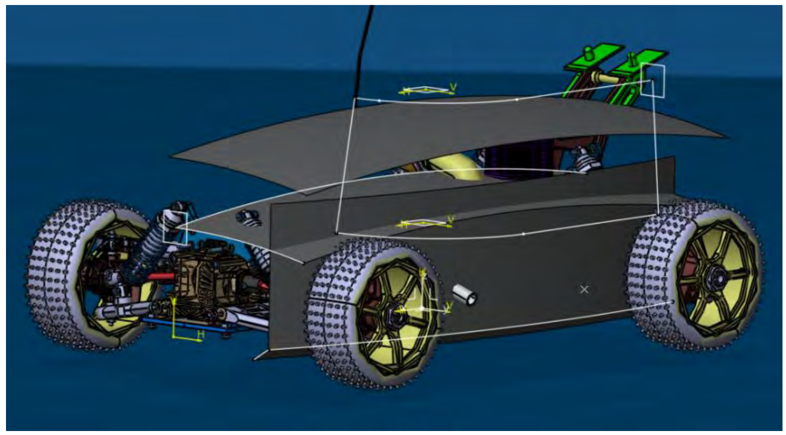




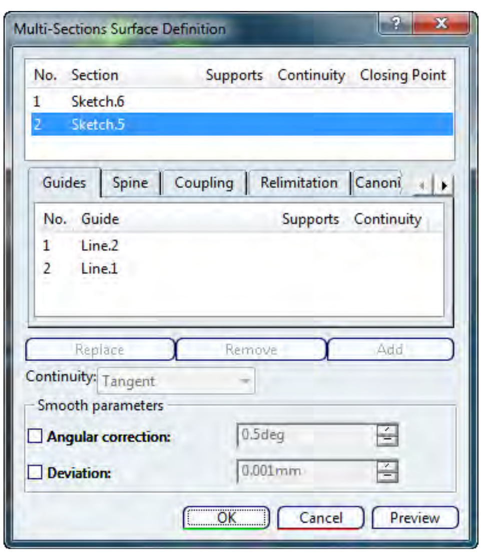
The Extrude tool is very helpful to create simple surfaces by only using only one element. Direction can be added by clicking on the desired arrow in the reference axis, or by using a previously created item to indicate direction.

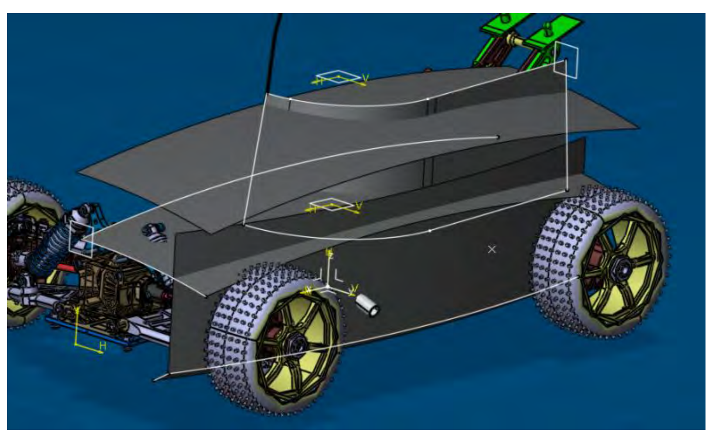
Use the sweep tool again to create another swept surface at the bottom of the extruded surface. Use Sketch.4 as the Profile and Sketch.3 as the Guide curve.





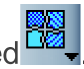
For the next surface you will use the Multisection  tool. It is necessary to have both guides and sections for this tool. Multiple sections and guides can be used to create a more accurate surface.
Fot the creation of the cabin utilize the sections and guides as described below in the dialogue box.





Now that the basic surfaces have been created, you can continue to the Trim and join functions.

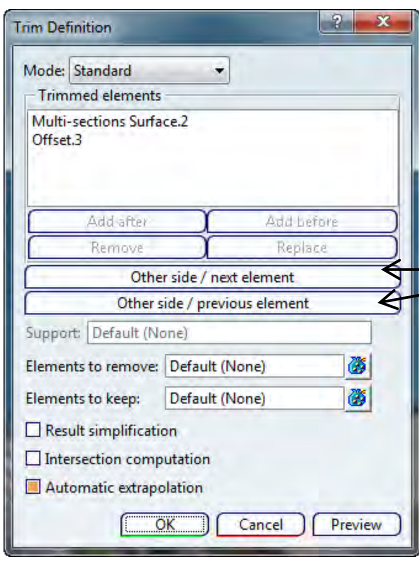
Trimming, splitting and joining

All the created surfaces are still separate entities and to create a coherent single surface all the surfaces must be trimmed or split to the desired specification and/or joined. When trimming two surfaces a new surface is automatically created from the result. Thus the trim tool also joins. Those surfaces that are not trimmed should be joined to make one surface.

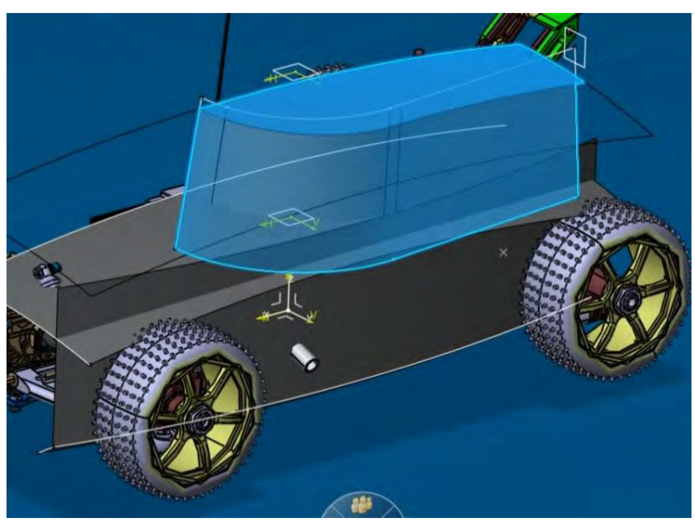
When trimming, your operations need to planned carefully as it is important to predict which surfaces need to be trimmed firstly.

The first surfaces to trim are that of the cabin roof and side.

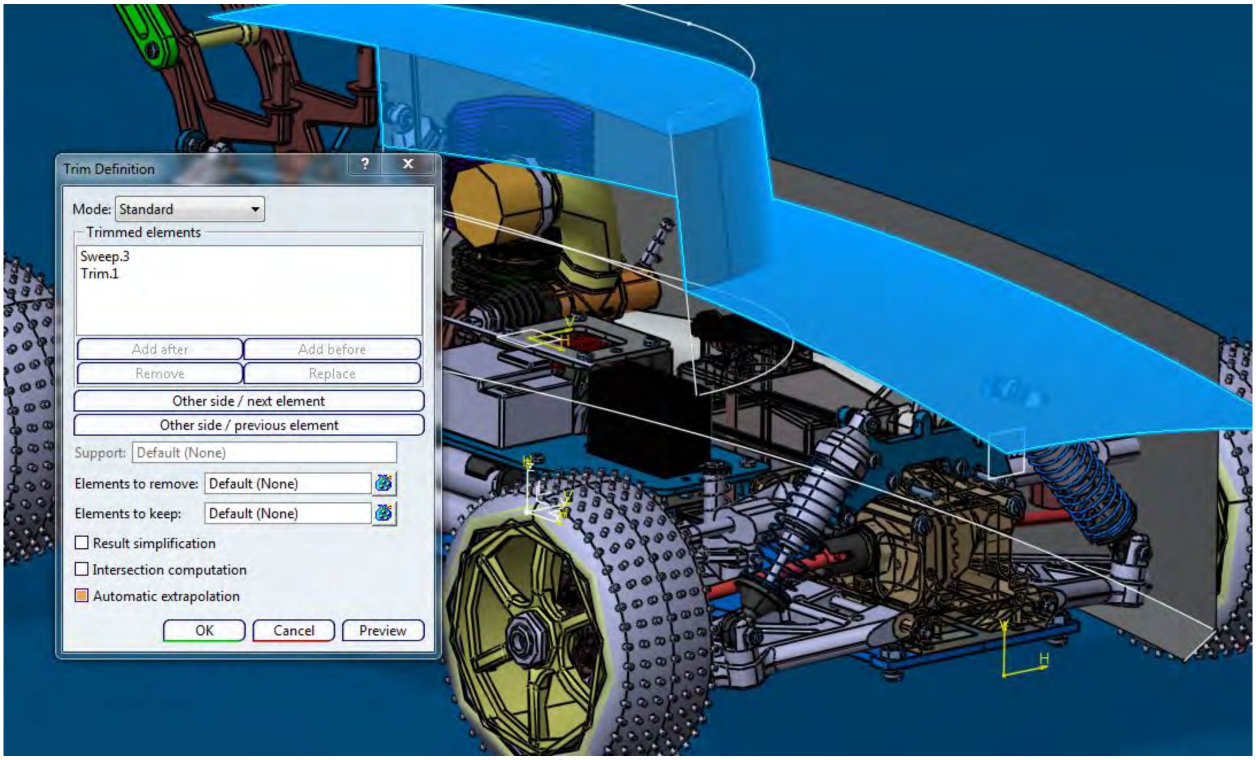
Select the trim function. (Click on the small black arrow that indicates that there are more tools hidden if the split tool is shown as the default tool to select the trim function)



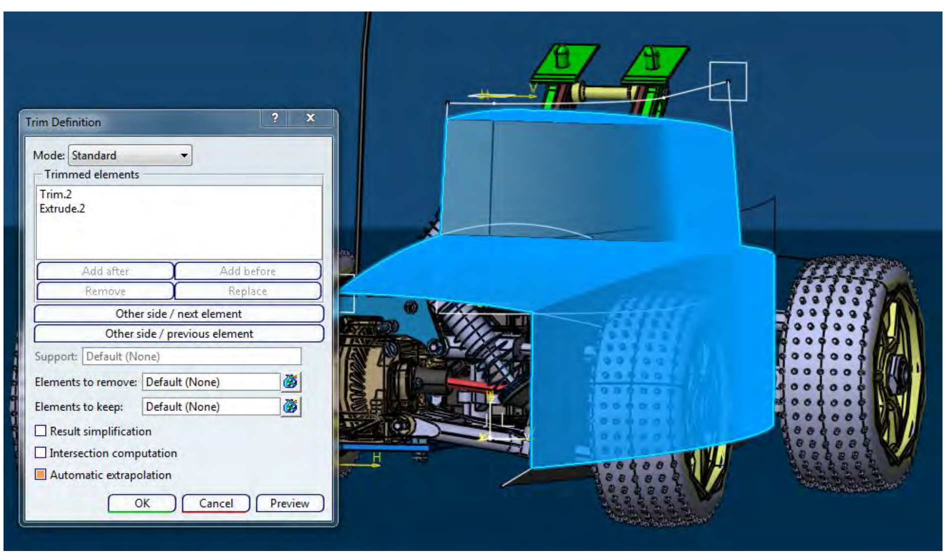
Click on the “other side/ next element” and “other side/ previous element” boxes to switch between different trim options. See the final result below.



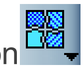
In a similar fashion, trim the cabin and the first sweep you created. Notice that the cabin trim is now one element. As mentioned earlier, the trim function automatically joins the result.

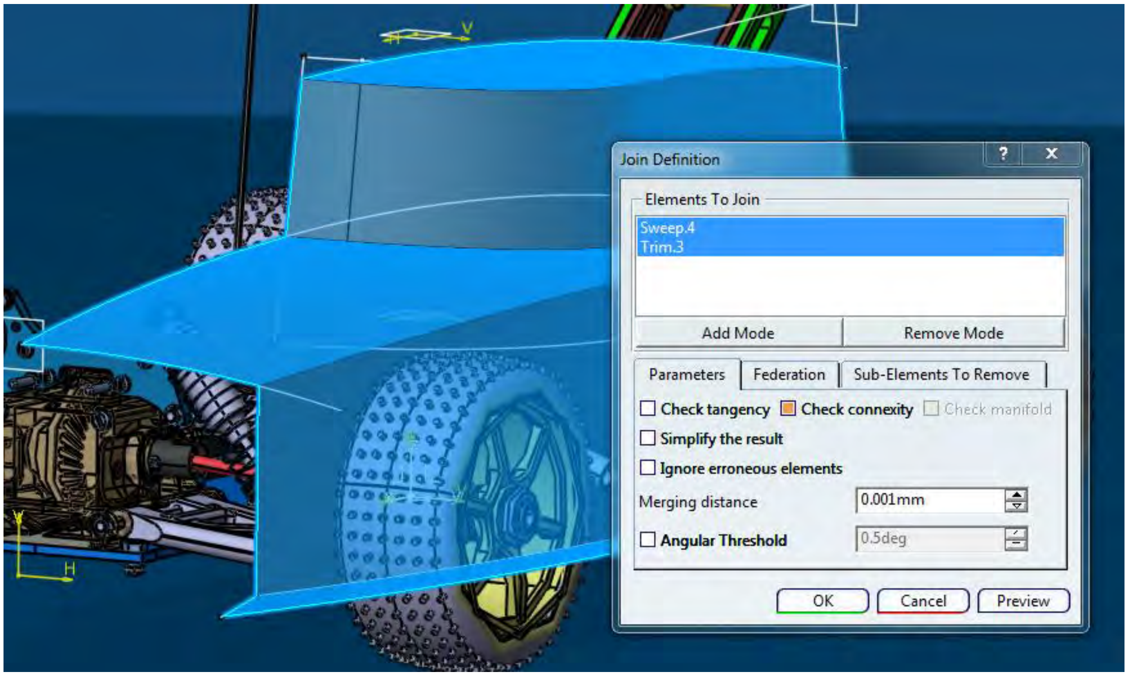


For the final trim, use the newly created trim and the vertical extrude to create the following result shown below.



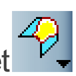
Now that all the surfaces have been trimmed to form one surface, the joining operation can be used to join the still separate sweep at the bottom to the trim surface.

Select the join function  and join the trim and bottom Sweep.

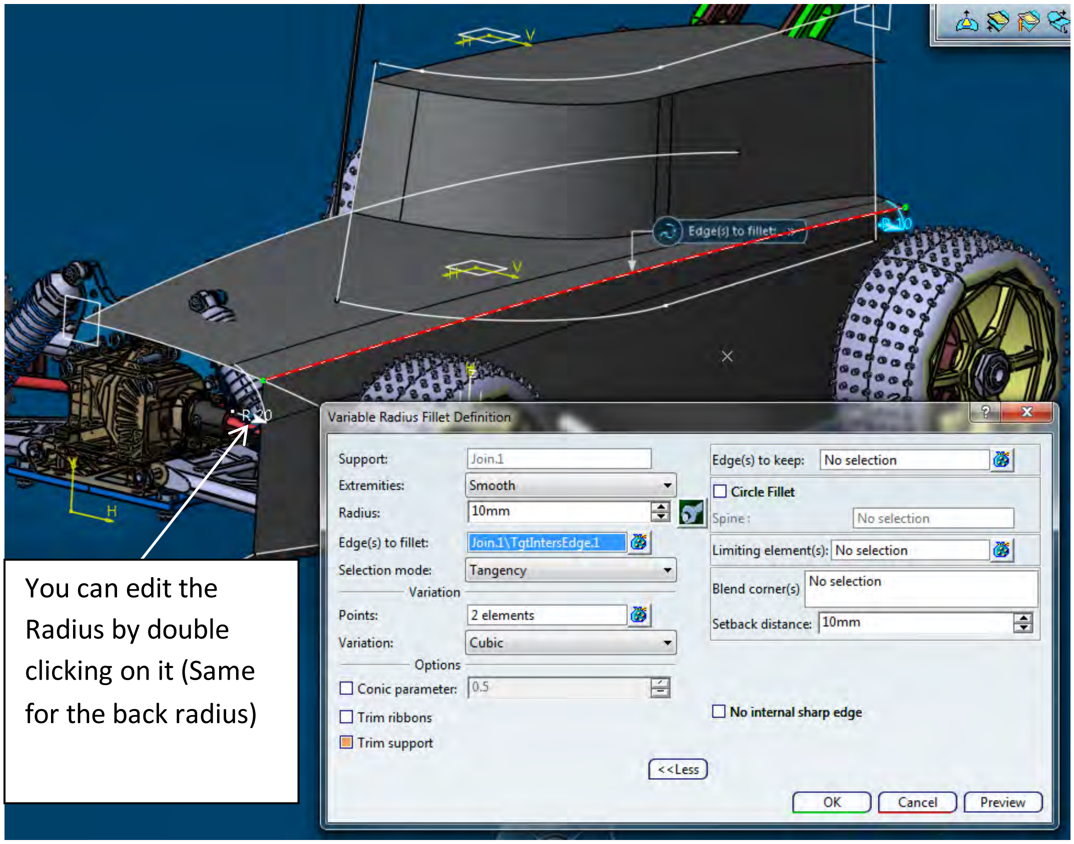


Using Fillets

By using fillets you can create “rounds” between various joined surfaces. There are a few different fillet functions, but for this exercise we will only use two different ones.

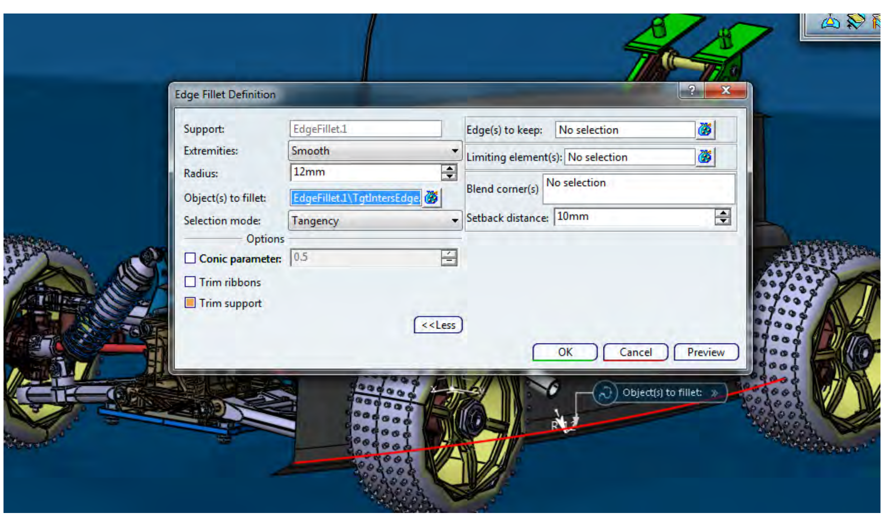
The first Fillet can be created with the variable fillet  tool. (Look for the tool by clicking on the black arrow if the Fillet tool  is displayed as the default.)

After the variable fillet tool has been selected, click on the edge as shown on the next page. The value of the radius can be altered from the start and the end of the edge. Input the front value (by double clicking on the displayed radius) to R20 and the back as R10.

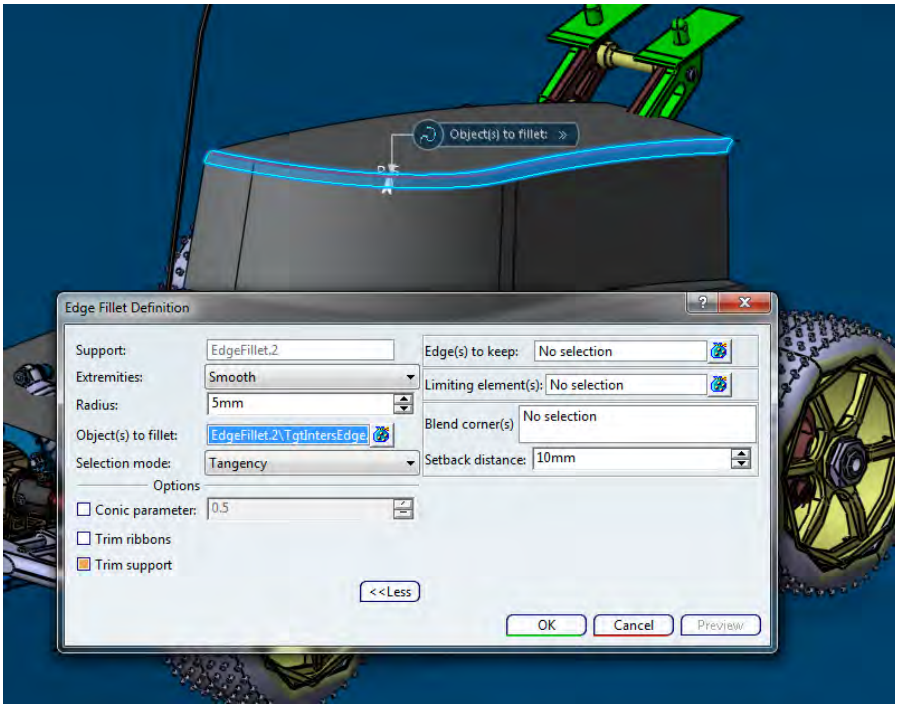


By using the edge fillet tool  create an edge fillet on the bottom edge as shown below.

(*Tip: It might be advantageous to hide the wireframe geometric set for this operation as there might be some confusion between the edge you want to select and the sketch that was originally created to create the surfaces.)*



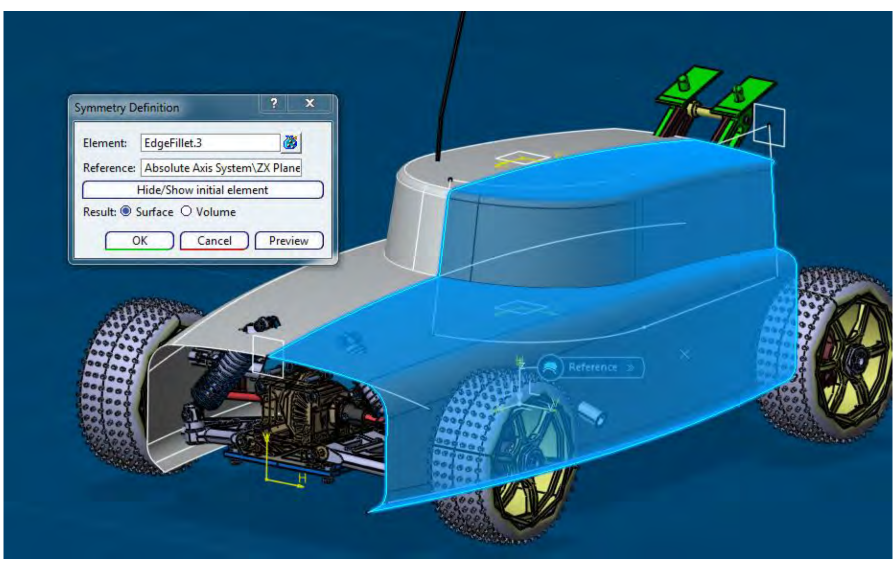
Using the edge fillet again, create a fillet with a radius of 5mm on the top of the canopy.



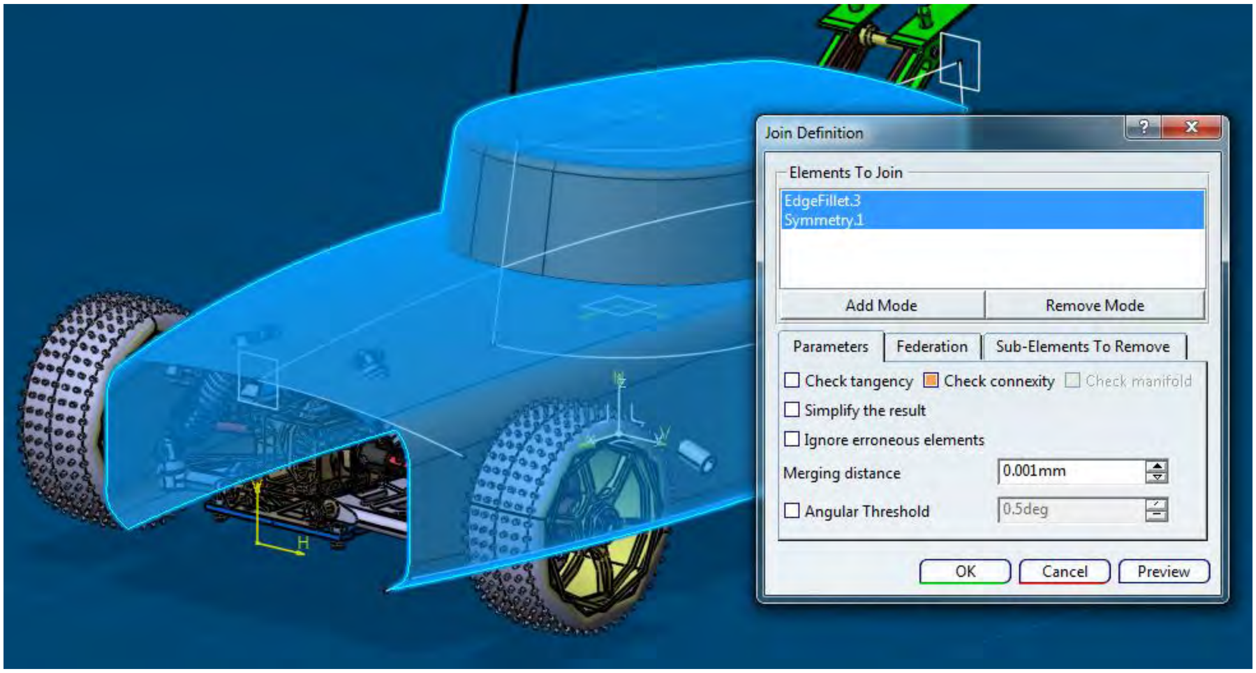
Symmetry and Fill

Now that the main body has been made, you can mirror it by using the symmetry  tool.

Click on the symmetry tool and then select the reference element (ZX plane)

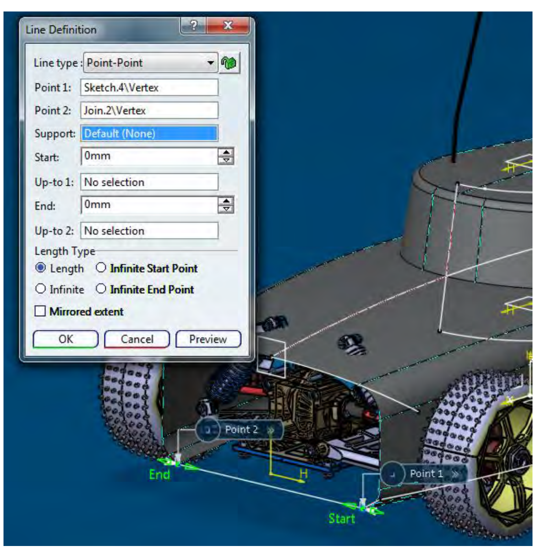


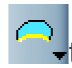
When the symmetry is completed you will notice that the two pieces are still separate. Use the join tool to join the two surfaces.

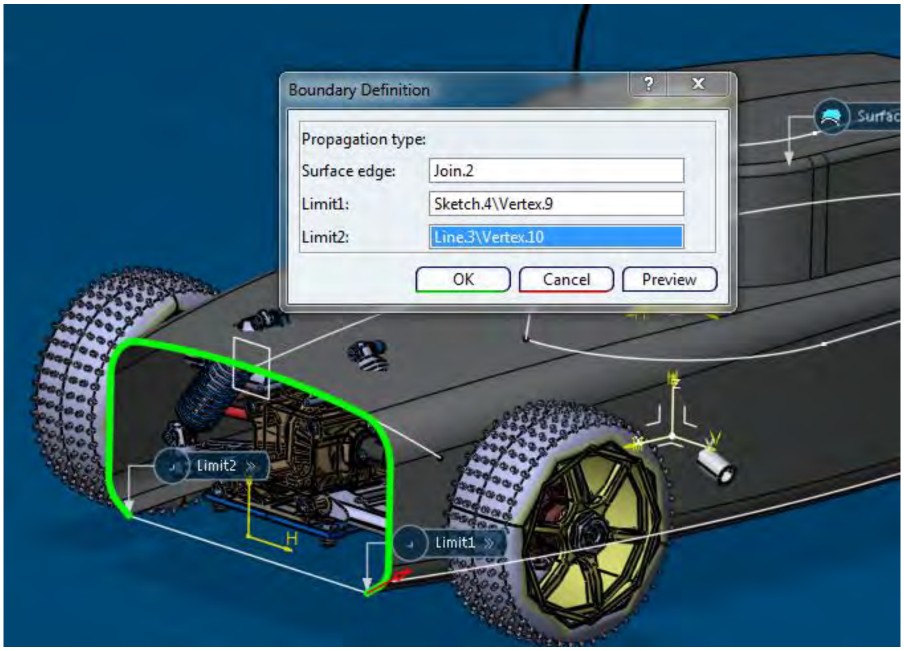


The Nose needs to be closed, for that you can use the Fill  function.

Before filling the nose part, you will first need to close it. This can be done by creating a line in 3D space. First you need to select the Wireframe geometric set as your "Define in work space". Select the line  tool and connect it by clicking on the bottom front end of the body as shown below.



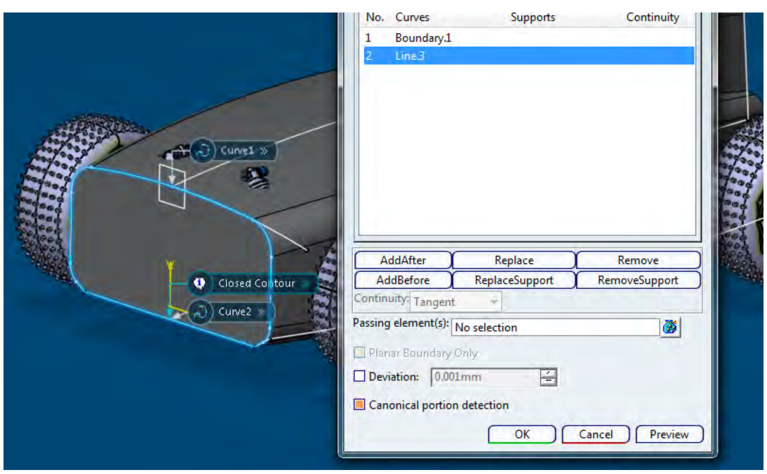
In order for the fill tool to work, you will need also create a boundary around the edge of the nose. For this you can use the boundary  tool.

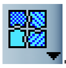


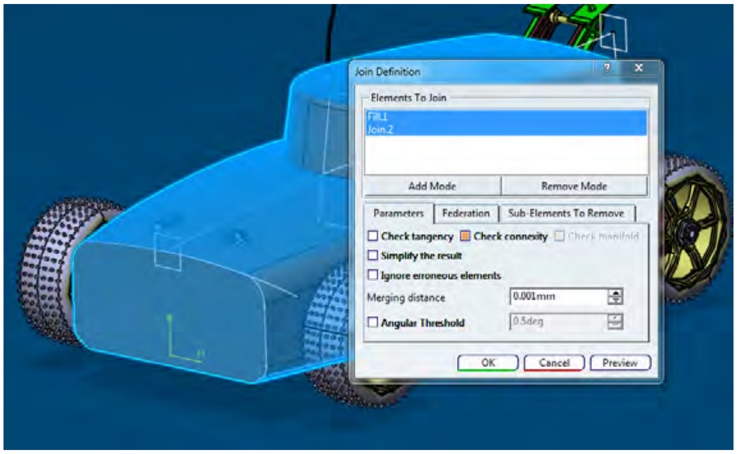
The boundary tool will highlight all the open edges of the surface in green. To define it, you can click on limits or points on the surface to limit the boundary extent. Select the bottom two corners, as shown above, as limits. If the opposite selection is highlighted, simply click on the **red arrow** to toggle between selections.

After the line and boundary creation, return to the surface geometrical set and make sure it is defined as the in work object.

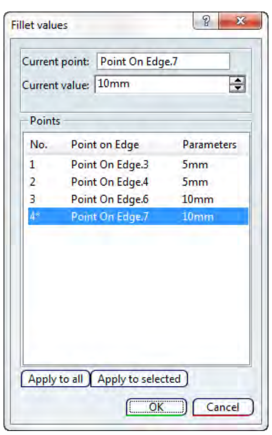
Now you can use the Fill function to fill the nose. Select fill and then click on the boundary and the line. The result should be as shown below.

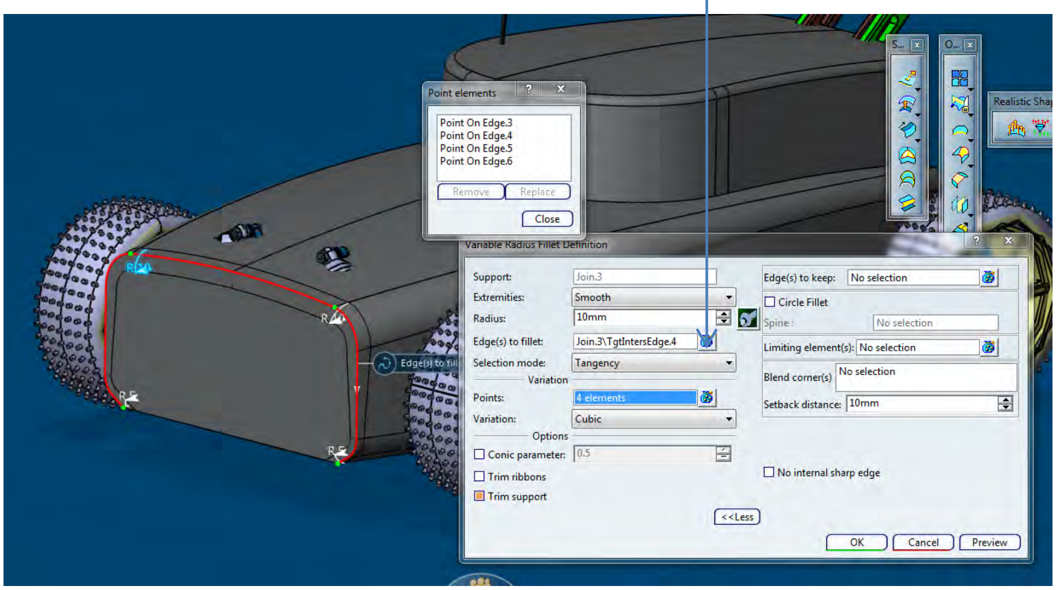


You need to join the newly created nose surface to the main surface. Use the Join tool .



Finally, create a fillet on the nose to give it a smoother appearance. Use the variable  fillet tool. Add more points by clicking on the add/remove  icon and click on the points as shown below. Make the top two radiuses R10 and the bottom R5.

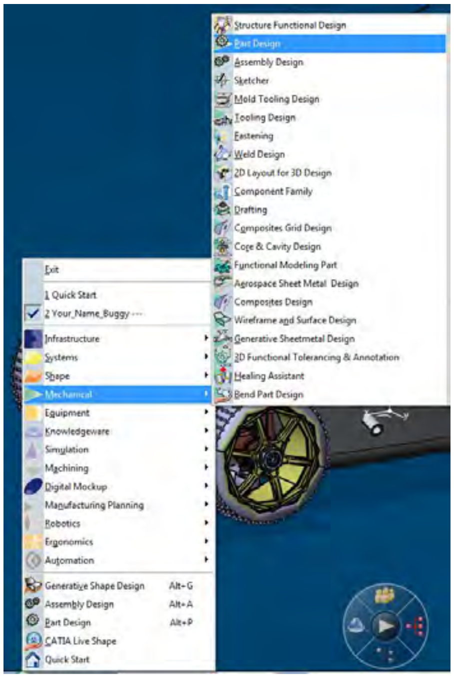




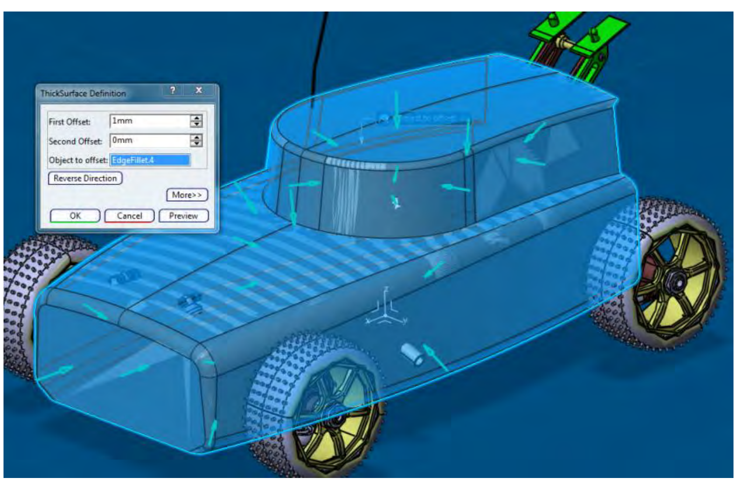
Thicken Surface

The next step will be to thicken the surface so that it actually becomes a part.

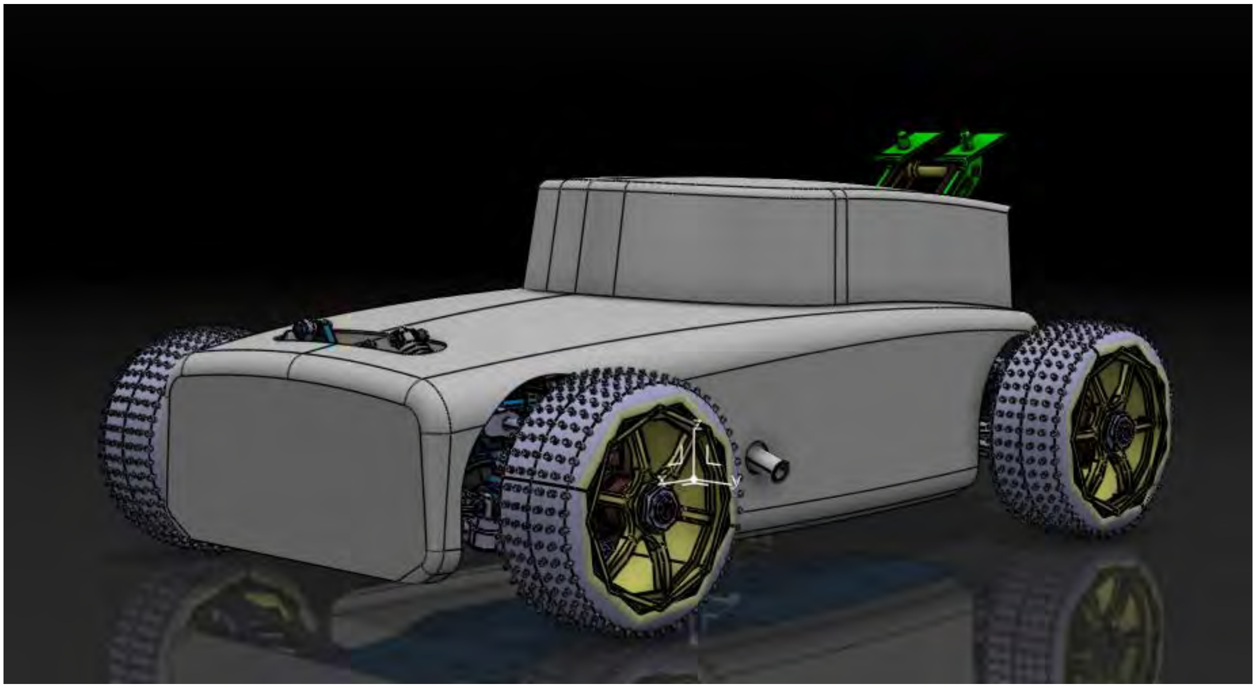
First you will need to switch the workshop from GSD to Part Design.



Once in the Part environment you need to convert the surfaces into a part. This can be done by using the Thick surface  tool. By using the Thick surface tool, an offset thickness is applied to the surface in the desired direction. Select the Thick surface function and click on the surface. Select a 1mm thickness to be added to the inside of the surface as shown below.



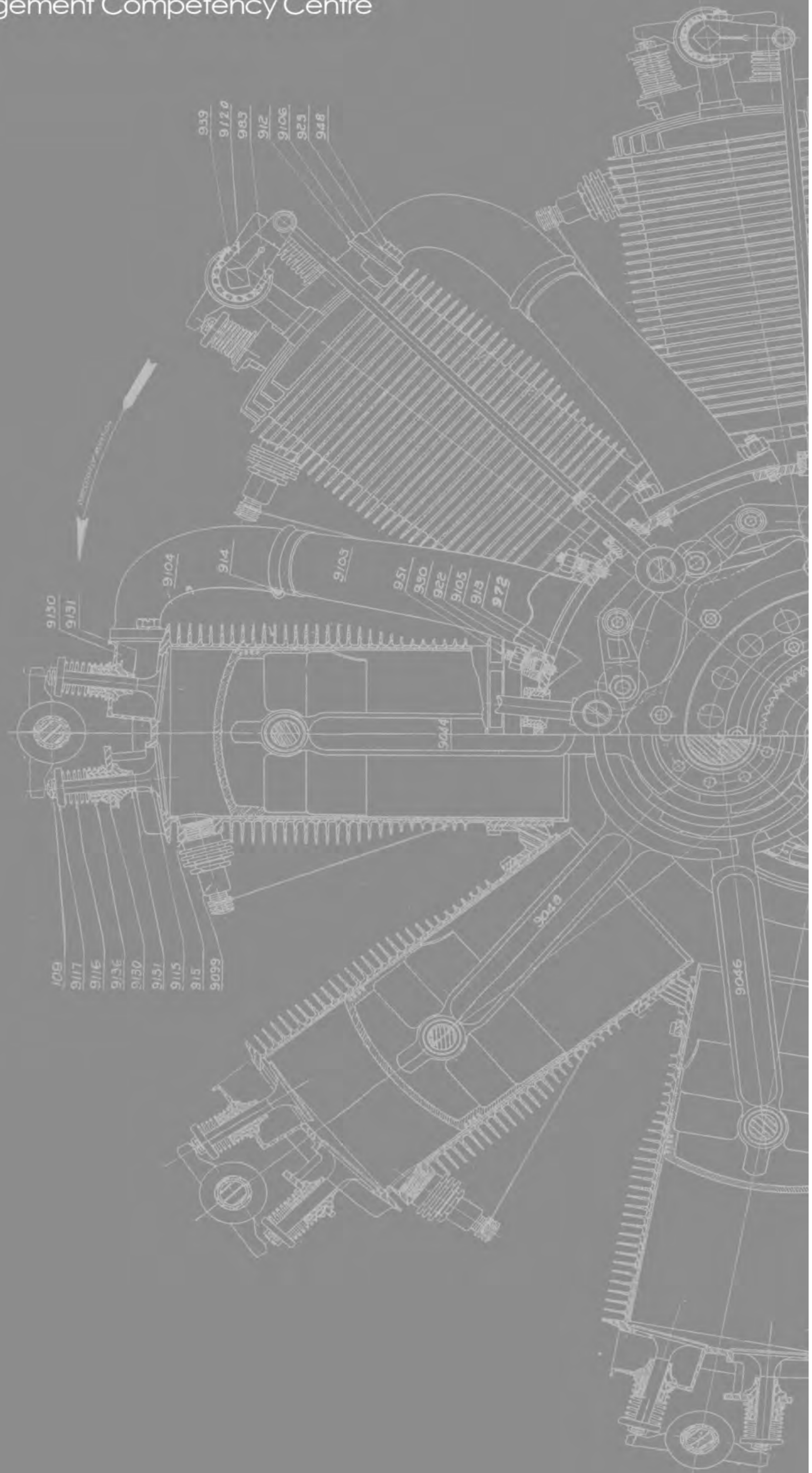
The surface is now a solid part. Using your knowledge of Part design, create cut outs on those parts where the body clashes with the mechanics and other functional parts of the buggy.



END OF MODULE

PLMCC

Product Lifecycle Management Competency Centre



Surface Design

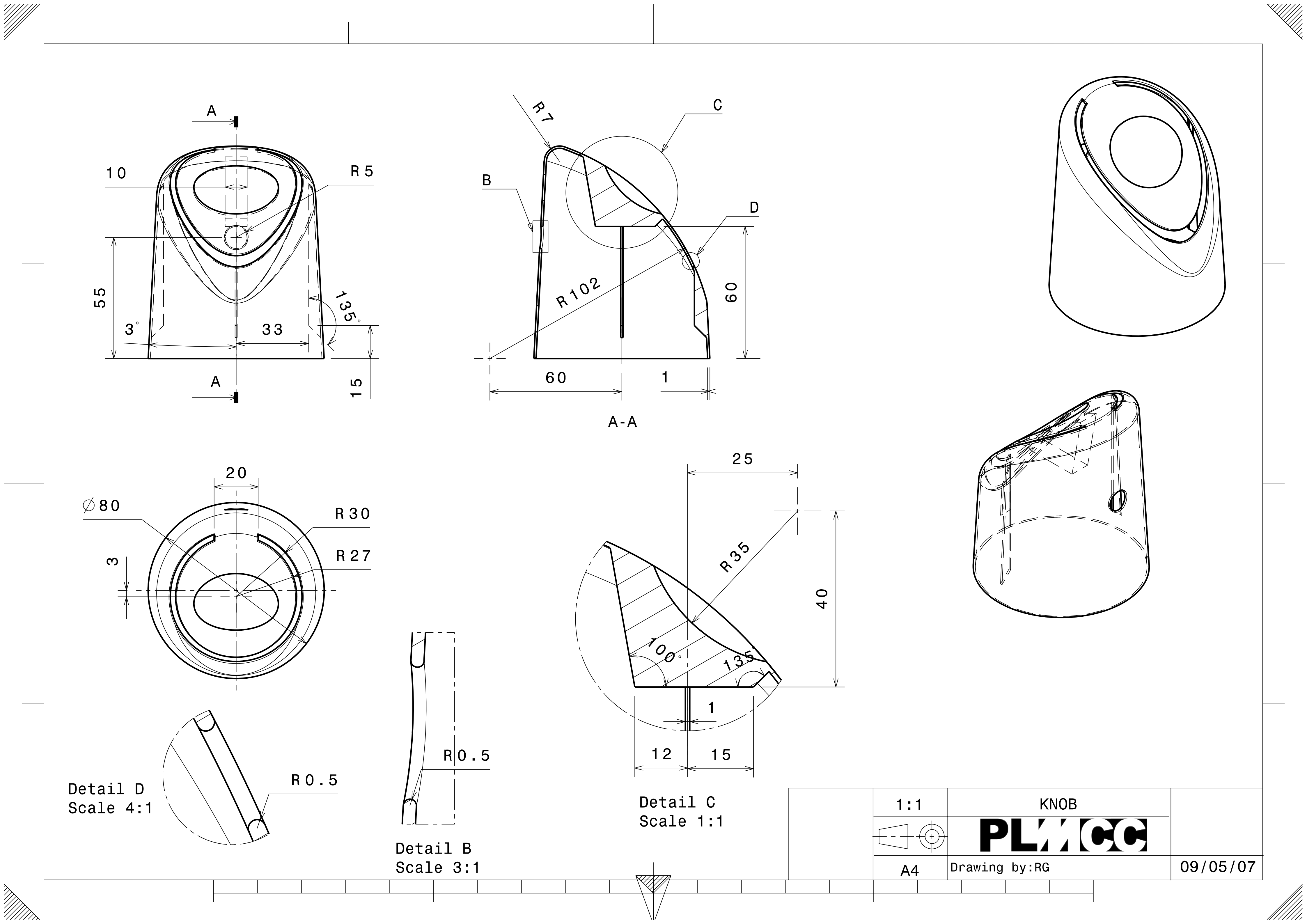
Exercises

Dassault Systemes

Cape Peninsula University of Technology

Liberté • Égalité • Fraternité
République Française

French ministry of education and research


A
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10
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B
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R102
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60
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135
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12
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R0.5
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Detail D
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Detail B
Scale 3:1
Detail C
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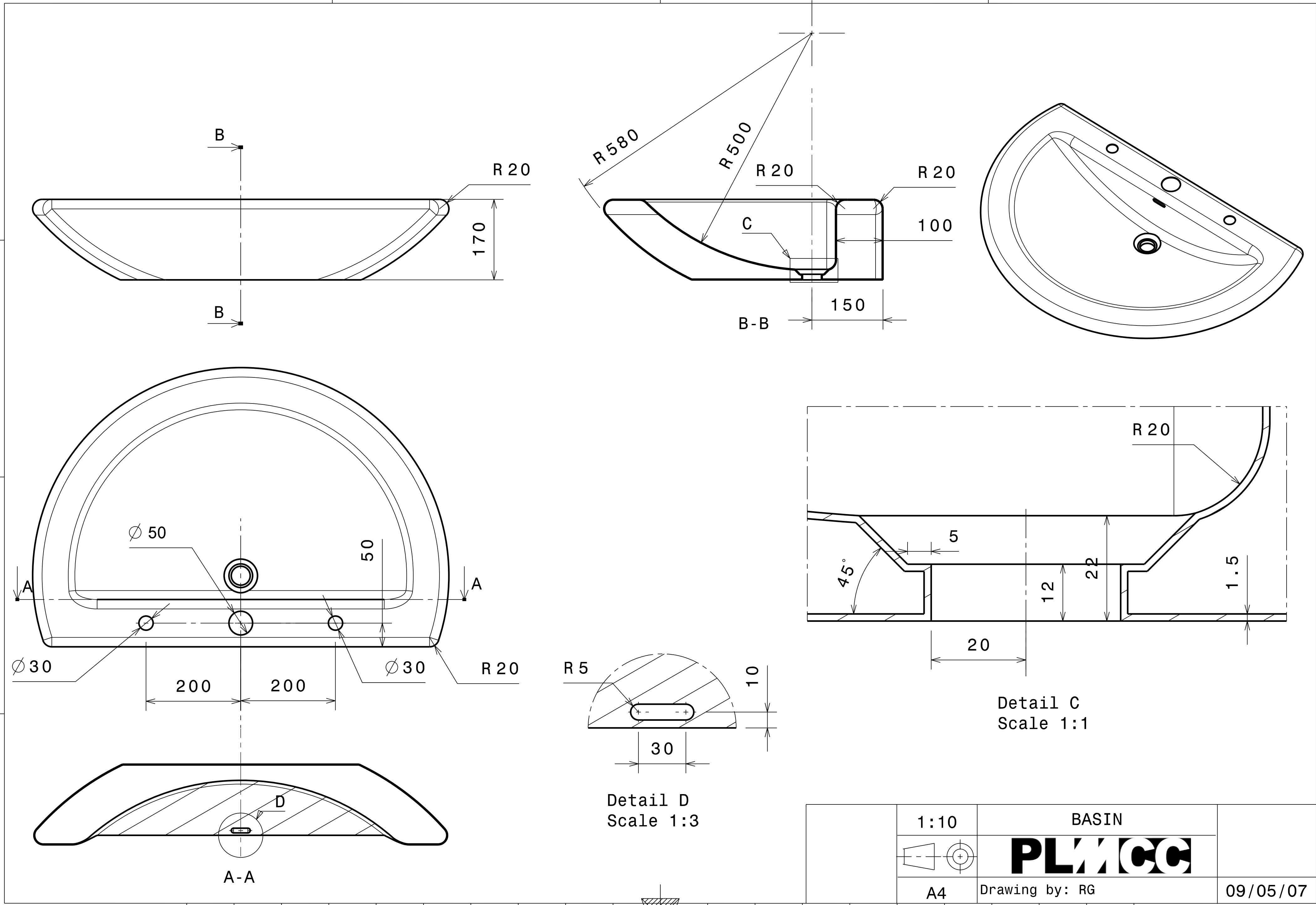
A4
Drawing by:RG
09/05/07


B
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Drawing by: RG
09/05/07