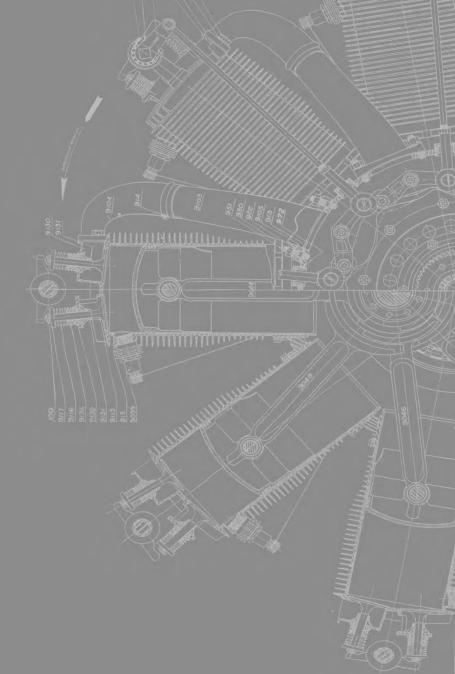


Catia V6 Introduction

Buggy Case Study









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....)

5 Connect	8 23
	З САТІА
	Data Source TESTSERVER (10.36.0.25 Version More) User Name Password Version Ok Cancel Options
1	ity Context Selection ? X ity Context: VPLMCreator.MyCompany.Dis_VS VPLMCreator.MyCompany.Dis_V4 VPLMProjectLeader.MyCompany.CAT_LIVE_SHAPE VPLMExperimenter.MyCompany.Collab_design

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:

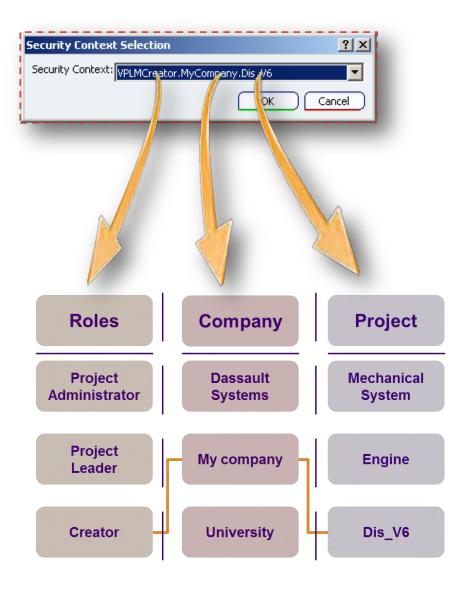
- Project Admin
- Project Leader
- Creator
- Experimenter
- Viewer

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.



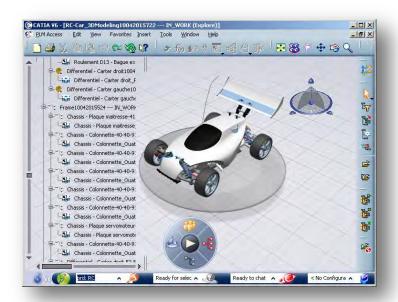


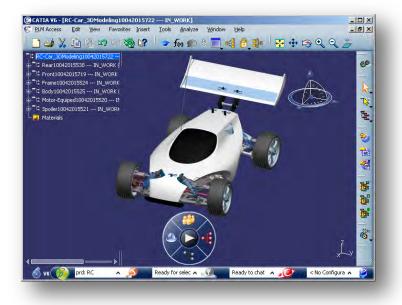


Presentation of V6 GUI In this step you will be introduced to the V6 PLM 2.0 interface. V6 includes two user interface windows, called Navigation and Authoring.

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.





PLMCC



- 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).





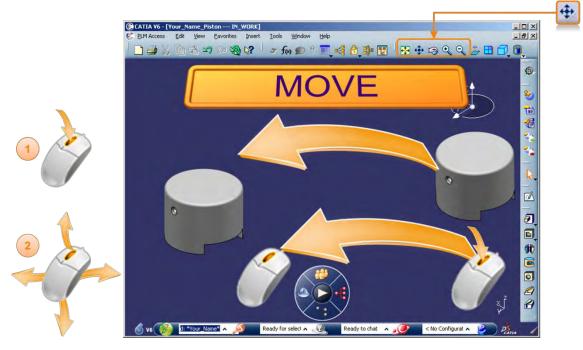
- 4. The Bar serves as a quick access tool for:
- a. Searching for data
- b. Examining the impacts of modifications
- c. Collaborating with people
- d. Saving the modifications to the database



Mouse Manipulation

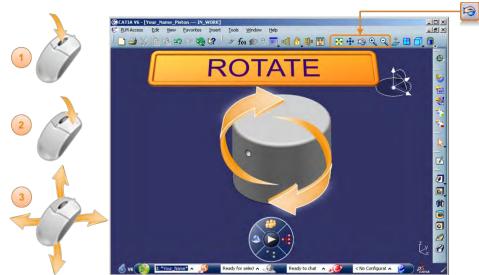
- To pan the view:
- 1. Hold-click on the middle button.
- 2. Then, while holding the middle button pushed, move your mouse to pan the view.

You can also use the Pan icon.

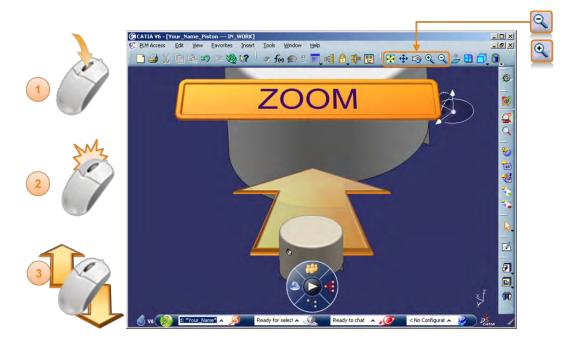




- To rotate the view:
- 1. Click and hold the middle button.
- 2. Then, while middle button is pushed, click and hold the right button.
- 3. Then, while holding middle and pushing the right button, move your mouse to rotate the view



- 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.

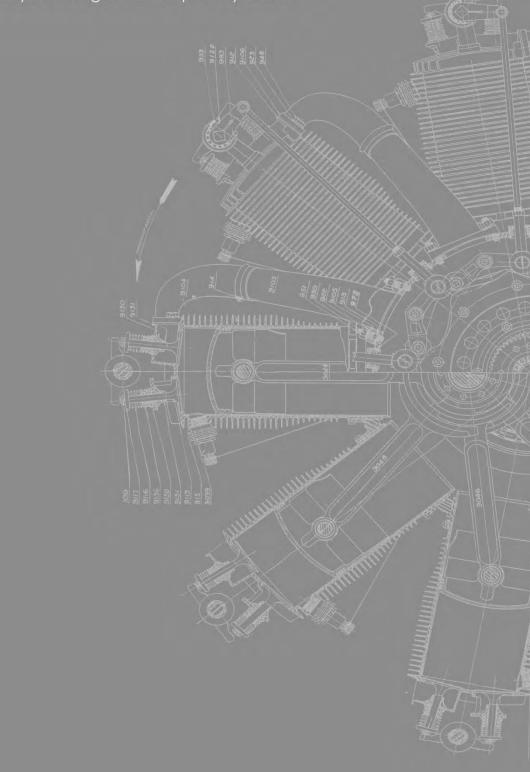




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	Isometric			Shading
	Front			Shading with Edge
	Back			Without Smooth Edge
	Left			With Hidden Edge
	Right	Ø		Shading with Material
	Тор			Wireframe
	Bottom		?	Customize









Part Design



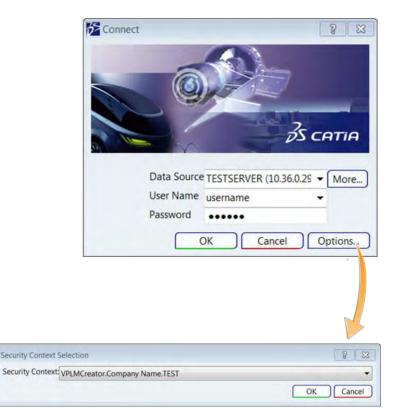


Part Design

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.)

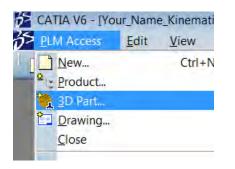
Design Process of the Piston





1. Create a new part

a. Select "PLM Access" > "3D Part…"



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

3D Part / Physical Product			
	🔒 3D Part	Physical Representation	
	Title	Your_Name_Piston	8
	Name	3dp004420	*

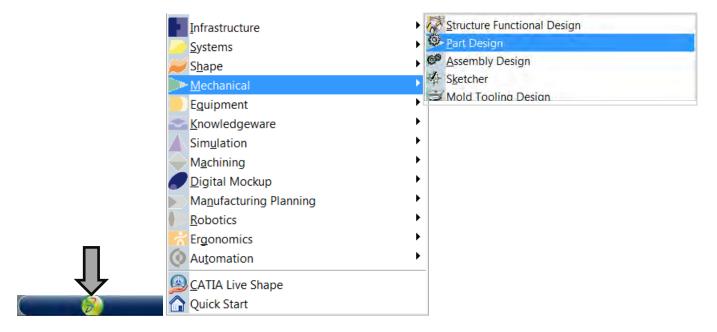
d. Type "Your_Name_Piston" in the field "Physical Representation Title",

 3D Part / Ph	vsical Product		• ×
👌 3D Part	Physical Representation	Characteristics	
Title	Your_Name_Piston		8
Name	3sh004572		*
			」 ◇ ☆

Click "Finish"



2. To access 'Part Design' click 'Start' in the bar > 'Mechanical' > 'Part Design.



The Part design workbench is opened as shown below.

	vorites Insert Iools <u>W</u> indow <u>H</u> elp	
	🖢 fo 🗩 🤊 📰 📢 👌 🤑 👿 🖢 💠 🙃	९९⊞C ,,22 ∞ \$. 58 0
		Part Design
Your_Name_Piston		workbench icon
Vour_Name_Piston		
Axis Systems PartBody		
		0
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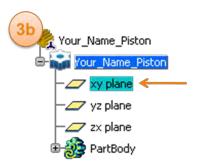
\delta vs 🧭 Ready to search 🔹	Ready for selection A Ready to chat	A 🧟 Ready for propage A 🖉 🕉 CATIA

PLMCC

3. Sketch the primary shape of the piston

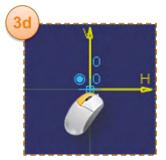
- a. Click on the "Positioned Sketch" icon
- b. Select the "xy plane"



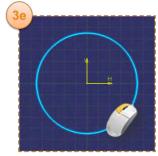


- c. Click on the "Circle" icon
- d. Select the origin to position the center of the circle





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

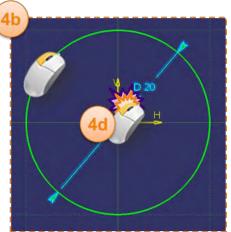




4. Define the diameter of the circle

- Click on the "Constraint" icon. a.
- Select the circle. b.

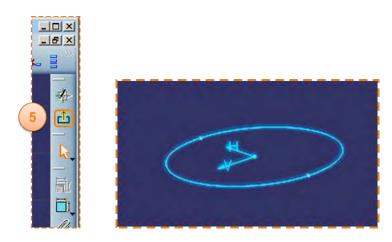




- Click anywhere on the graphic area to validate the dimension. Double click the dimension to modify C.
- d.
- Enter [14mm] e.
- Click "OK" f.

Constraint Definition	?×
	D 54
Dimension Diameter	
(41)	re>>>

5. Click on the 'Exit Workbench' to exit the sketch

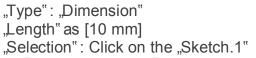




Now extrude previously created sketch and generate a cylinder 6.

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



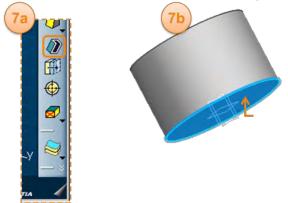




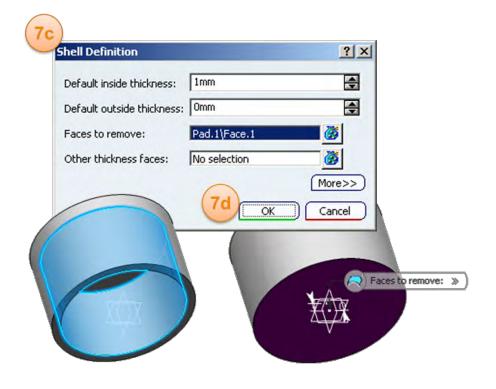
7. Now perform a shell to remove matter inside the piston.

a. Click on the "Shell" icon





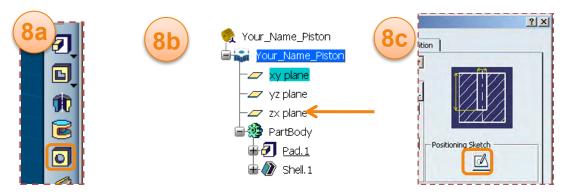
- c. Define the panel as shown below. Enter [1 mm] as "Default inside Thickness" "Face to remove" : "Pad.1\Face.1"
- d. Click OK



PLACE

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

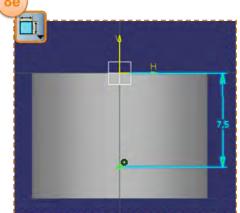
- Click on the "Hole" icon. a.
- Select "zx plane". b.
- Click on the "Positiong Sketch" icon to position the center of the hole. C.



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

Y	C	Constraint Definition	ו <mark>?</mark>
		Distance	Fix
		Length	Coincidence
		Angle	Concentricity
		🔲 Radius / Diameter	Tangency
•	<i>U</i> _	Semimajor axis	Parallelism

Define the distance between "xy plane" and the center of the hole as 7,5 mm. e. 8e

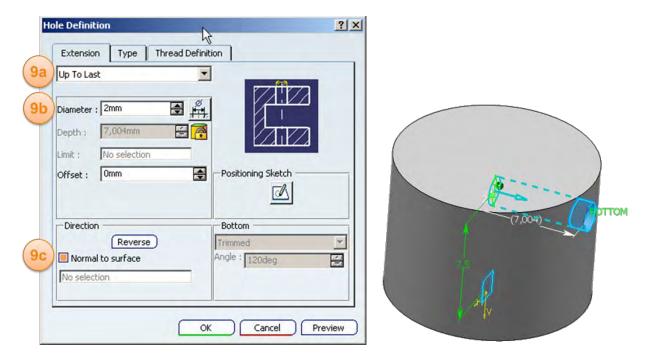


Then exit the sketch by clicking on the "Exit" icon. f.

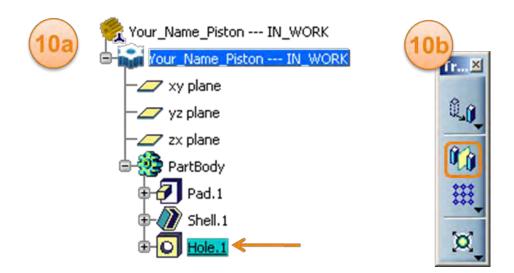




- 9. Define the hole as indicated opposite.
- a. "Extension": "Up To Last".
- b. "Diameter" as [2 mm] .
- c. "Check "Normal to surface".

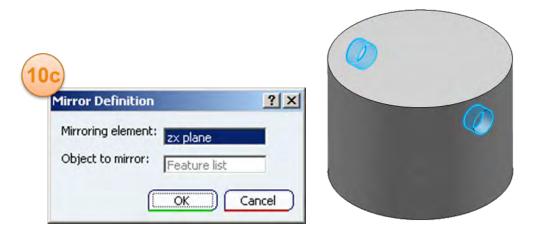


- 10. Perform a symmetry to define the hole on the other side of the piston.
- a. Select the hole created previously on the feature tree.
- b. Click on the "Mirror" icon. It should be on the lower right corner of your screen.

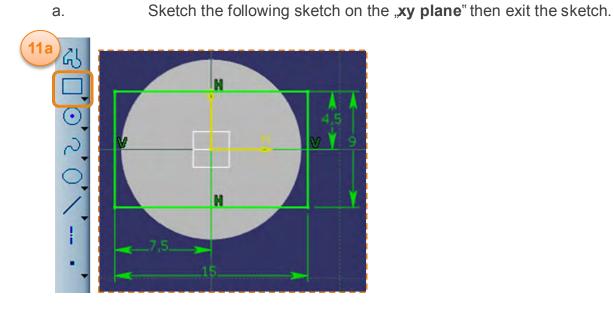




c. Select the "zx plane" as "Mirroring element" as indicated opposite.



11. Define the bottom face of the piston.



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

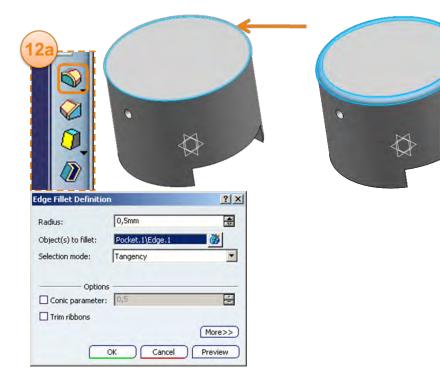


	Pocket Definition
	First Limit Type: Dimension Depth: 2mm Limit: INo selection
	Profile/Surface Selection: [Sketch.5 2] Thick Reverse Side
	Mirrored extent Reverse Direction
(1)	More>> v OK Cancel Preview

12. To finish, define dress up features.

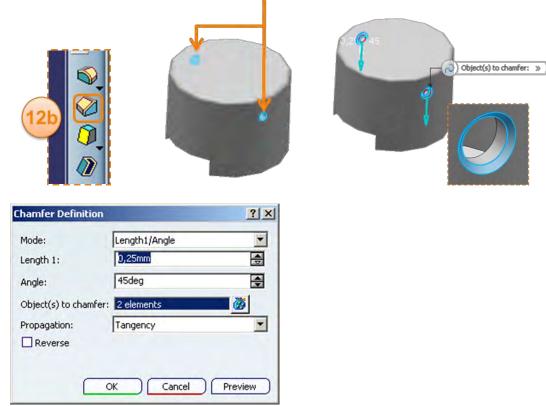
а.

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.

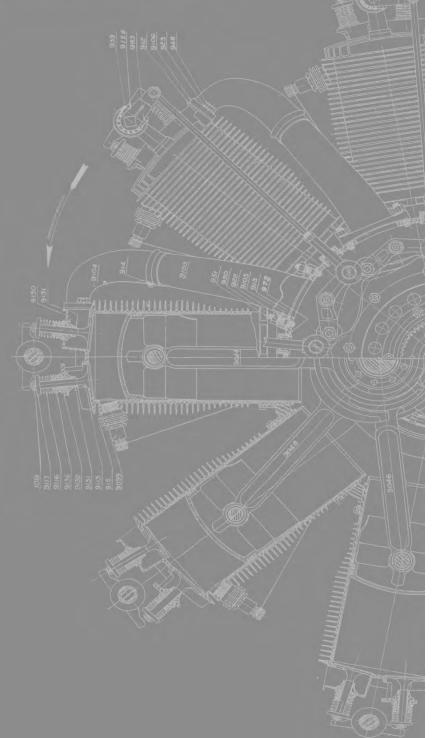








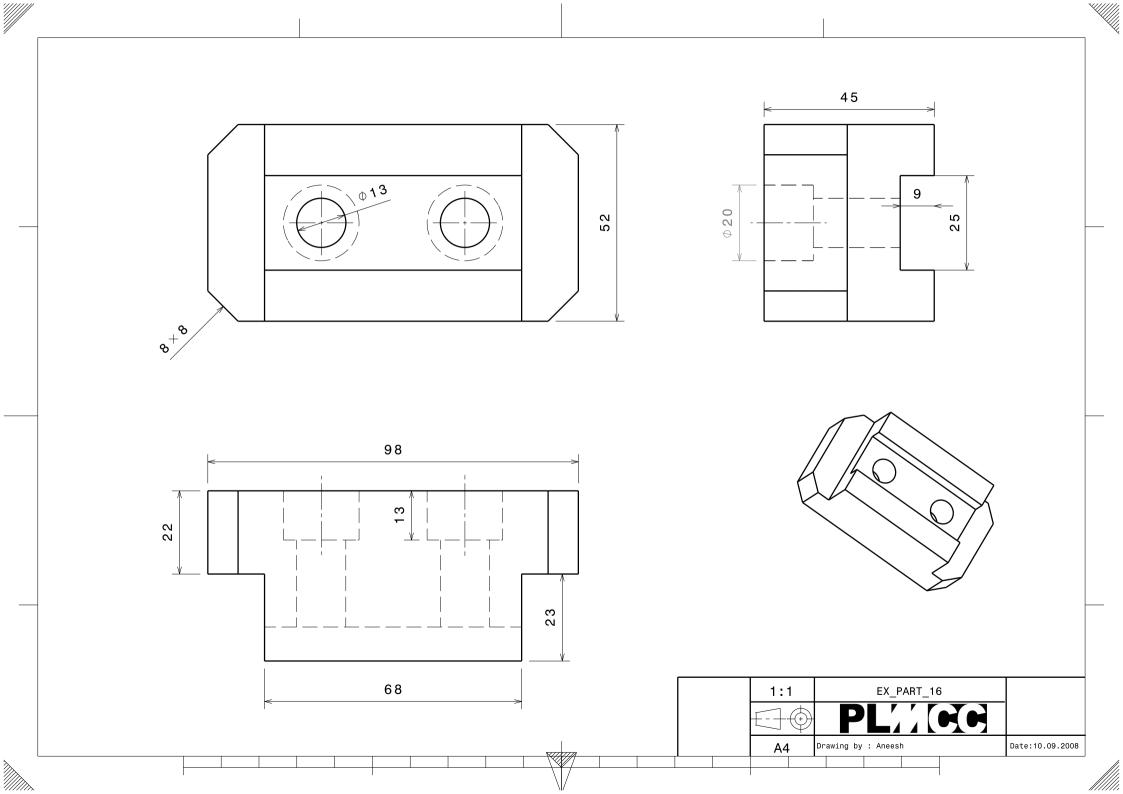
Exercises

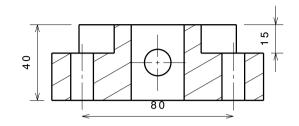


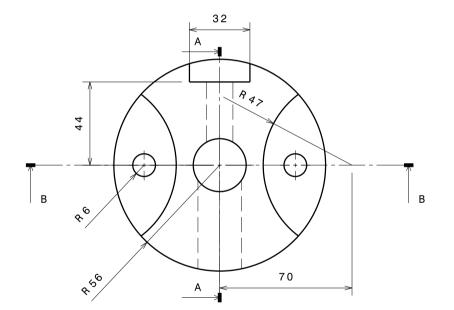


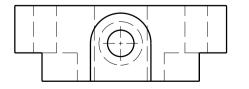


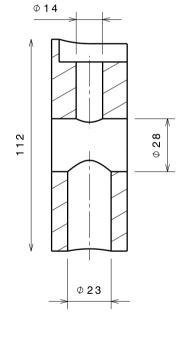




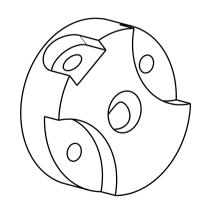


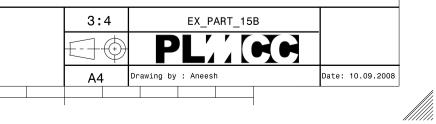


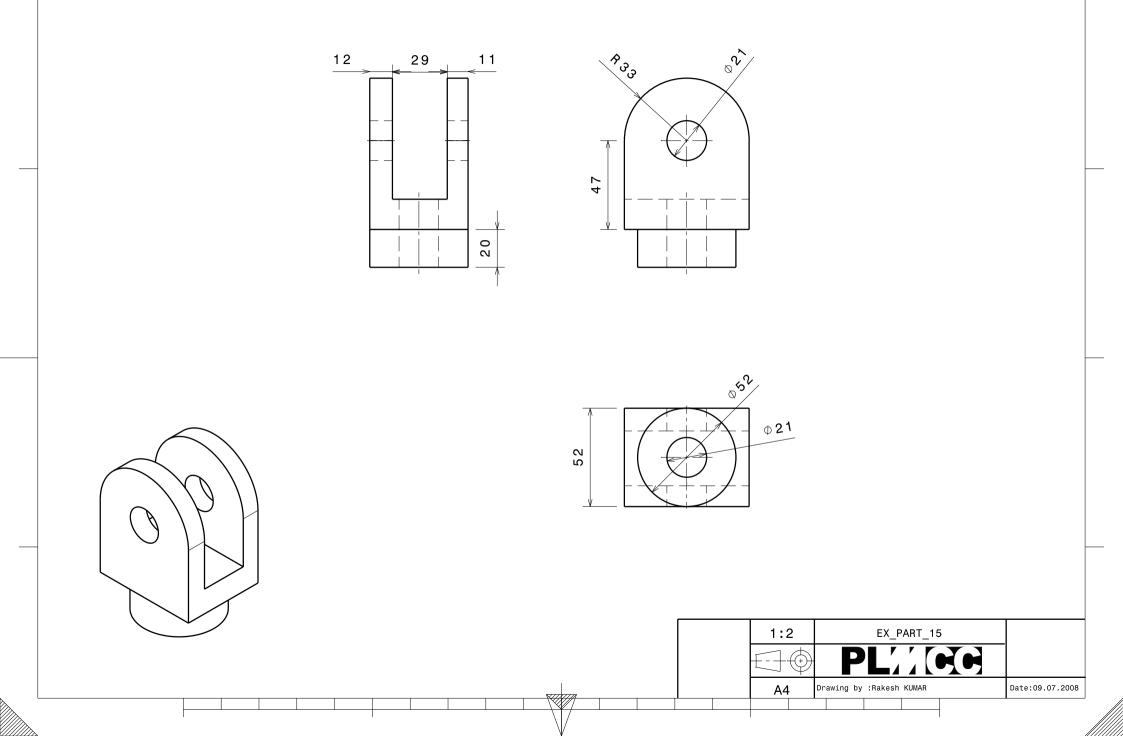


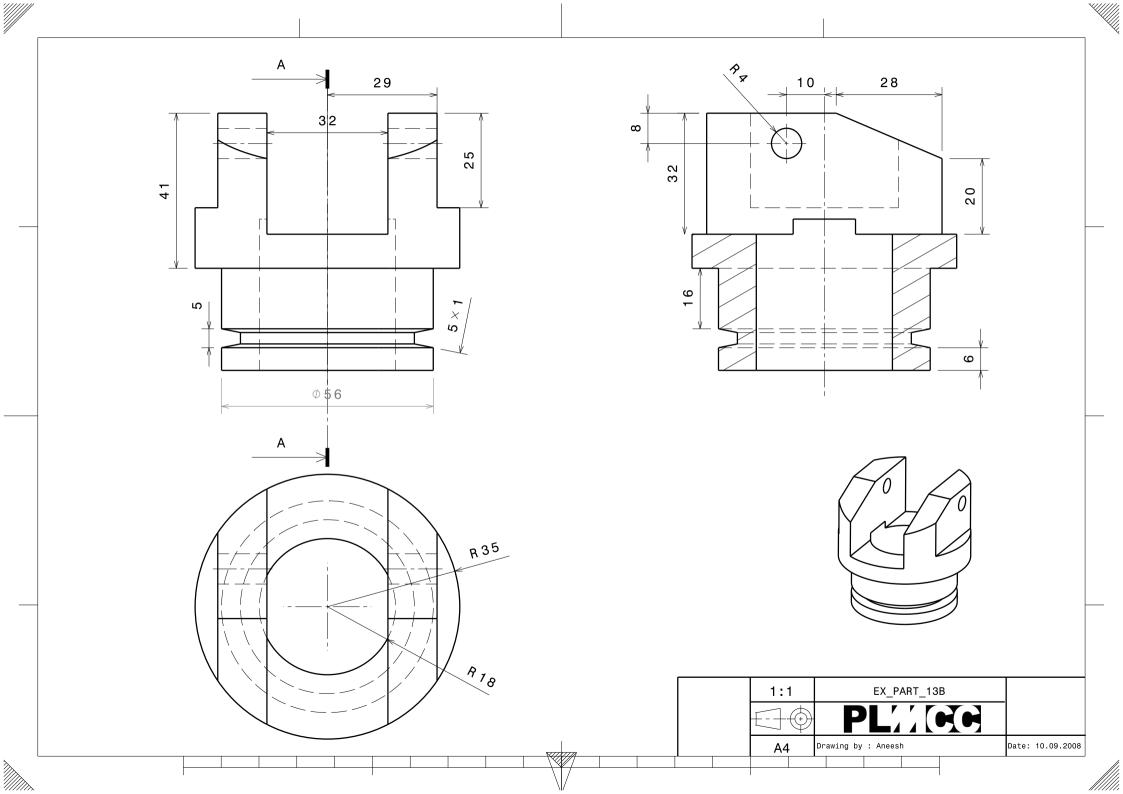


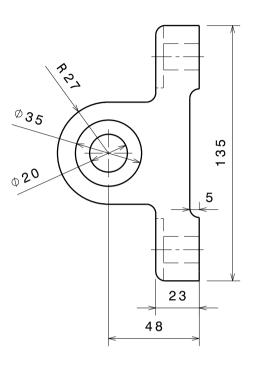
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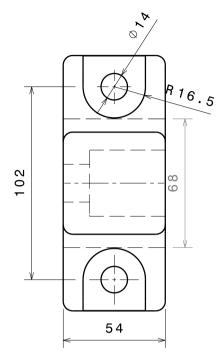


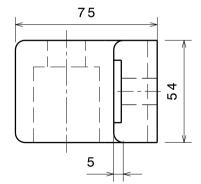


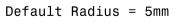


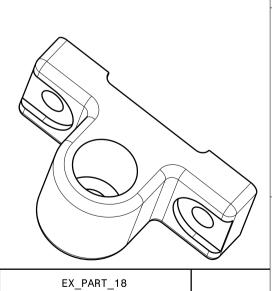


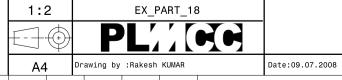


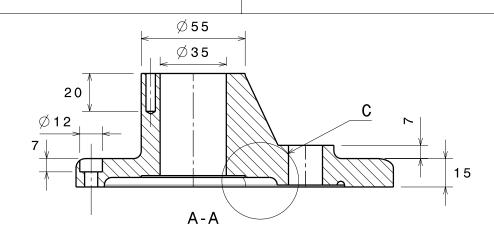


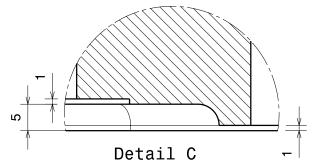


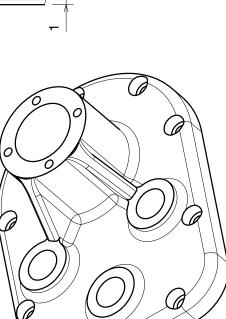




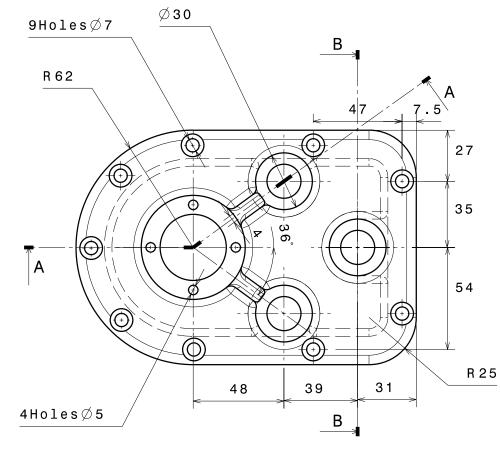


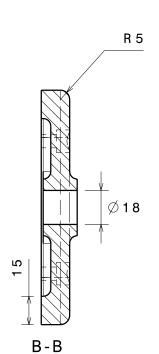


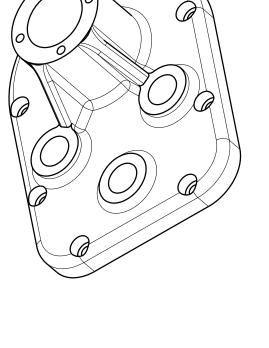


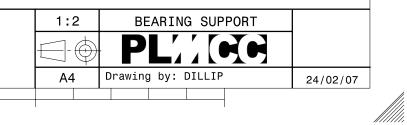


Radius=4











Functional Modelling







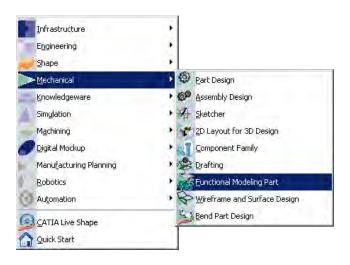
Functional Modelling Part

During these steps, you will design the Crankshaft. Functional Modeling Part offers a new approach to the development of 3D digital models.

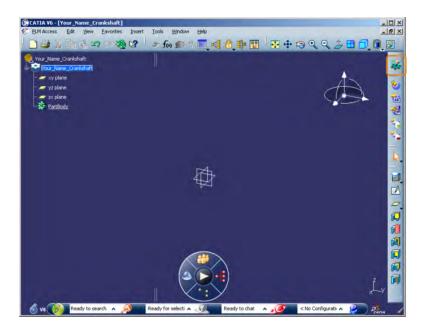
The objective of Functional Modeling Part is to enable product designers to focus on the functional goals and design constraints of their product.

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.



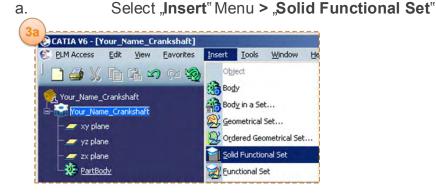


We will define 3 functional sets to design the "Crankshaft":

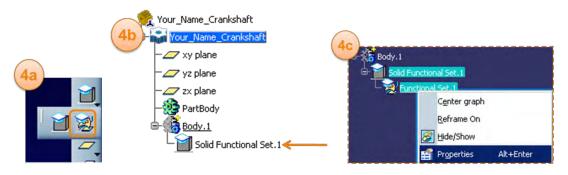
- General Form
- Connecting Rod
- Fuel_Air

We will finish the design by dress up features

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

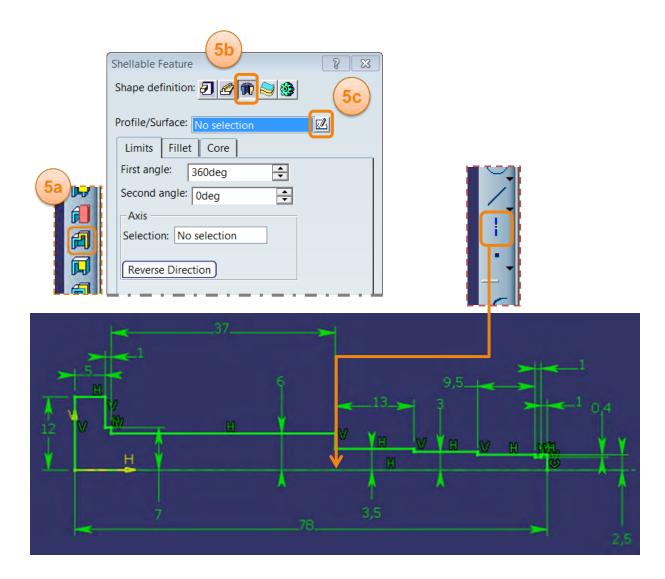


- 4. Create the first "Functional set"
- a. Click on the "Functional Set" icon
- b. Select "Solid Functional Set"
- c. Right click "Functional Set.1" > "Properties'
- d. Rename the "Functional Set.1" to General Form.



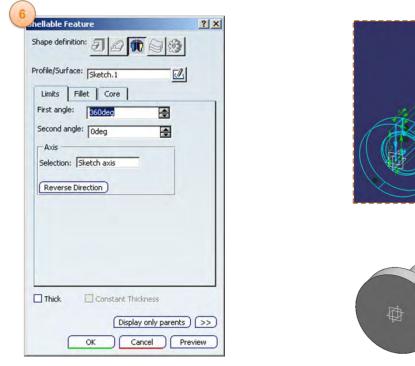
- 5. Create the first shape of the Crankshaft
- a. Click on the "Shellable Feature" icon
- b. Select "Revolve" as "Shape definition"
- c. Select "Sketch" icon and select "zx plane"
- d. Sketch as indicated below
- e. Exit the Sketch



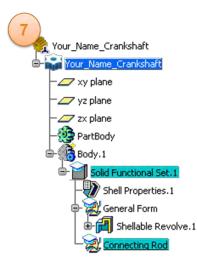




6. Define the dialog box as indicated to create a 3D part using **shaft**.



7. Create a new "Functional Set" and rename it "Connecting Rod"





- 8. Create a "Core Feature"
- a. Click on the "Core Feature" icon

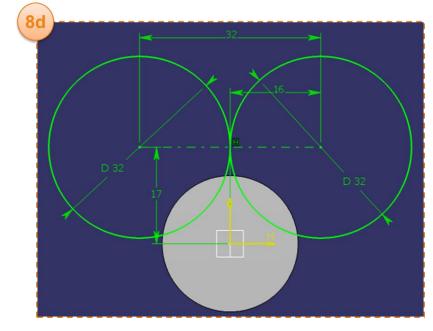




b. Select "Prism" as "Shape definition"
c. Select the "Sketch" icon and select the face of the crankshaft as shown opposite.



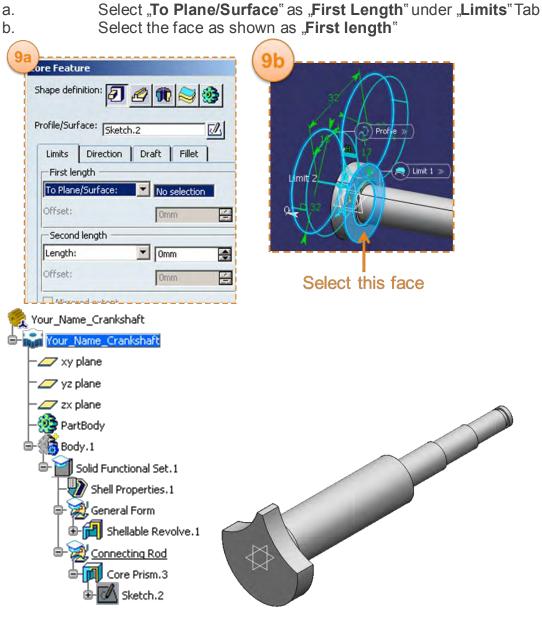
d. Sketch as indicated below



e. Exit the Sketch



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

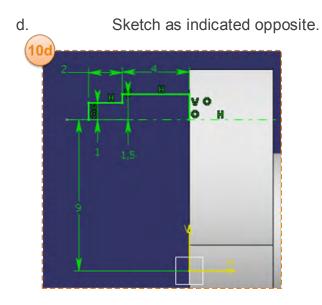


10. Create an "Added Feature"

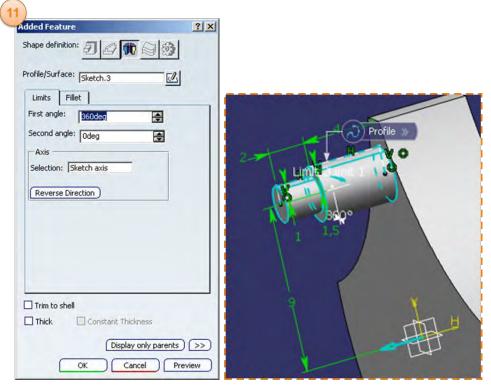


- a. Click on the "**Added Feature**" icon
- b. Select "**Revolve**" as "**Shape definition**"
- c. Select "Sketch" icon and select the "ZY plane"

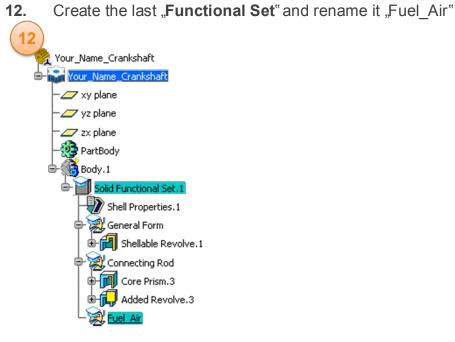




- e. Exit the Sketch
- 11. Define the dialog box as indicated to create a 3D part using **shaft**.



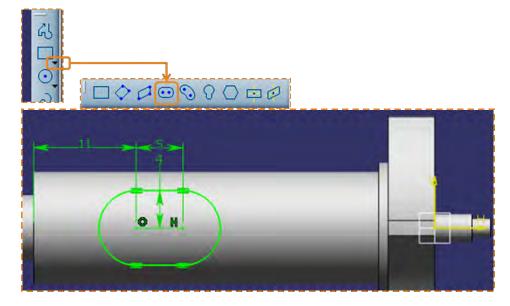




13. Create "**Protected Feature**"



- 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.

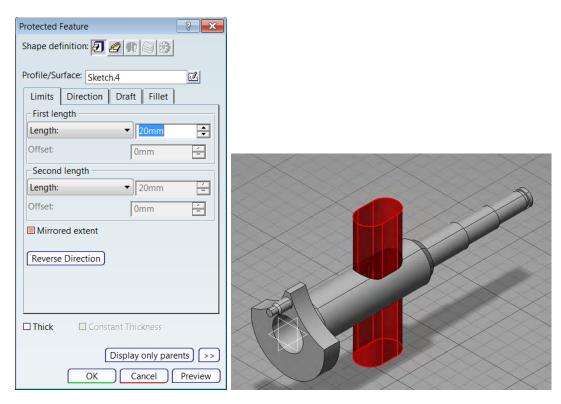


e. Exit the Sketch

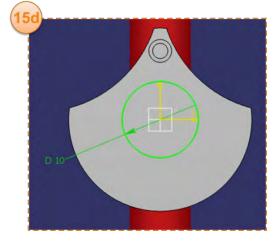


- 14. Define the dialog box as indicated opposite.
- a. Check "Mirrored extent"

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

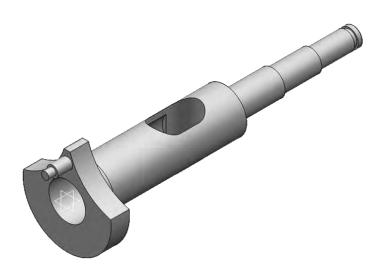


- 16. Define the dialog box as indicated opposite.
- Uncheck "Mirrored extent" a.
- Specify the "First length" as [31mm] b. C.
- If necessary use "Reverse Direction

Profile/Surface: Sketch.5	These the second		
Limits Direction Draft Fillet First length Length: 31mm			
Length: 31mm			
			First length
	Roffe 2 - 31	🔽 31mm 🛃	ength:
Offset:		0mm 🔚)ffset:
Second length			Second length
Length: 🔽 Omm		🔽 Omm 🛃	ength:

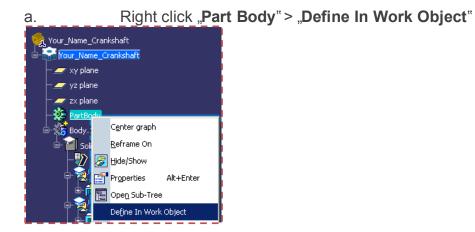
- Hide previously created "Protected Prism" 17.
- Right click "Protected Prism.5" > "Hide/Show" a.
- Repeat this step for "Protected Prism.6" b.





PL//ce

18. Finish the design by "Dressup" Features



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

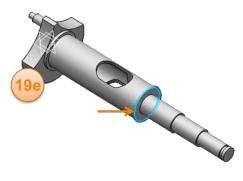


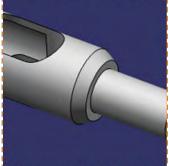
- a. Click on the **"Chamfer, local modifier**" icon.
- b. Select "Length1/Angle" as "Mode"
- c. Specify "Length1" as [2mm]
- d. Specify "Angle" as [30deg]

Chamfer Definition	?
Mode:	Length1/Angle
Length 1:	2mm
Angle:	30deg
Object(s) to chamfer:	Shellable Revolve.1\Edge.1
Propagation:	Tangency
Reverse	

e.

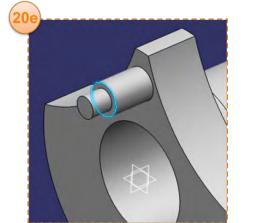
Select the edge as shown opposite.

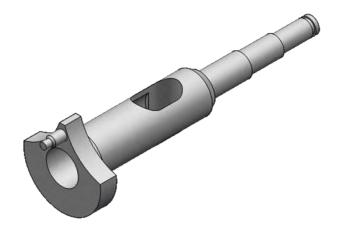




PLACE

- 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

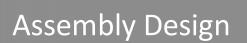


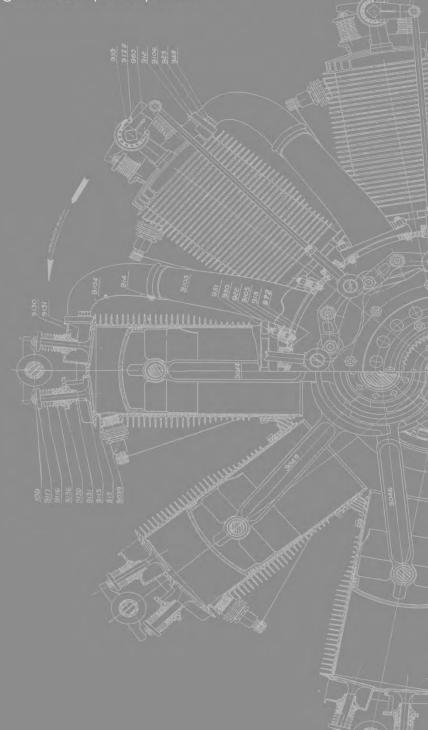


Don't forget to "Propagate" your work to save it in the data base.

















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.

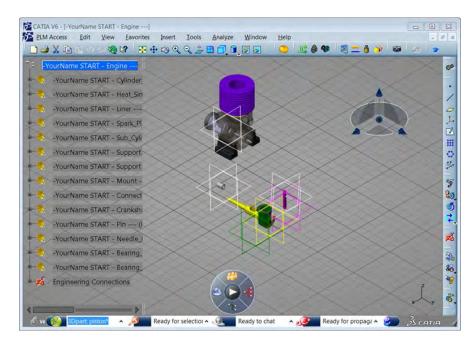
- 1. Import the micro motor 3DXML
 - a. Click PLM Access > Import > 3D XML....
 - b. You will find it in the **Assembly Design > Starting_Data>Assembly** folder.
 - c. Select the file named : Engine_Starting_Data.3dxml
 - d. Check Import As New.
 - e. As duplication string enter "your name_'
 - f. Click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

	Import	3D XML		2	3
CATIA V6 - [Title Like *] <u>PLM Access</u> Edit <u>Vi</u> ew <u>F</u> av	Path: over File N	V6\Data_Discover_PLM_2_0_on_V ame:	6_Platform 4_5_Asse	mbly_Design\Starting_Data	
New Ctrl+N		ne Date Crea Ingine_starting_data 05/30/12	ted size 10:50 19529 KB		
3D Part Drawing Close		ston-starting_data.3 05/30/12	10:50 90 KB		
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Open Propagate Ctrl+Shift+S Propagate All Local Save		t Option			
Export	XML			YourName	
Print Ctrl+P	TIA File		port into folder:	OK Cancel	
Exit				ssembly Design (un	- der
Infrastructure	•			cal) Workbench to c	
Systems	and a	e Functional Design	assembly		
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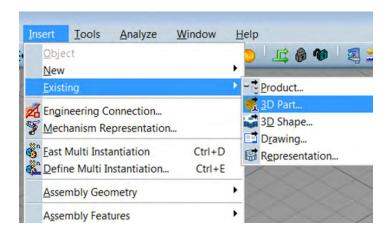


2. The assembly will open in a Navigator window.

- 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.

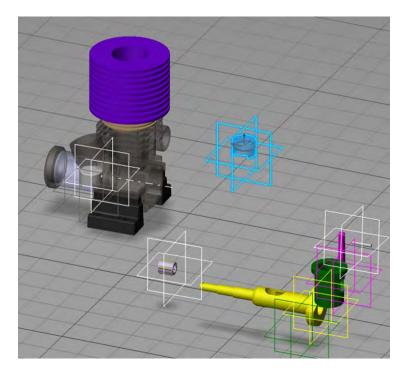


- d. Enter *"3dpart: YourName_piston**"in the search bar.
- e. Click Search.
- f. Double click on the matching search result.



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



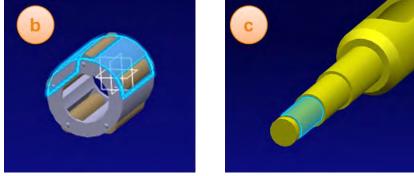


- 4. Assemble the crankshaft and the needle bearing.
 - a. Click on the **Engineering Connection** icon. A panel will open.

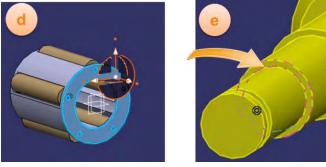
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50		Select an elemer	nt



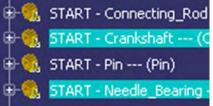
- b. Click on the **Surface** of the needle bearing to select its **axis**.
- c. Click on the Surface of the crankshaft to select its axis.



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

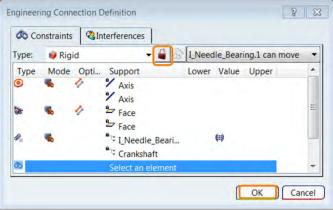


f. Select the the crankshaft and the needle bearing in the **specification tree**.



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







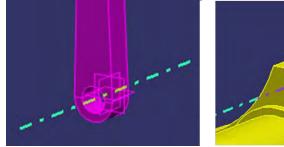
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[Yes	No	Cancel	

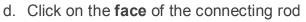
i. Then click Yes to validate the interference.

5. Assemble the Crankshaft and the connecting rod

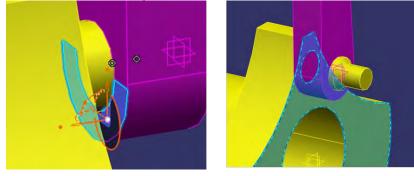


- a. Click on the **Engineering connection** icon.
- b. Click on the temporary axis in the hole of the connecting rod.
- c. Select the temporary axis of the rotational axis of the crankshaft





e. Click on the parallel face of the crankshaft

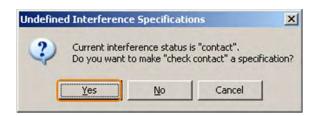


- f. Click on the unlocked padlock
- g. Click **OK** to close the pannel

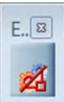


Co Co	onstraints	In	terferences	5							
Type:	S Rev	olute		-		(a)	Conne	cting_Rc	d can m	ove	•
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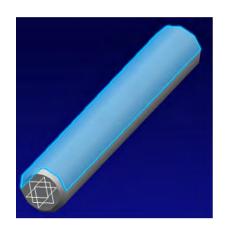
h. Click Yes to validate the interference



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



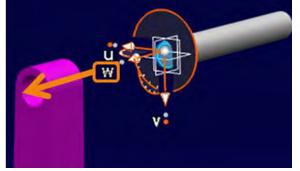
- a. Click on the Engineering connection icon
- b. Click on the **Surface** of the pin to select its axis.
- c. Click on the Surface of the connecting rod to select its axis.



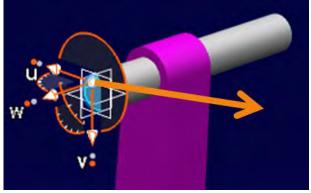




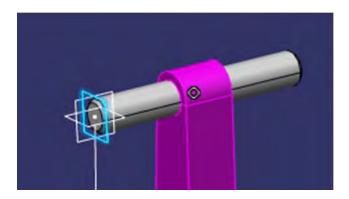
- d. Take the **Robot** and put it on the surface of the **Pin**.
- e. When the **Robot** is on the surface, pull the **axis w** to move the **pin** between the two surfaces of the **connecting rod**.

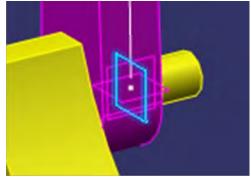


f. After having moved the **Pin**, remove the robot by dragging it off the part to the background.



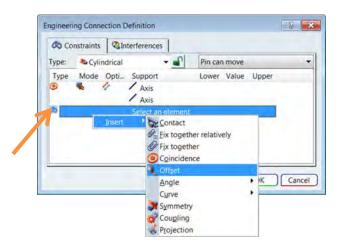
- 7. Create a revolute joint the pin and the connecting rod (to remove)
 - a. Click on the **first plane** of the pin as shown opposite.
 - b. Click on the **parallel plane** of the connecting rod as shown opposite.



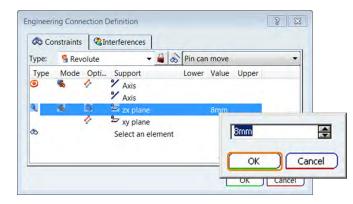




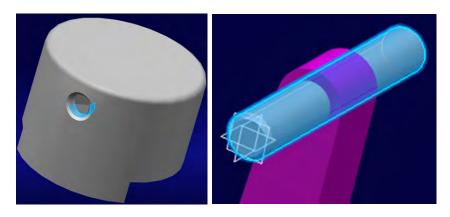
c. Right click on the connection type and select **Offset.**



- 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

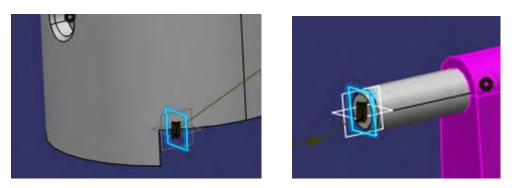


- 8. Create a rigid joint between the piston and the pin
 - a. Click on the surface of the piston hole and the Pin

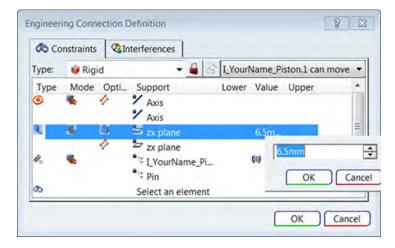


PLMCC

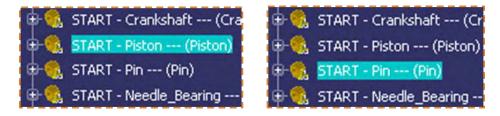
- b. Select the **plane** of the piston as shown.
- c. Select the **parallel plane** of the pin as shown.



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



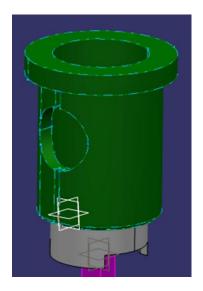
f. Click on the piston and the pin product in the specification tree .The Engineering connection type has been changed to **rigid**.



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

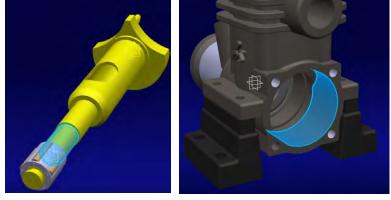


- b. Lock the padlock
- c. Click OK

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Туре	Mode Op				Lower	Value	Upper	
۲	s 1	Axis Axis						
00		Select ar	n eleme	ent				



- 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

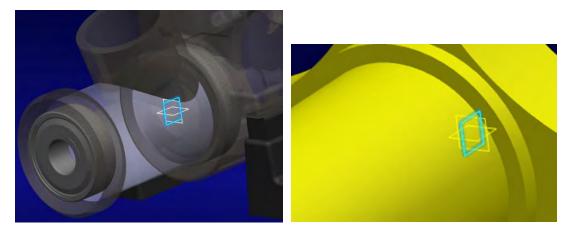
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c. Right click the connection mode as shown below and select **Controlled.**

Гуре:	nstraints Cylindrical	- ¶	Cranksh	aft can	move	
Type		xis xis	Lower	Value	Upper	
8 8	Controlled	ction ction n element				

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.**



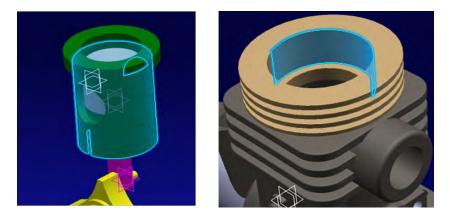
- 11. Hide the Cylinder_head, the Heat_sink, and the Spark_plug.
 - a. Select the Cylinder_Head, the Head_Sink and the Spark_Plug.
 - b. Right-click and select Hide/Show.



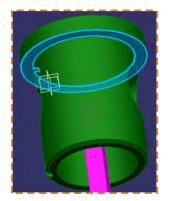
	ourName START - Engine	1	A	1	1
	-YourName START - Cylinder_Head	R	Center graph	1	-
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			<u>C</u> opy	Ctrl+C	1
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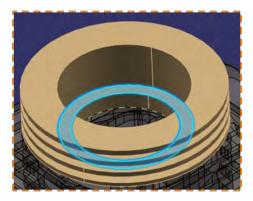
12. Create a rigid joint between the 'liner' and the Sub_Cylinder_head

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



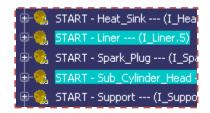
b. Create a planar coincidence between the liner and the **Sub_Cylinder_head**.







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



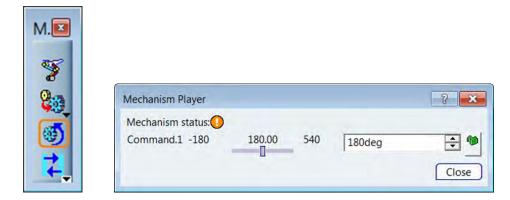
d. Validate the Engineering connection

13. Create a mechanism Representation

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



c. Simulate by clicking on Mechanism Animation



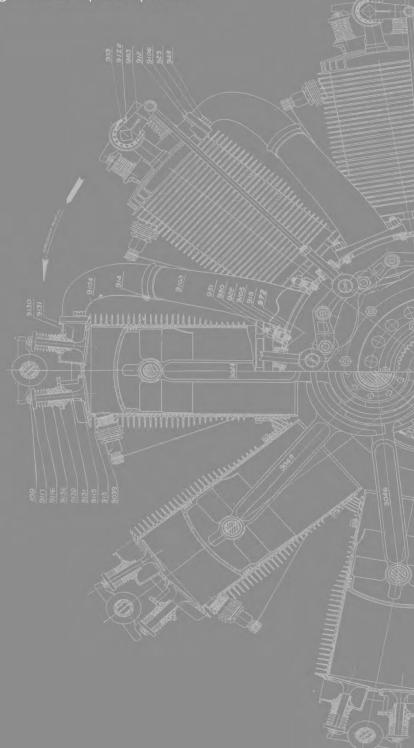
Don't forget to 'Propagate' your work to save it in the data base







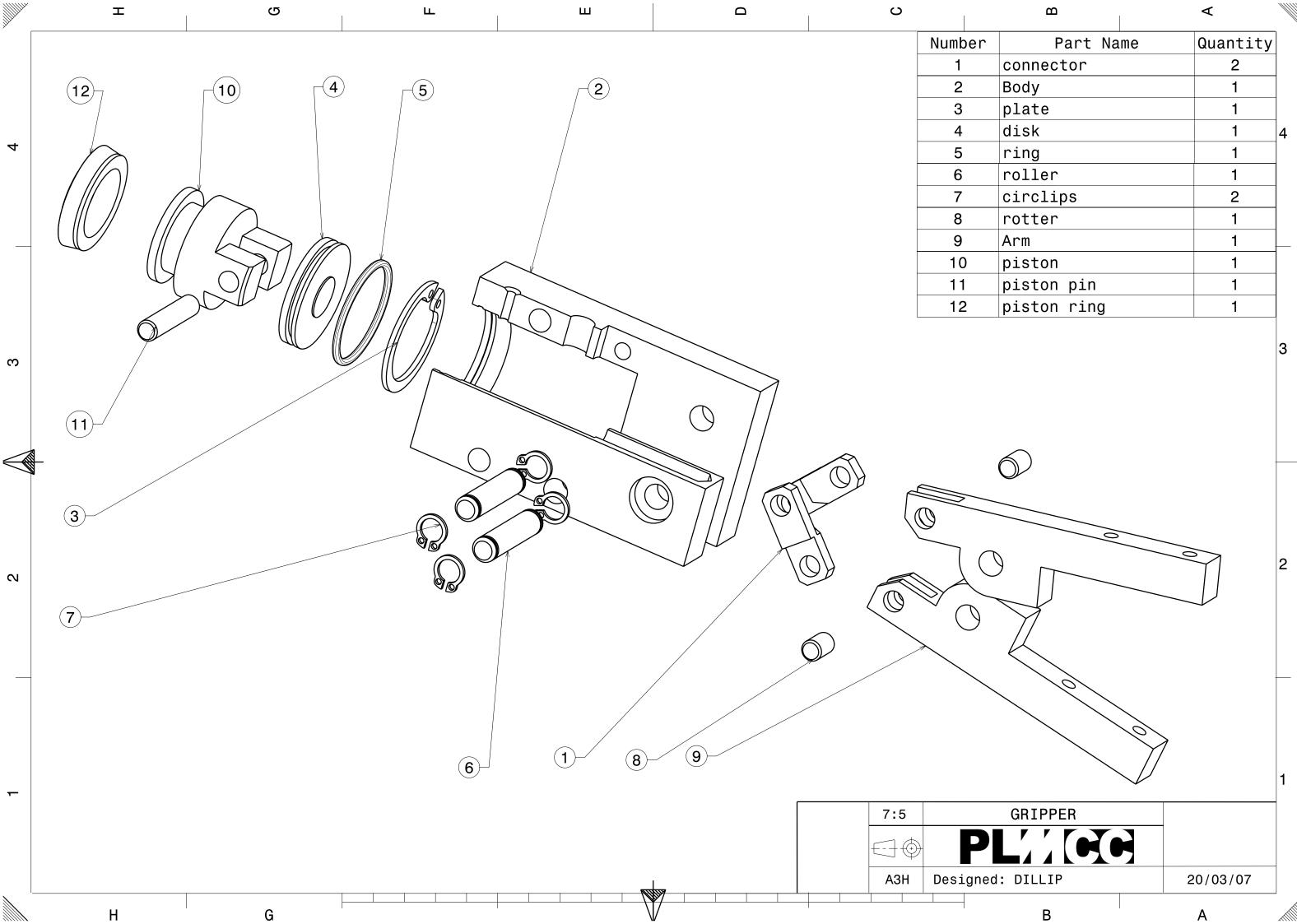
Exercises



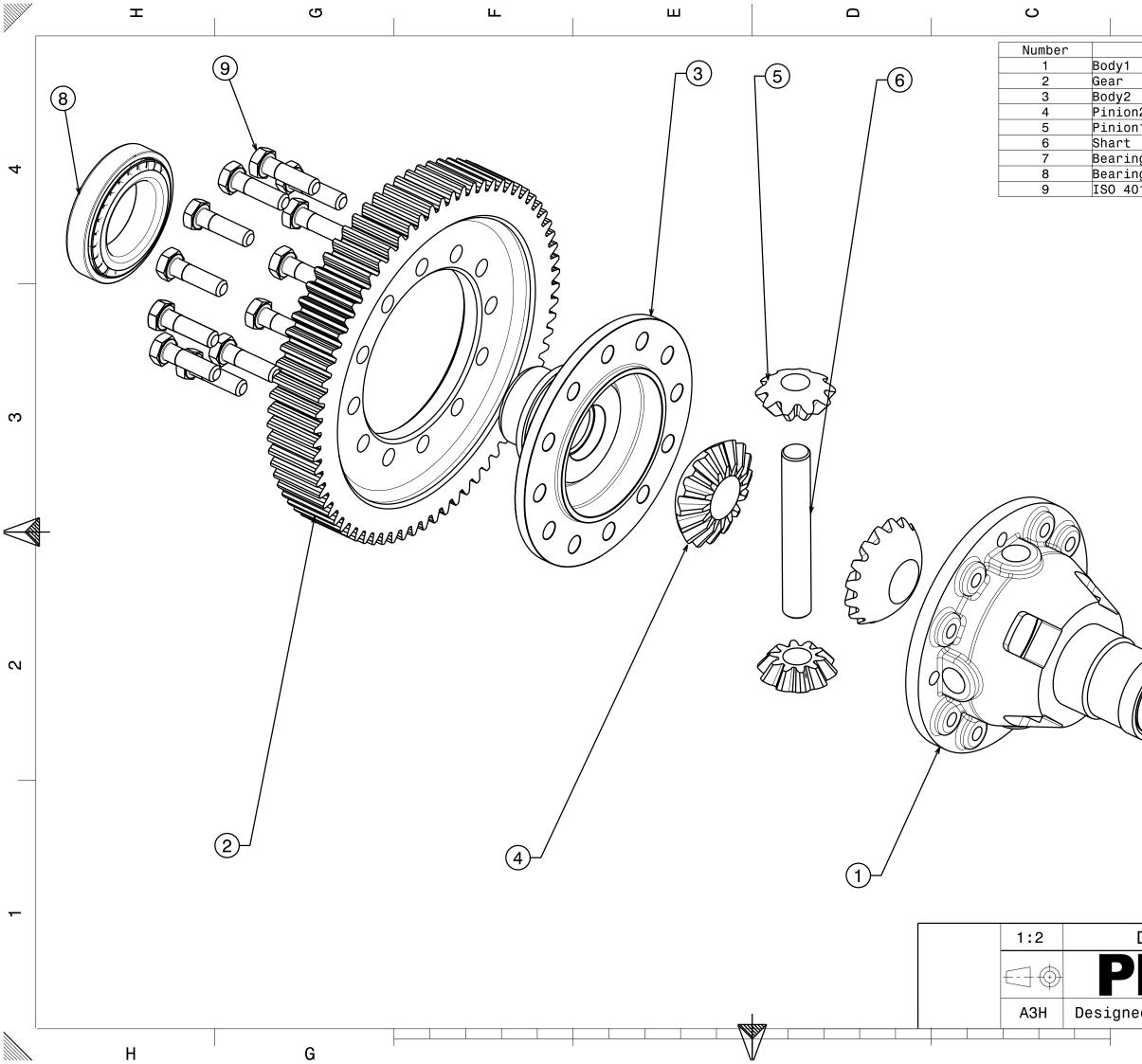




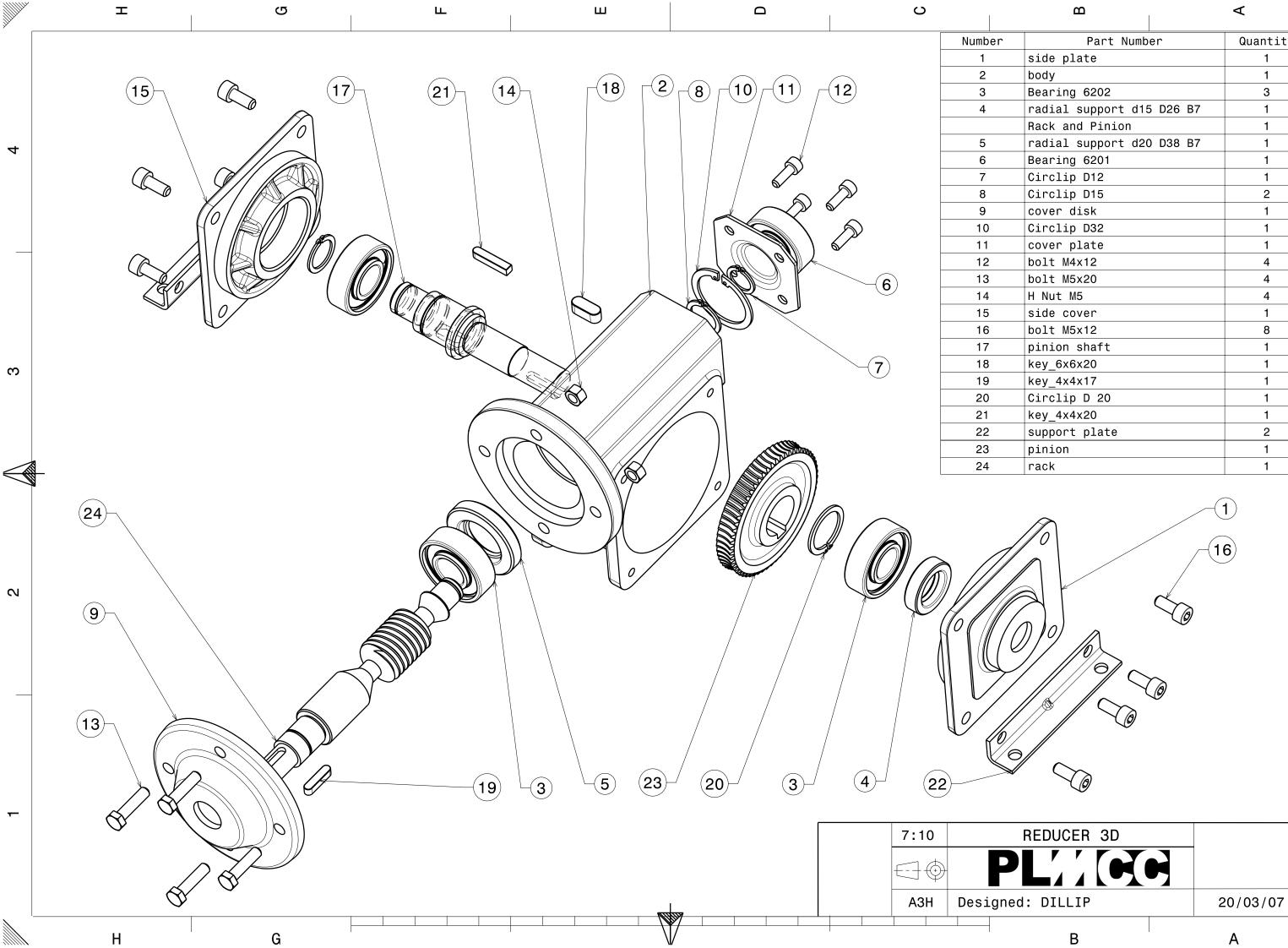




Ш		R	
Part Na	me	Quantity	
connector		2	
Body		1	
plate		1	
disk		1	4
ring		1	
roller		1	
circlips		2	
rotter		1	
Arm		1	
piston		1	
piston pin		1	
piston ring		1	

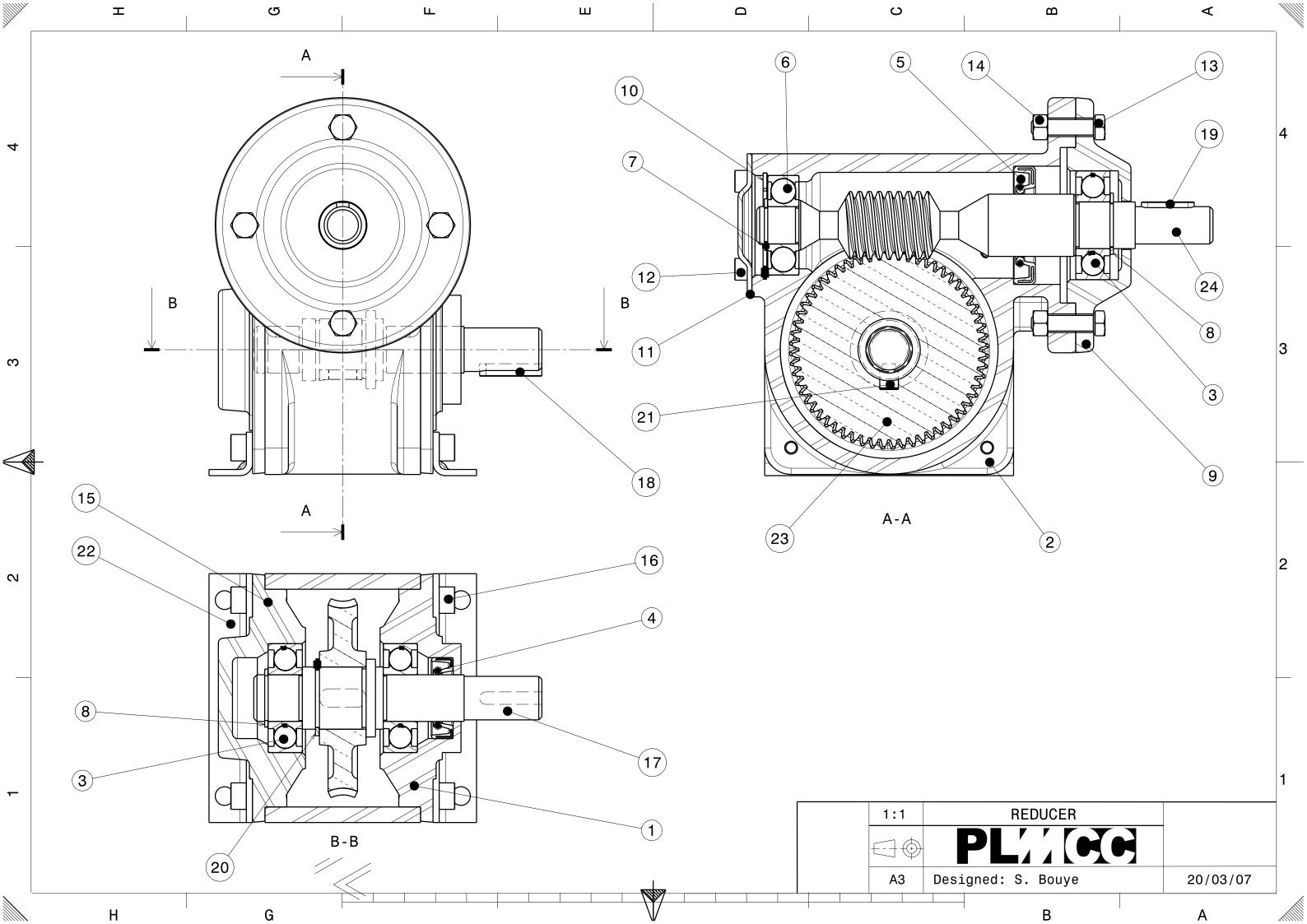


۵	A	
Part Name	Quantity 1	
n2	1 1 2	
n1	2	
ng 18690_18620 ng 32010	1	4
014 Bolt M10x45	13	
		3
_		
		2
		-
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DIFFERENTIAL	•	
ed: DILLIP	20/03/07	
B	А	



B	A	
Part Number	Quantity	Ì
side plate	1	1
body	1	1
Bearing 6202	3	1
radial support d15 D26 B7	1	1
Rack and Pinion	1	
radial support d20 D38 B7	1	4
Bearing 6201	1	1
Circlip D12	1	1
Circlip D15	2	1
cover disk	1	1
Circlip D32	1	1
cover plate	1	1
bolt M4x12	4	1
bolt M5x20	4	-
H Nut M5	4	-
side cover	1	-
bolt M5x12	8	-
pinion shaft	1	-
key_6x6x20	1	3
key_4x4x17	1	-
Circlip D 20	1	-
key_4x4x20	1	-
support plate	2	-
pinion	1	-
rack	1	-
	- <u>1</u> (16)	2
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Kinematic Analysis





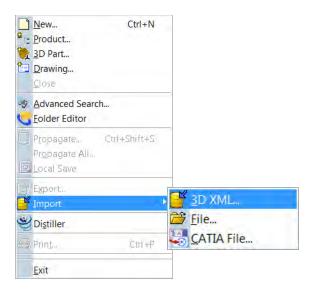




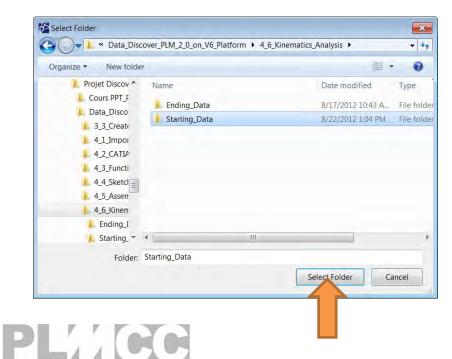
During these steps, you will simulate the movement of the piston and the crankshaft using kinematics analysis.

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



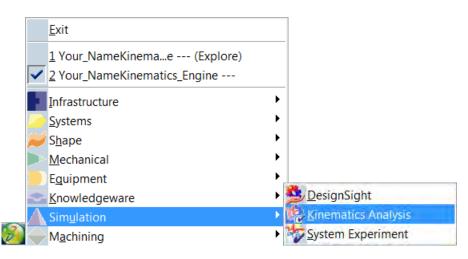
- d. Select Kinematics_Engine.3D XML file to import.
- e. Check Import As New.
- f. As duplication string enter Your_Name
- g. Click OK.

Import 3D XML				X
Path:				
erV6\Data_Discover_PLM_2	2_0_on_V6_Platfo	rm\4_6_Kiner	matics_Analysis	Starting_Data 🛄
File Name:				
Name	Date Created	size		
Kinematics_Engine.3d	08/22/12 13:04	19649 KB		
			•	
Import Option				
Import As Reference				
Import As New	Duplica	tion string:	Your_Name	
	Dapito		Tour_Nume	
3D XML Playe	er Preview	~) [ОК	Cancel

- 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.

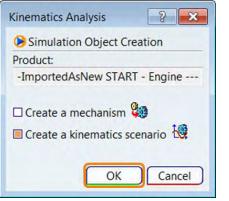
2. Access the workbench

- 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

Name Ssim000081	
	*
Description	

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.

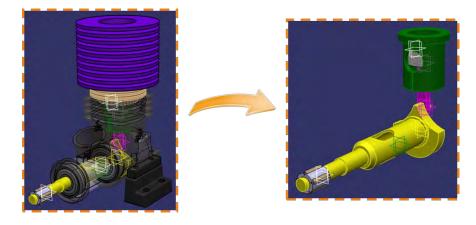
Excitations	Probes	Parameters	_
Start time:	Os		
End time:	100s		
Time step:	1s		



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**.



KinSim_YourName	
Model ImportedAsNew START - Er	Law Excitation
MechanismBuggy	Name: Law Excitation.1 Supports: 1 Command Angle formula: 0deg
-ImportedAsNew START	OK Cancel

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

Law Excitation	
Name: Law Excitation.1	
Supports: 1 Command	
Angle formula: Odeg	f⊛ <u>E</u> dit formula
ОК	C Add tolerance
	Add Multiple Values
	Range
	💋 Ed <u>i</u> t Comment
	🔒 Lock

- 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

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Excitations\Law Excitatio	on.1\/	Angle	=
`Excitations\Law Excitati	ion.1	Time` *10deg/1s	
Dictionary		Members of Parameters	Members of All
Parameters		All	`Excitations\Law Excitation.1\Angle`
Design Table Law List Math Messages and macros Object Operators	III	Renamed parameters Angle Time Feature	'Excitations\Law Excitation.1\Time' 'Scenario.1\Scenario Parameters.1\CATKins' 'Scenario.1\Scenario Parameters.1\CATKins' 'Scenario.1\Scenario Parameters.1\CATKins' Excitations 'Excitations\Law Excitation.1'
Excitations\Law Excitation	on.1\7	lime	Os T



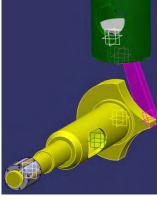


- g. Click simulate and generate results icon
- h. Click **OK** and wait till the end of the calculation.

Simulate and Generate Results	<u>? ×</u>
Scenarios: Scenario.1 - Kinematics Sce	ei 듣
OK Cancel Pre	view

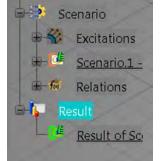
- View the results of the simulation 6.
 - a. Click on the **play button** of the compass to see the motion of the piston
 - b. You can control the motion of the kinematics by clicking and holding the cursor at the top of the compass





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.





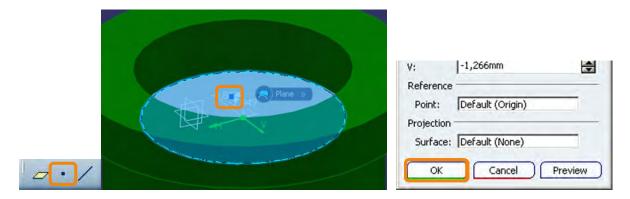


- b. Click on measure probe icon
- c. In the specifications tab, select Minimum distance
- d. Select the face at the top of the piston
- e. Select the face at the top of the Liner
- f. Click OK



	Measure Between	8 🛛
	Specifications Customize	
R	Mandatory	
	Type of measure Minimum distance	
	Calculation mode: Exact else approxima	ate 🔻
	Selection1 Plane in Solid.1Pisto	on 🦉
	Selection2 Plane in Shaft.1I_Lir	ner 🖉
	Parameters	
	Selection 1 mode Any geometry	-
	Selection 2 mode Any geometry	-
	Other Direction Compass Direction	•
	ОК	Cancel

- 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



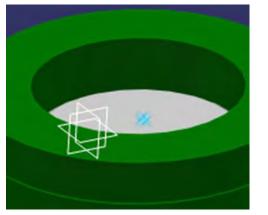
- 10. Create Speed and acceleration measure
 - a. Double click on Scenario in the specification tree





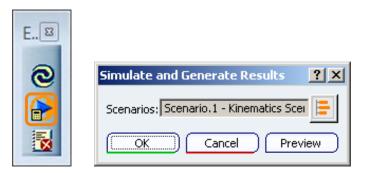


- b. Click on the point, speed and acceleration icon
- c. Select the point you have just created
- d. Click OK



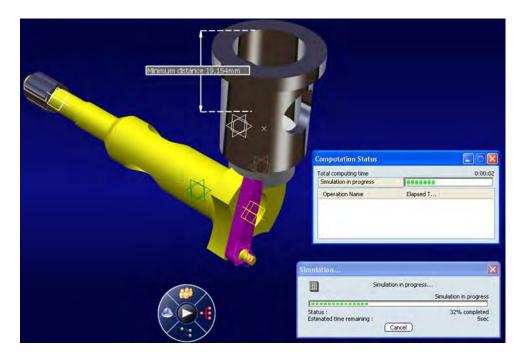
Position,Speed and Acc
Name: Position,Speed and Acceleration.1
Supports: 1 Point
Reference product: -ImportedAsNew ST/
Projection axis: No selection
Customize View
OK Cancel

- 11. Simulate the results
 - a. Click simulate and generate results
 - b. Click OK



c. The software creates the simulation.

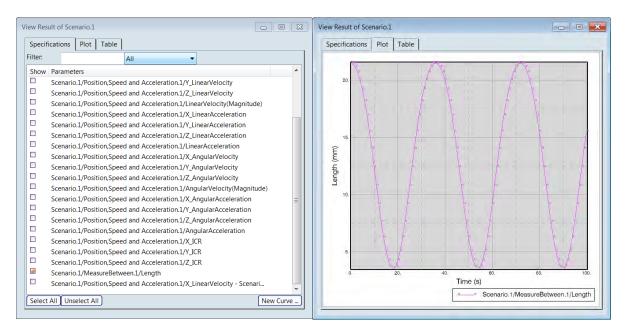




12. View the results:

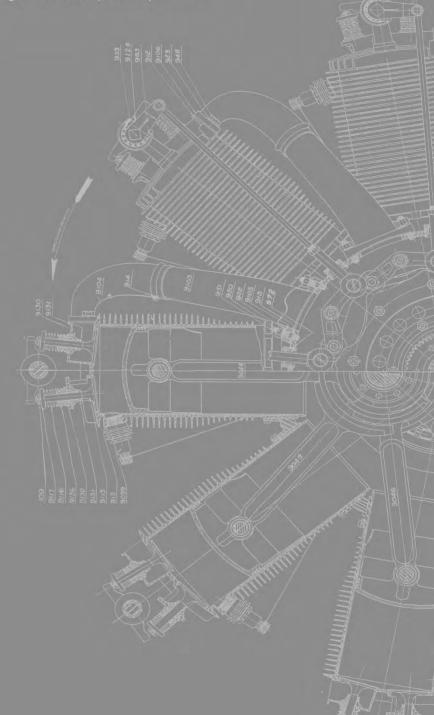


- a. Click on the **export probe results** icon.
- b. Click on a Scenario.1 result e.g. Scenario.1/MeasureBetween.1/Length.
- c. Click on the plot or table tab to view results















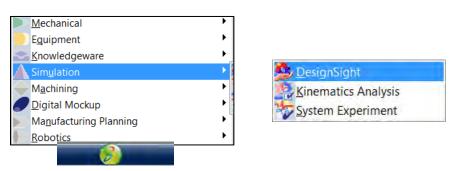


During these steps, we will perform a **stress simulation** and analyse the connecting rod.

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.



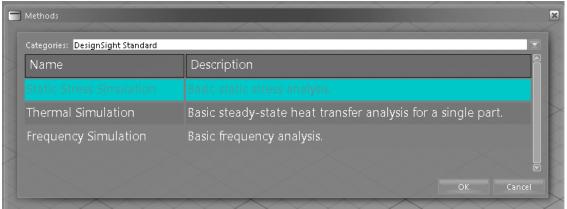
- 6. Name it as [Your_Name_Simulation_Connecting_Rod]
- 7. Click "Finish" to validate the name

۵	DesignSight ,	/ Simulation (SMB)		• 🗙
	👌 Simulat	tion (SMB)		
	Title	Your_Name_Simulation_Connecting_Rod		8
	Name	Ssim000163		*
	Description			
	· ·			
		Previous Next Finis	h C	Cancel

8. Click "Static Stress Simulation"



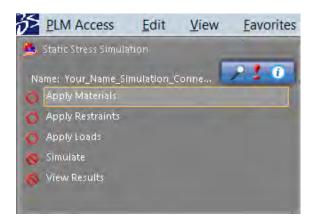
9. Click "OK" to validate



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

Finite elemen	t model / Physical Representation	
🔓 Finite e	element model	
Title	Your_Name_Finite_Element	8
Name	Sfem001022	*
Description		
-		
	Previous Next Finish (Cancel

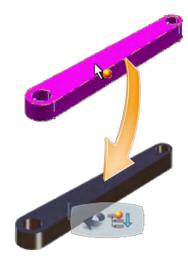
11. Click on "Apply Materials"



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



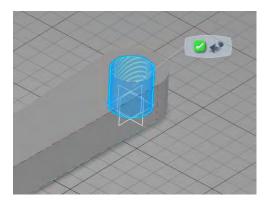
Display name 📥	Title
😑 -ImportedAsNew Aluminium Aluminium	-ImportedAsNew Aluminium
🗢ImportedAsNew Aluminium Aluminium	
💛 -ImportedAsNew Brass Brass	-ImportedAsNew Brass
😑 -ImportedAsNew Brass Brass	-ImportedAsNew Brass
😳 -ImportedAsNew Bronze Bronze	-ImportedAsNew Bronze
😑 –ImportedAsNew Bronze Bronze	-ImportedAsNew Bronze
🦲 -ImportedAsNew Chroma Chroma	-ImportedAsNew Chroma
🍑 -ImportedAsNew Chroma Chroma	-ImportedAsNew Chroma
🌙 -ImportedAsNew Copper Copper	-ImportedAsNew Copper
💛 -ImportedAsNew Copper Copper	-ImportedAsNew Copper
😑 -ImportedAsNew Gold Gold	-ImportedAsNew Gold



- Click on "Apply Restraints"
 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.

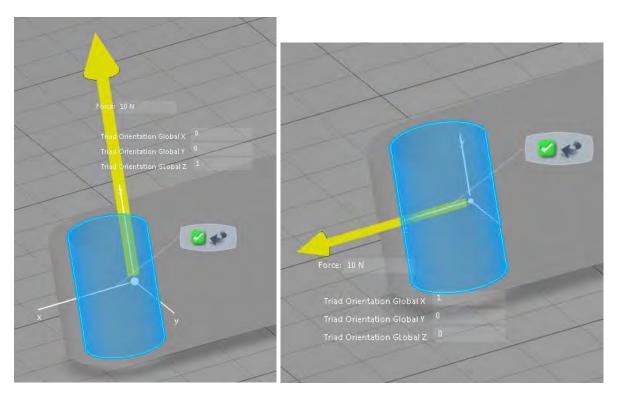


- 16. Click on "Apply Loads"
- 17. Apply a remote force on the inner surface of the smaller hole.



🏂 Static Stress Simulation	
Name: Your_Name_Simulation_Conne	Editor
🥜 Apply Restraints (1)	
O Apply Loads	Name: Remote Force,1
👩 Simulate	Support: 2 Faus 🔆 Geometry
🔞 View Results	Ci OK Cancel

- 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.

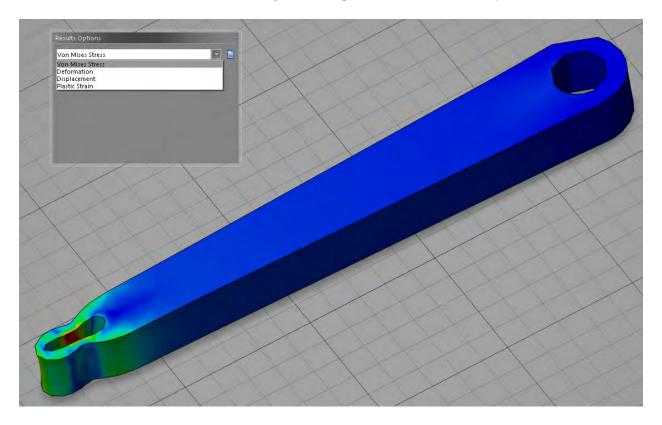




21. The window shown below will appear to show you the progress of your simulation



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







Machine Programming





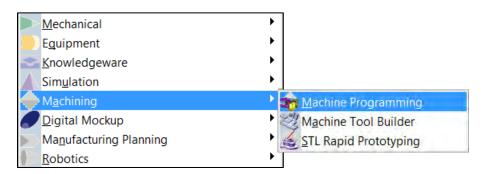




Machine Programming

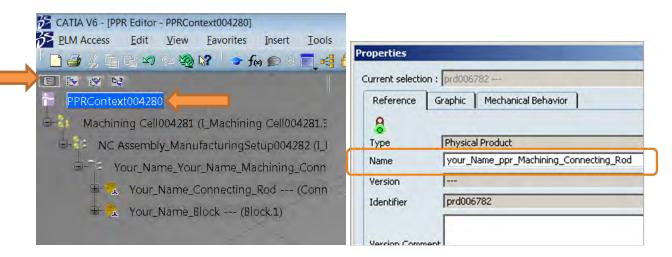
During these steps, we will generate the NC program to machine the connecting rod. **Prismatic Machining** enables you to define and manage NC programs dedicated to machining parts designed in 3D wireframe or solids geometry using 2.5 axis machining techniques.

- 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.



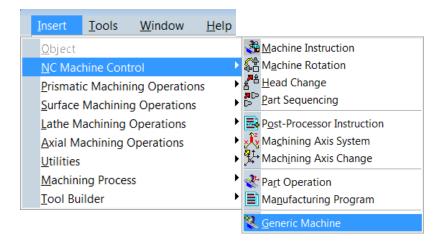
4. We will rename the current 'ppr file'.

- 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.

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7. The following dialogue window opens.

Generic Machine
ie ei Ver Fr
Name: 3-axis Machine.1_NCMachine000793
Comment:
Spindle Tooling Compensation Numerical Control NC Output
Home point X: Omm
Home point Y: Omm
Home point Z: 100mm
Crientation J: 0
Crientation K: 1
OK Cancel

8. Click ok to accept the Generic Machine.

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

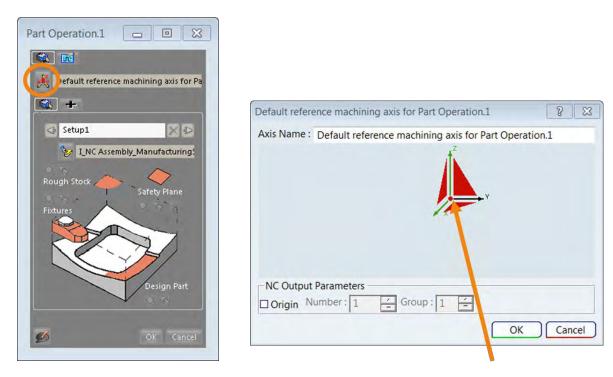
- a. Double click on the "Part Operation.1" activity.
- b. The "Part Operation" dialogue window opens.



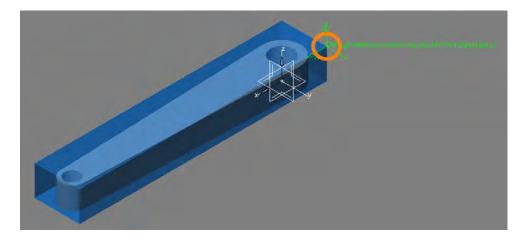
			Part Operation.1
Activities Process Tree	Computed		Default reference machining axis for Pa
- 🥐 Part Operation.1			Setup1
L I Manufacrogran	h.1	1	Rough Stock Safety Plane
(FF-27)		1910	Design Part
			Ø Ok Cantel

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.

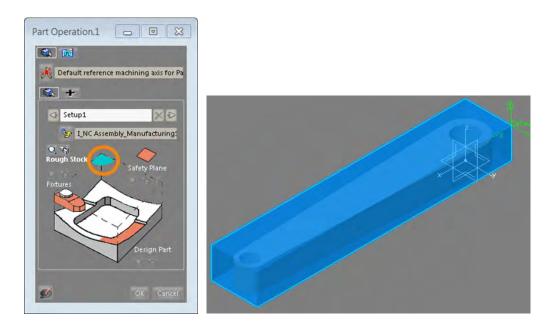






11. We will now define the 'Rough Stock'

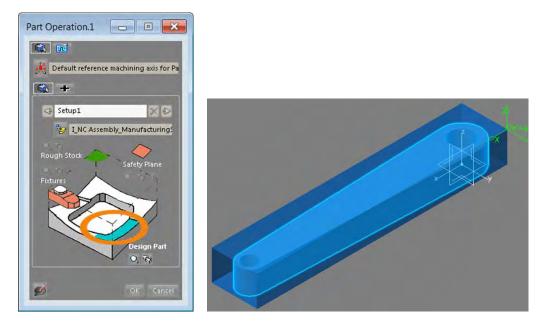
- a. Click on the "Rough Stock" icon
- b. Double click on the "PartBody" of the "Block" part as shown below.





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.

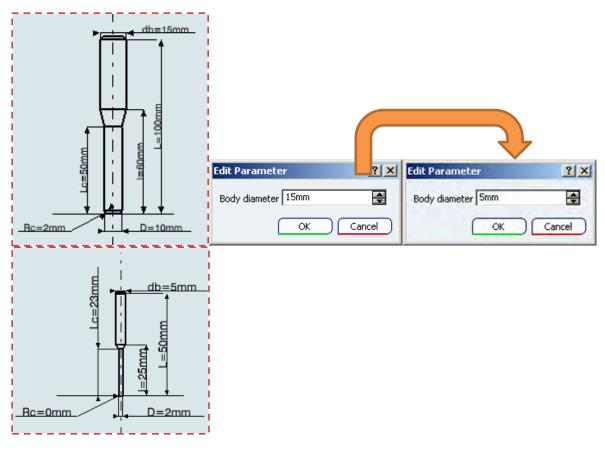


- 13. We will now face the top surface of the stock. We will start by the tool definition then we'll define the facing operation
 - a. Select "Insert" Menu > "Tool Builder" >
 - b. Then select "Mill Tool" > "End Mill Tool"

Insert Tools Window Help	Replace Resources	
Object <u>N</u> C Machine Control	Import/List Probing Tools	
Prismatic Machining Operations	Milling Tools	🕑 💈 <u>E</u> nd Mill Tool
Surface Machining Operations	Lathe Tools	Eace Mill Tool
Lathe Machining Operations	Drilling Tools Boring and Chamfering Tools	Conical Mill Tool
Utilities	Insert Tools Holder Tools	, <u>∐</u> <u>T</u> -Slotter Tool , <u>∏</u> <u>B</u> arrel Mill Tool
Tool Builder	<u>A</u> ssembly Tools	→ 📅 T <u>h</u> read 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.

Mil	lling Tool creat	ion	? ×
i r			
1	Name :		_
		Your_Name_T1_End Mill_D02	
	Comment :		
	Ball-end tool		
L i			

- 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.



Name :	Your_Name_T	1_Holder
Comment :		
Tool holder stage	5:2	A
11=10mm		d2=0mm D3=20mm d1=0mm D2=5mm
		OK Cancel

- 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

Tool Assembly creation	<u>? ×</u>
Name : Your_Name_T1_Ass	
Comment :	
Tool :	<u>a</u>
Gage : In	

- 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



ject	Selection		?
Fron	Search From Session		
Sele	ct Retrieve Loaded Data		
2 ite	ms found		**
#	Display name	Ne Version	Identifier
1	Your_Name_T1_End Mill_D02	Y2	T Your_Na
2	Your_Name_T1_Holder	Yer	T Your_Na
4			

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

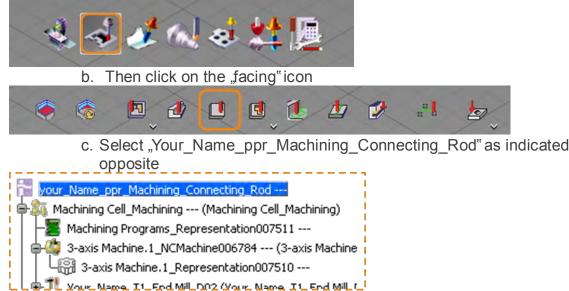
ool Assembly creation	? ×
Name : Your_Name_T1_Ass	
Comment :	
Tool : Your_Name_T1_End Mill_D02	2
Gage : 50	
Power : Fixed	
Holder :	

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



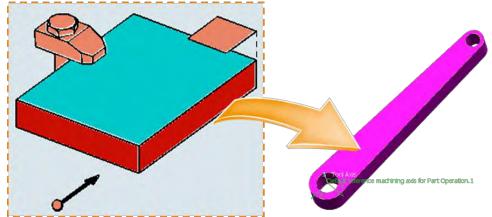
	n Search From Session	_		
	ems found			88 80 8
#	Display name	Ne	Version	Identifier
1	Your_Name_T1_End Mill_D02	Y2		T Your_Nar
2	Your_Name_T1_Holder	Yer		T Your_Nar

- 16. Hide the 'Block'
- We will now define the first facing operation
 a. Click on the "Prismatic Machining Operations" icon

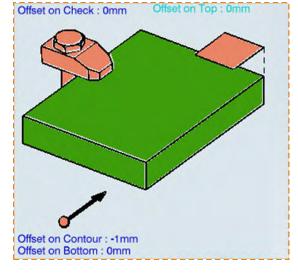


PLMCC

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



Double click on the value of the "Offset on the contour" and enter [-1mm] t on Check : 0mm Offset on Top : 0mm



e.



ame: Facing.5	ion	
	164	
Move the curso	r over a sensitive area.	
	T	
	110	
//	10	ATA
		7
		1-1.
	Z	
	X	0.5mm
		0.5mm
Tool path style:		O.Smn Om. Inward helical
Machining Radia		nward helical
		nward helical
Machining Radia	Axial Finishing	Inward helical
Machining Radia Direction of cut:	I Axial Finishing	Inward helical
Machining Radia Direction of cut: Machining tolerance:	I Axial Finishing Climb 0.1mm 0.1mm	Inward helical
Machining Radia Direction of cut: Machining tolerance: Fixture accuracy:	I Axial Finishing Climb 0.1mm 0.1mm	Inward helical

18. We will now define the 'StrategyShortHelp' tab

Machining	Radial	Axial	Finishing	HSM		
Mode:	Tool diamet	er ratio		•		
Distance bet	Distance between paths: 1mm					
Percentage	of tool diam	eter: 50		4		
End of path:			ſ	In 🔻		
Overhang:		10	0	-		
Tool side app	proach clear	ance: Om	m			
Machining	Radial	Axial	Finishing	ням		
Mode: Numl	per of levels			• ?		
Maximum de	pth of cut:	0.5mn	n	2 ?		
Number of le	evels:	4		?		
Machining	Radial	Axial	Finishing	ням		
Mode: No	finish pass			•		
Bottom finis	h thickness:	Omm		2?		
Machining	Radial	Axial	Finishing	HSM		
High Speed Milling						
Corner	Transition	1				
Corner rad	iust	1mm		₩?		
Limit angle	:	10deg		€ ?		
Extra segn	nent overlap	: 0.5mm		2		

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

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

Facing.5	<u>?</u> ×
Name: Facing.5	
Comment: No Description	
Assembly :	
Tool :	
	Search Tool Assembly
<u> 1</u> 0	Assembly Tool
	Vour_Name_T1_Ass I_Your_Name_T1_End Mill_D02.11
	1 Tool Assembly found
	Cancel

Facing.6		?×
Name:	Facing.6	
_	No Description	
tit		
Assembly	/: Your_Name_T1_Ass (Your_Name_T1_Ass)	Dalal
Tool :	Your_Name_T1_End Mil_D02 (I_Your_Name_T1_End	
Tool num	ber : 1	
3	11	
Commer	nt:	-
D Bal-	end tool	-
	Bc=0mm	(More>>)
	S. S.	
	OK Preview	Cancel

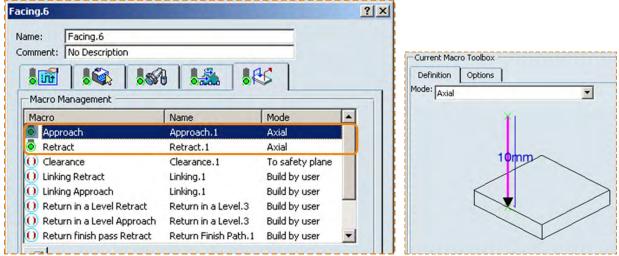
20. Complete this tab as indicated opposite

	1	34
Feedrate -		1
📃 Automatic c	ompute from tooling	Feeds and Speeds
Approach:	300mm_mn	÷
Machining:	1000mm_mn	
Retract:	1mm_mn	
Finishing:	0.1mm_mn	
Transition:	Machining	
	5000mm_mh	12
Unit:	Linear	•
Spindle Speed	1	
Automatic c	ompute from tooling	Feeds and Speeds
Spindle outp		
Machining: 70	iturn_mn	- E -
Unit:		



21. We will now define the 'Macro' tab

- 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"

Activities Process Tree Activities Part Operation.1	Cut Copy Baste	Ctrl+X Ctrl+C Ctrl+V		
Generation (Construction of the second		Del	Activities Process Tree Activities Activities Process Tree	Copy
	<u> </u>		Hanufacturing Program.1	Delete

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



Activities Process Tree	
Part Operation.1 Manufacturing Program.1 Tool Change.2nd Mill_D02.11)	
Your_Name_Facing_T1 ?	
	Your_Name_Facing_2_T1
	Your_Name_Facing_2_T1 Name: 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

Your_Name_Facing_2_T1		
Name: Your_Name_Facing_2_T1	Tool Axis	<u>? × </u>
Comment: No Description	Feature defined	-
	Components	
Move the cursor over a sensitive area.	O Angles	
	I: 0	
): 💽 🚅	
	K: 1	
	Reverse Direction	
	Display tool	
	Default position User-define	d position
0.5mm	ОК	Cancel

- 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



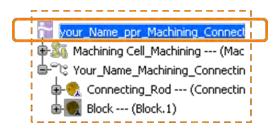
Your_Name_Facing_2_T1	
Name: Your_Name_Facing_2_T1	
Comment: No Description	
	3
Move the cursor over a sensitive area.	
Offset on Check : 0mm Offset on Top : 0mm	e maching assertor Part Operation.1
Offset on Bottom : 0mm	
Bounding envelope	

25. Now, we will define the profile contouring operation

a. Click on the "Profile Contouring" icon

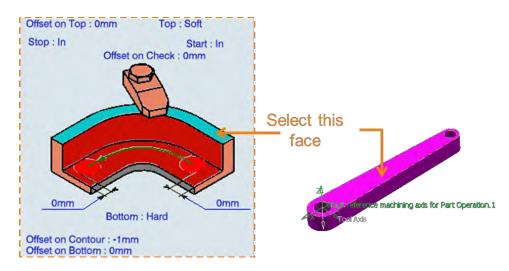


b. Then click on "Your_Name_ppr_Machining_Connecting_Rod"

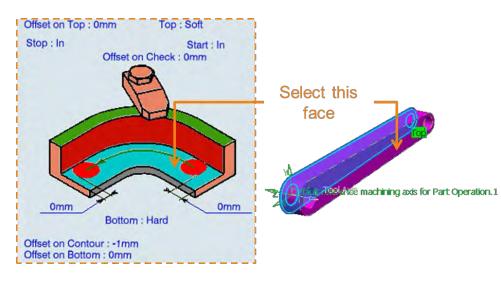


- c. Select the face as shown below
- d. Then select the top face of the connecting Rod





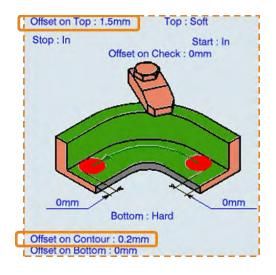
- e. Select the bottom face as shown
- f. Then select the bottom face of the connecting Rod





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

	escription		
Move the Machining	Stepover Finishing H	SM	
equencing:	Radial first		• ?
Radial Strate			_
istance betwe	een paths: 1mm 🚔 ?	Number of paths: 1	2 ?
overhang for i	rework areas:	50	-
Axial Strateg	y (Da)		
1ode:	Number of leve	ls	• ?
Maximum dept	h of cut: 0.563mm 🔁 Nun	nber of levels: 8	÷

- 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



illing Tool cr	eation		? ×
The tool num You may hav	ber 0 already exists. e unpredicable results	; in output file.	
Name : T	'our_Name_Drill_T2		
mm20=01			
		D=2mm	
			1ore >>) Cancel

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

Name :	Your_Name_Dri	ill_T2_Holder
Comment : Tool holder stage	es ; 2	•
<u> </u>		d2=0mm
12=10mm		D2=5mm



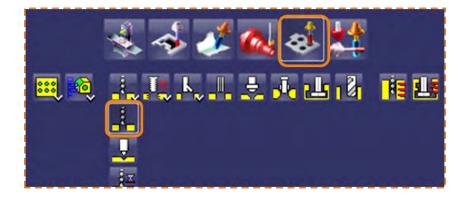
29. We will now assemble the 'T2'

- 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

Comment :	
Tool : Your_Name_Drill_T2	
Gage : 50	-
Power : Fixed	
Holder : Remove	
Your_Name_Drill_T2_Holder	

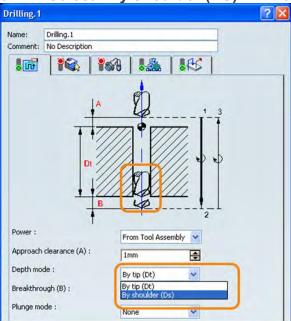
30. Now we will drill the smaller Hole.

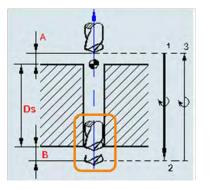
- e. Click on the "Axial Machining Operations" icon then click on the "Drilling" icon
- f. Click on "Your_Name_ppr_Machining_Connecting_Rod on the Specification Tree.





31. Select 'By shoulder (DS)' in 'Depth mode'.

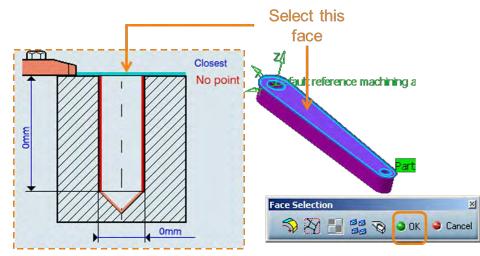




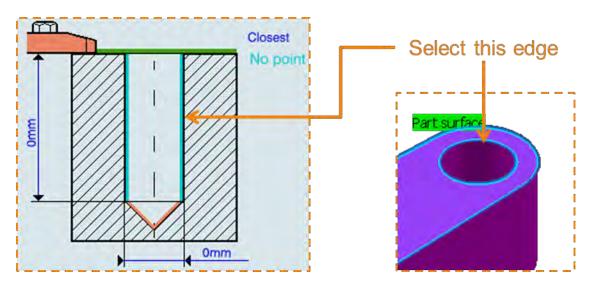
32. We will now define the drilling operation

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

PLATEG



- 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"

Drilling.1	X
Name: Drilling.1	
Comment: No Description	Search Tool Assembly
	Assembly Tool
Assembly : Your_Name_T1_Ass (Your_Name_T1_Ass)	Your_Name_Drill_T2 I_Your_Name_Drill_T2.7
Tool : Your_Name_T1_End Mill_D02 (I_Your_Name_T1_End)	1 Tool Assembly found
Tool number : 1	Cancel

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]

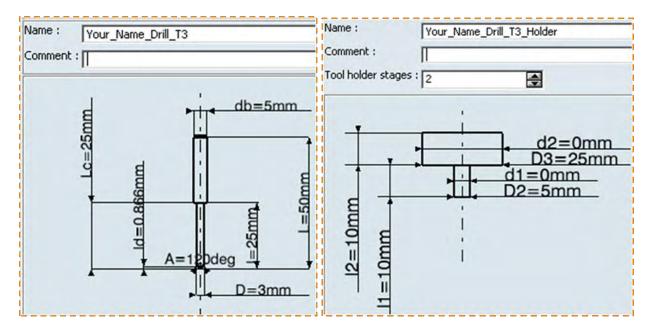


- d. Repeat the previous step for "Retract" macro
- e. Click "OK" to validate the drilling operation

Drilling.1 Name: Drilling.1 Comment: No Descript			
		8 9 AS	L II
Macro Management			
Macro	Name	Mode	10mm
Approach	Approach.1	Build by user	
Retract	Retract.1	Build by user	
		<u>/x</u> /x [

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]

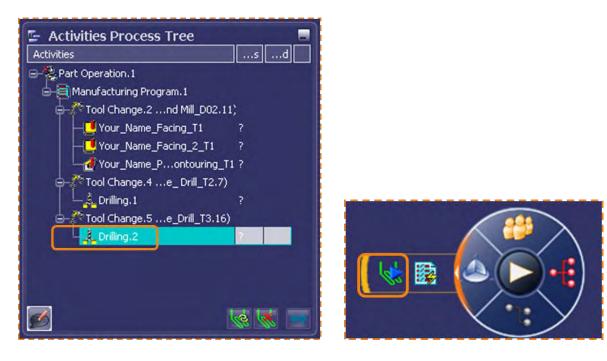


36. Now we will perform the last drilling operation

- 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

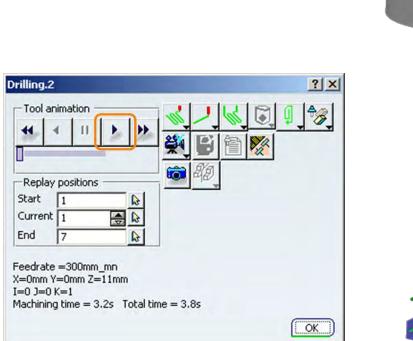
37. We will now generate a tool path for each machining operation

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



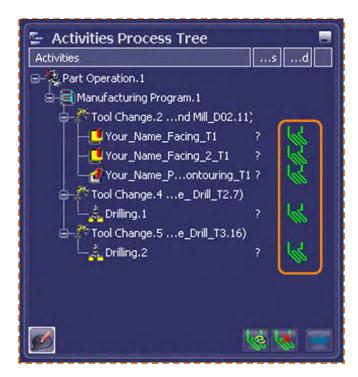


c. Click on the "Forward replay" icon





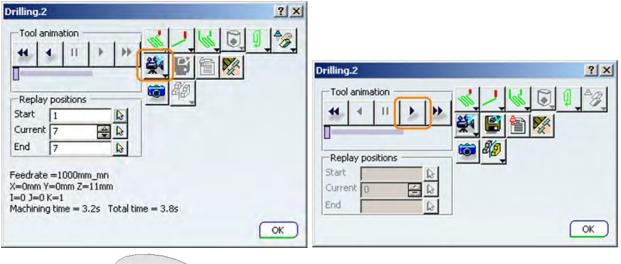
d. Repeat this operation for each operation





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

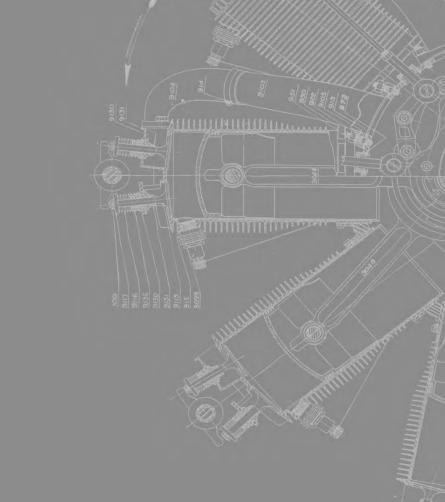


- b. Rename it as [Your_Name_CR_Manufacturing_Program_1.aptsource]
- c. Click on "Execute" to generate the program

enerate NC Output	Interactively	?
In/Out Tool mot	ions Formatting NC Code	
		_
Input	and have been	
Selection () Pa	rt Operations	
Pro	ograms	
Manufacturing	Program.1	
Resulting NC Data -		
💦 🞯 NC data t	vpe: Lee	1
	APT	
One file	○ for all selected programs	
	O by program	
	by machining operation	
Output NC Prov		
	_Manufacturing_Program_1.aptsource	
	-named NC Program	
	namos ne rrogram	
Machining Cell NC dat	ta generation	
🛐 📃 Lock operat	ions	
Execute)		_
Execute		
		Close









Drafting of Part



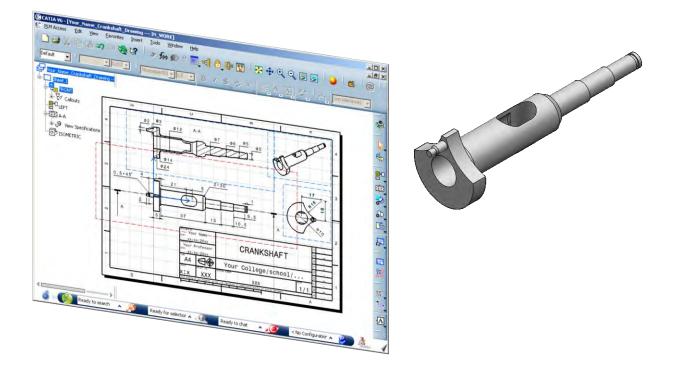


Drafting

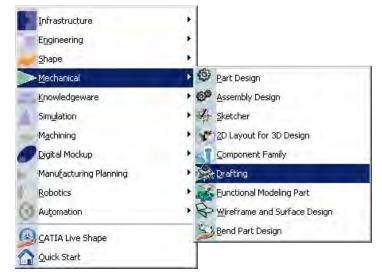
During these steps, you will generate the crankshaft drawing.

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"



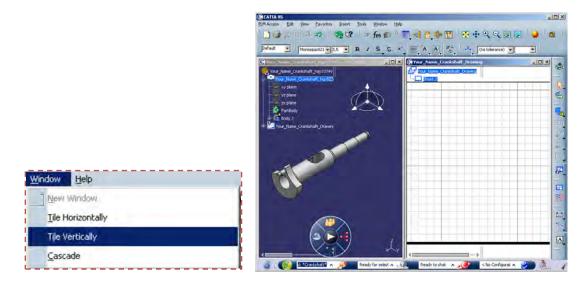


- 3. Under the **"Drawing**"tab, enter-"Your_Name_Crankshaft_Drawing" as "**Representation Name**"
- 4. Under the "Drawing Information" tab
 - a. Select "ISO" as "Standard"
 - b. Select "A4 ISO" as "Sheet Style"
 - c. Check "Landscape"
 - d. Select "Empty Sheet" as "Automatic View Creation Layout"

Drawing / Representation DS		
Drawing Drawing Information Standard ISO Sheet Style A4 ISO Format A4 ISO Paper size = 210 × 297 mm Global scale = 1:1 O Portrait O Automatic View Creation Layout		

5. We will separate the window into 2 parts

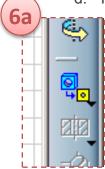
Select "Windows" Menu > "Tile Vertically

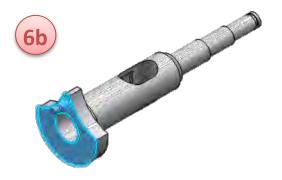


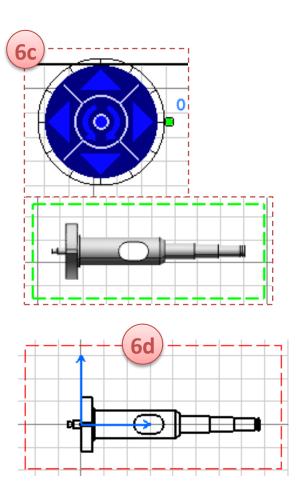


6. Create the front view

- 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



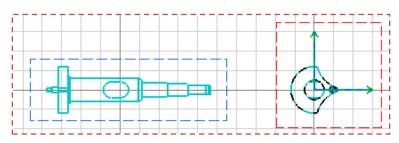






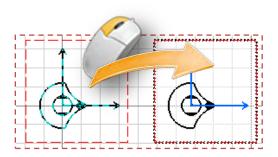
To activate a view:

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



To move a view:

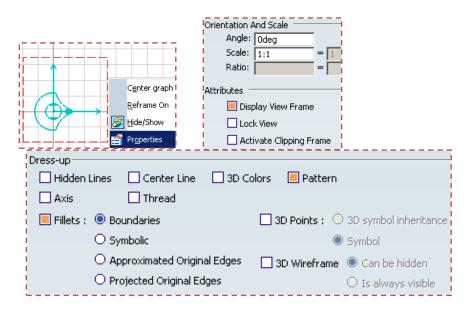
Drag and drop the frame of the view



To modify view properties:

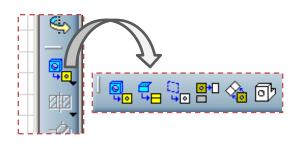
Right click on the frame of the view > "**Properties**"

You can modify the scale and the dress up

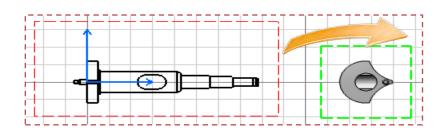




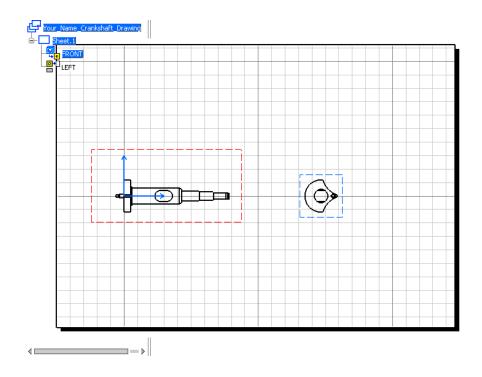
- 7. We will now create the left view
 - a. Click on the "Projection View" icon



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

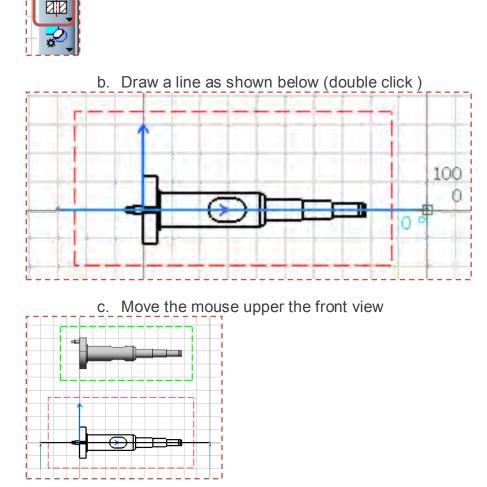


c. Click anywhere on the sheet to validate the view





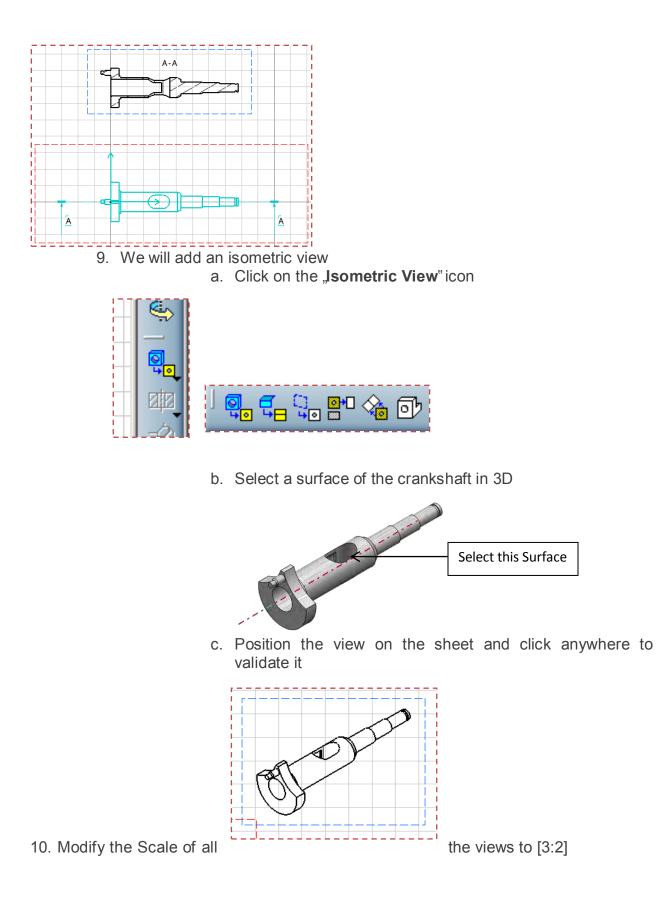
8. We will now create an offset section view



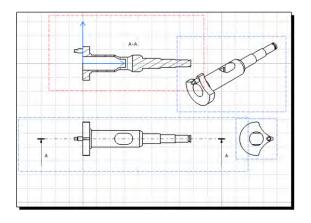
a. Click on the "Offset Section View" icon

d. Click anywhere on the sheet to validate the view

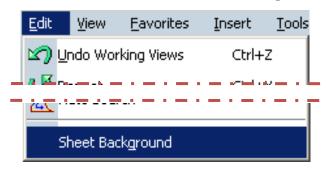








11. We will now insert a background to the drawing sheet a. Select "Edit" Menu > "Sheet background"



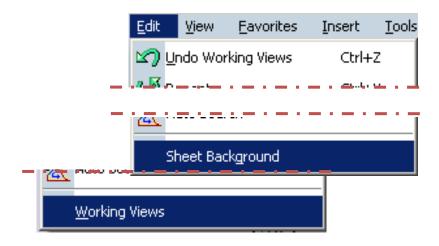
b. Click on the "Frame and Title Block" icon



- 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:

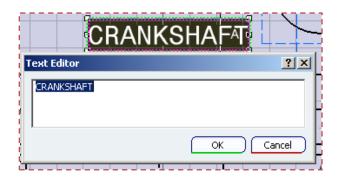


Select "Edit" Menu > "Sheet background" if you are in "Working view" mode

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

To edit the frame:

Double click on the text to modify it



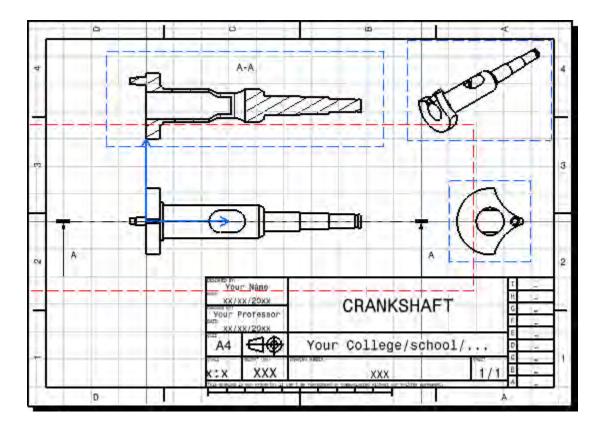


12. Fill the Frame as indicated:

DESIGNED BY: YOI	ur Name		19
XX/	xx/20xx	CRANKSHAF	T A
Your	Professor xx/20xx		•
A4		Your College/schoo	01/
SCALE	WEIGHT (kg)	DRAWING NUMBER	SHEET
x:x	XXX	XXX	1/1
This drawing	is our property; it	can't be reproduced or communicated without our written agreement.	

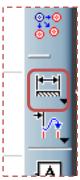
Adjust the view to the new frame

- 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.

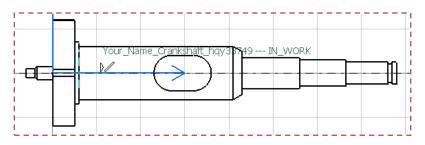


PLMCC

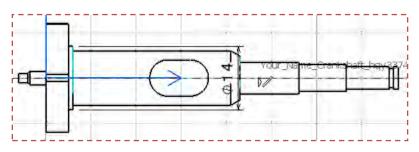
- 14. We will now add the dimensions to the front view
 - a. Activate the "FRONT" view if it has not already done
 - b. Click on the "Dimensions" icon



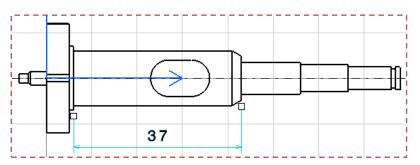
c. Select the first edge



d. Select the second edge



e. Click to validate the position of the dimension





Drafting of a part - Dimensions

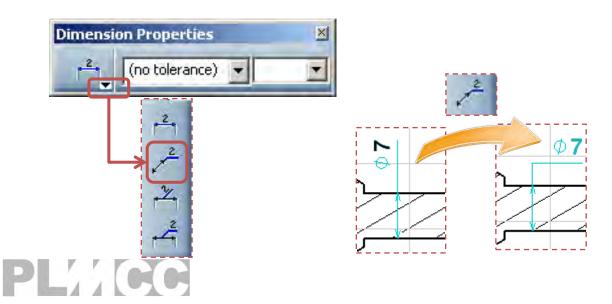
On the top of the window, you can find some tools to customize the dimensions

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

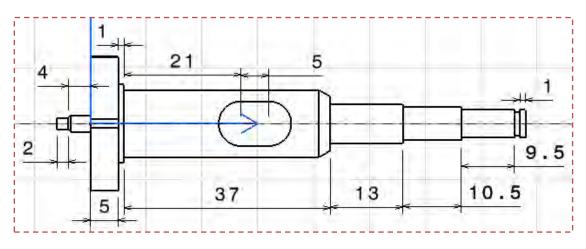
Text Properties
Monospac821 \bullet 3,5 \bullet B I S S X A A ϕ
Graphic Properties
Dimension Properties
(no tolerance)
(MATA You (Star A You A Strategy - P. Weit) C (B Marcos (2): Sym (Sevents Jonet Tools Weiter 1995 □ ゆう パ マ ぶ か い
+ Sociality - Bolton - Classical constraints -

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

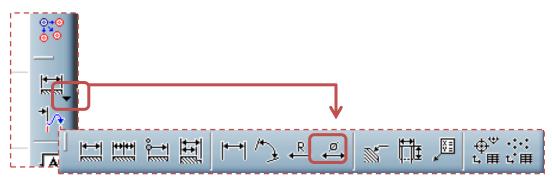
- Click on the "Dimension Line" icon
- Select a type of line



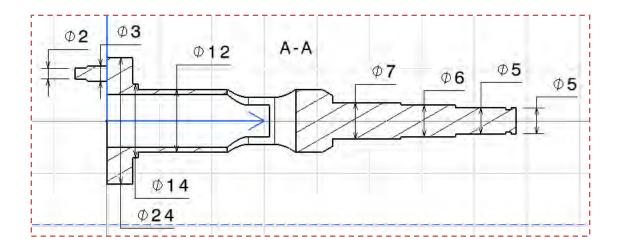
15. Repeat the previous steps to add the dimensions to the "Front" view as indicated opposite



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



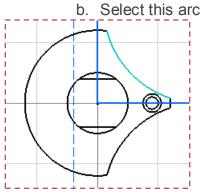
- b. Select an edge of a cylindrical surface
- c. Click to validate the position of the dimension



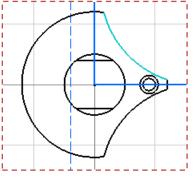


We will add the dimensions to the "LEFT" view

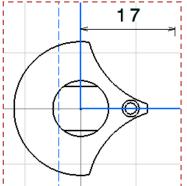
- 17. Horizontal dimensions
 - a. Click on the "**Dimensions**" icon





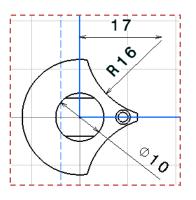


d. Click to validate the position of the dimension



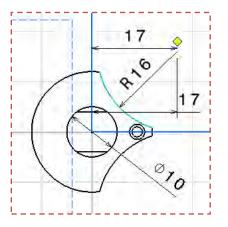


18. Add radius and diameter dimensions as shown opposite



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

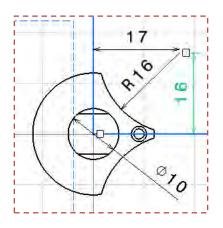


e. Select "Dimension Representation" > "Force Vertical Dimension in view"

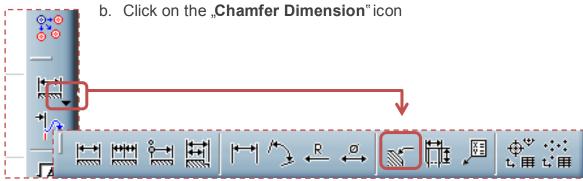
🔽 Di	istance		
Ha	alf Dimension		
Dį	imension Representation 💦 🕨 🕨	6	Projected Dimension
E	xtension Lines Anchor	Ś	Eorce Dimension on Element
-	dd Funnel		Force Horizontal Dimension in view
		<u> </u>	Force Vertical Dimension in view
⊻a	alue Orientation	Ŕ	Force Dimension along Direction
<u>R</u> e	estore Value Position	Ż	True Length Dimension
I <u>n</u>	tersection point detection		-



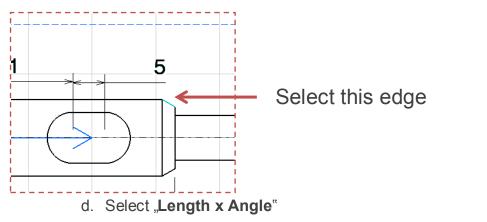
f. Click to validate the position of the dimension



- 20. To finish we will add the dimensions to the 2 chamfer on the front view
 - a. Activate the "FRONT" view



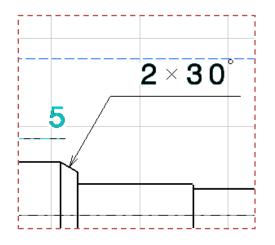
c. Select the edge of the chamfer







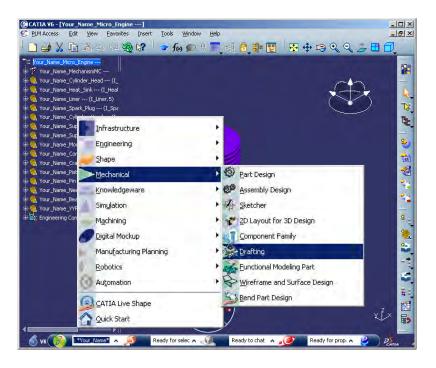
e. Click to validate the position of the dimension



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**"

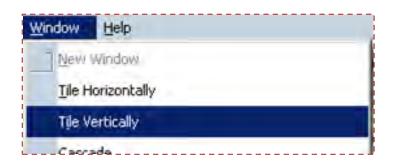
D	rawing /	Physical Representation
	🔒 Dra	wing 🔒 Drawing Information
	Name	Your_Name_Micro_Engine_Drawing
	Identifier	drw008511

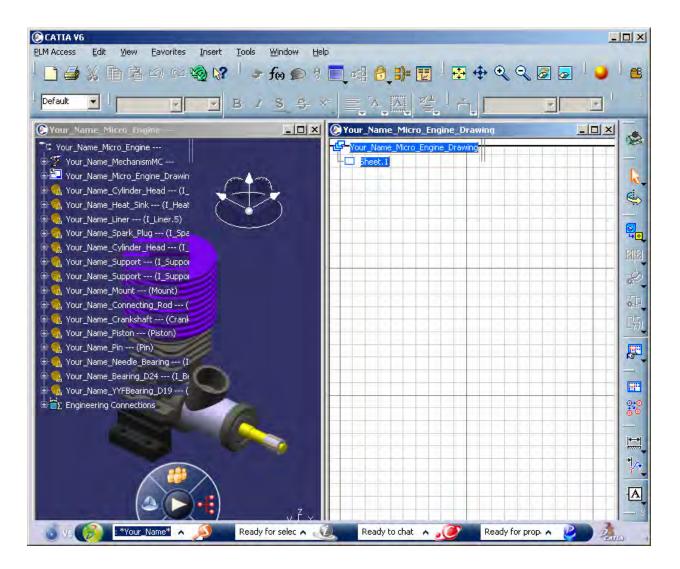
- 4. Under the "Drawing Information" tab
 - a. Select "ISO" as "Standard"
 - b. Select "A3 ISO" as "Sheet Style"
 - c. Check "Landscape"
 - d. Select "Empty Sheet" as "Automatic View Creation Layout"
 - e. Click "Finish" to validate the drawing informations

Drawing	Drawing Information
Standard	
ISO	
Sheet Style	
A3 ISO Format A3 ISO	
Global scale = 1:1	
A	O Portrait O Landscape
Automatic View Cr	Landscape
Automatic View Cr	Landscape



- 5. We will separate the window into 2 parts
 - a. Select "Windows" Menu > "Tile Vertically"

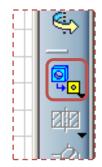




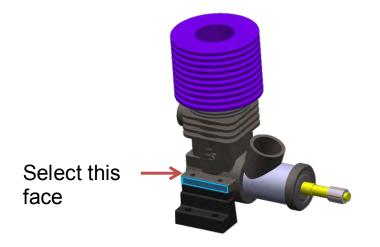
PLMCC

Create the front view

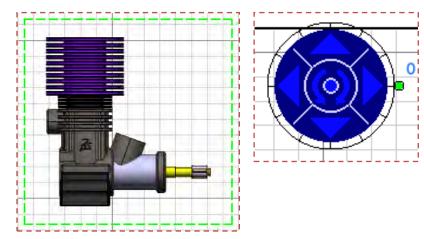
 Click on the "Front view"icon



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

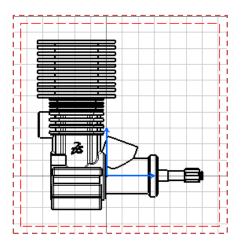


c. use the arrows to orientate the view as indicated opposite, if you need to.



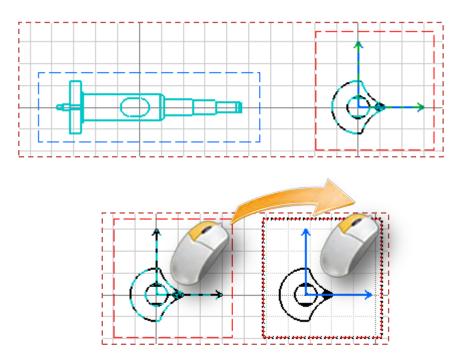


d. Then click anywhere on the sheet to validate the view



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



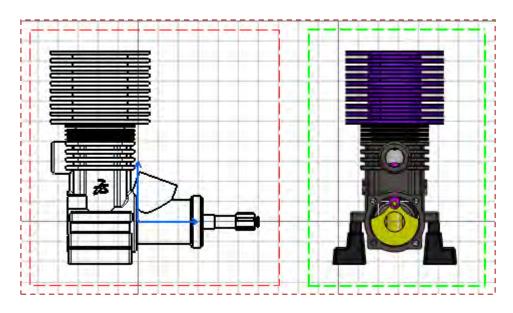
To modify the view properties

Right click on the frame of the view > "**Properties**" You can modify the scale, the dress up



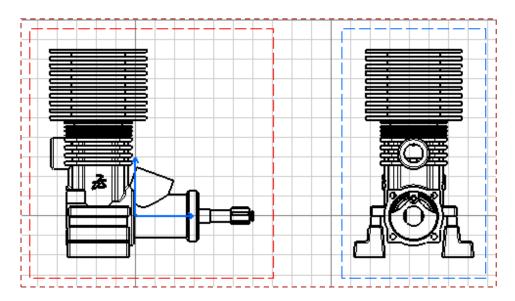
	Orientation And Scale
	Angle: Odeg
	Scale: 1:1 = 1
	Ratio:
Center gra	Attributes
Reframe O	n 📃 Display View Frame
Hide/Show	Lock View
🖵 💶 💶 💶 💶 🛃 Properties	Activate Clipping Frame
Dress-up	
🗌 Hidden Lines 🗌 Center Line 🛛	3D Colors 📃 Pattern
7 Axis Date Thread	
Boundaries	□ 3D Points : ○ 3D symbol inheritance
- C Symbolic	Symbol
	Edges 🔄 3D Wireframe 🔘 Can be hidden
O Projected Origina	📕 🛄 📴 🖓 🕞 ^{ways visible}

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

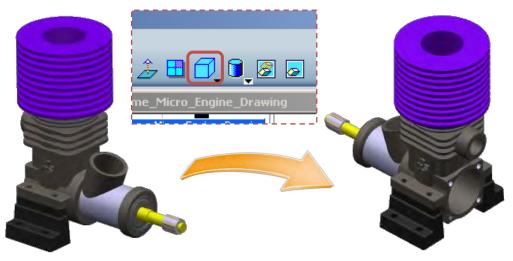


c. Click anywhere on the sheet to validate the view





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

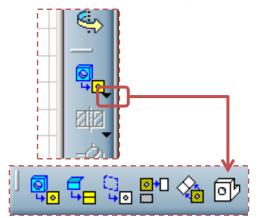


- b. Click on the bassement of the robot
- c. Position the Micro Engine as indicated opposite

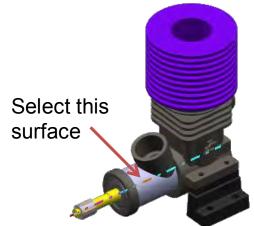


10. We will add an isometric view

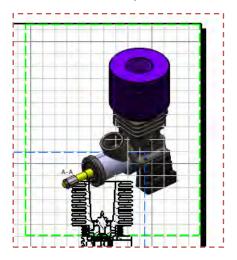
a. Click on the "Isometric View"icon



b. Select the surface in 3D as indicated opposite

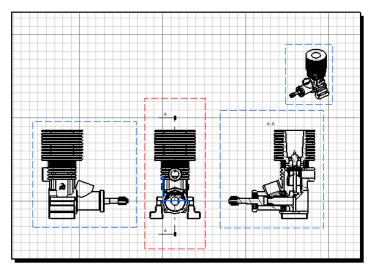


c. Position the view on the sheet and click anywhere to validate it

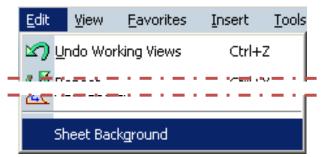




11. Modify the Scale of all the views to [1:2]



- 12. We will now insert a background to the drawing sheet
 - a. Select "Edit" Menu > "Sheet background"



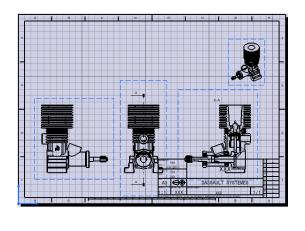
b. Click on the "Frame and Title Block" icon



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



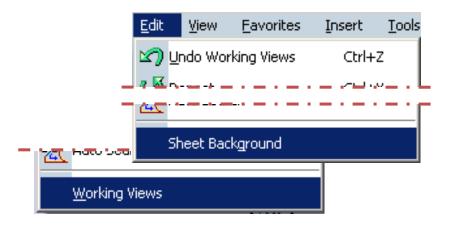
d. Select "Create" as "Action"



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Drawing Titleblock Sample 1	A CALL ST ALL ST
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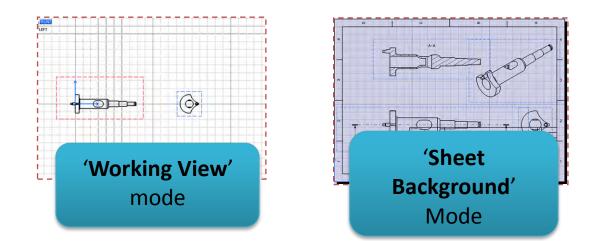
To activate the sheet background mode or to work on views*

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





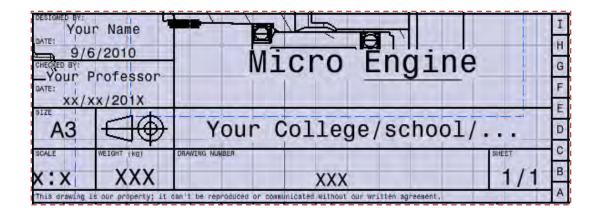
Select "Edit" Menu > "Working view" if you are in "Sheet background" mode



To edit the frame

Double click on the text to modify it

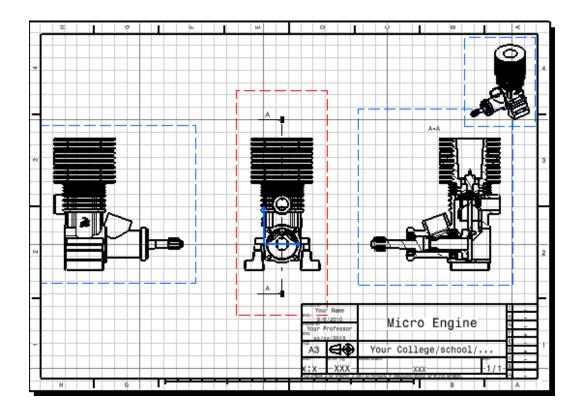
13. Fill the Frame as indicated opposite



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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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Exercises



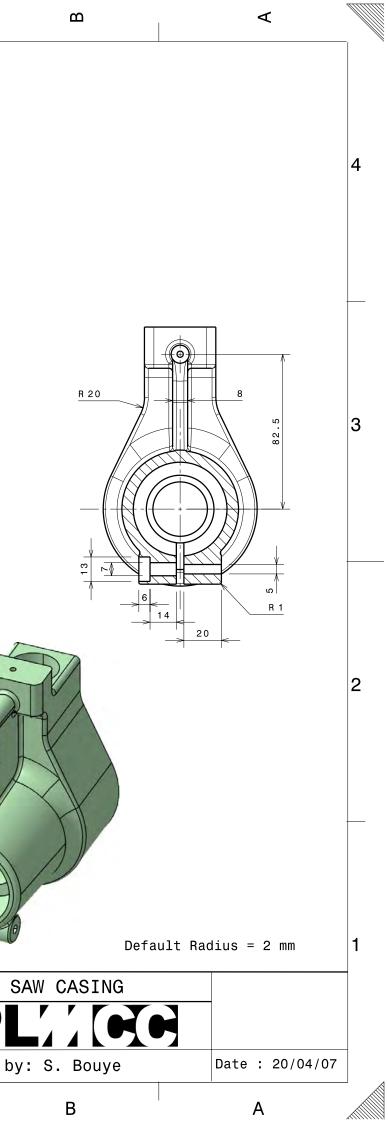




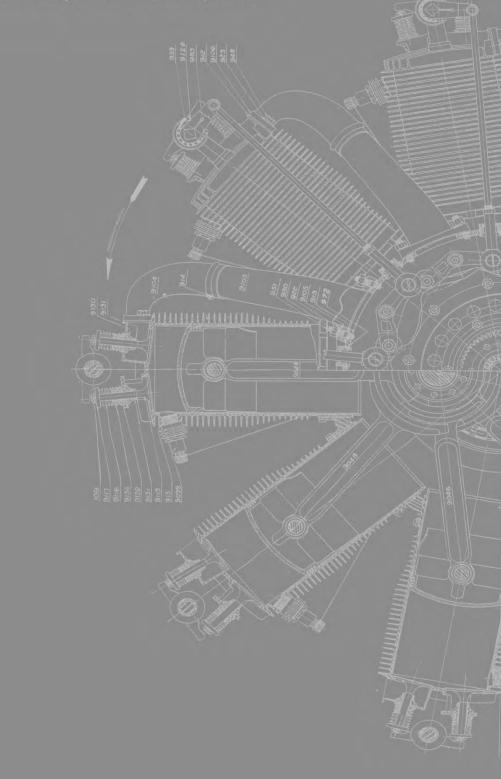
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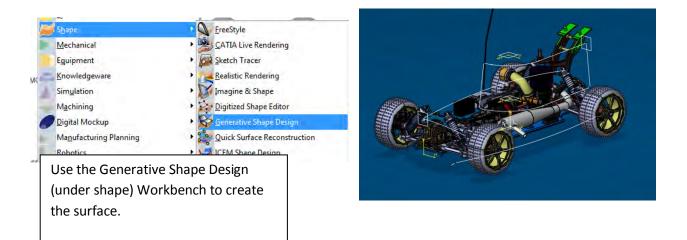


Basic Surface Design

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

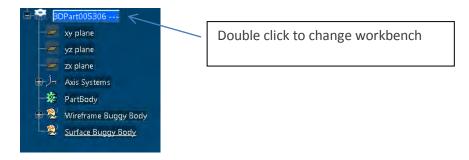
Click PLM Access > Import > 3D XML.... You will find it in the Buggy_3Dxml> Starting_Data>surface folder. Select the file named: -your-Name-buggy.3dxml Check Import As New. As duplication string enter *"your name_*' Click OK.



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.





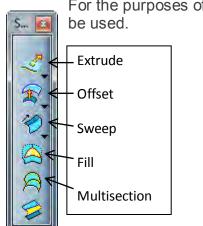
Notice that the geometrical set "Surface Buggy Body" is active, so any creation, be it surfaces or sketches, will take place under the Surface geometrical set.

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.

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	Constraints	•
rename baggy boay	Annotations	
Surface Buggy Body	Analysis	

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.

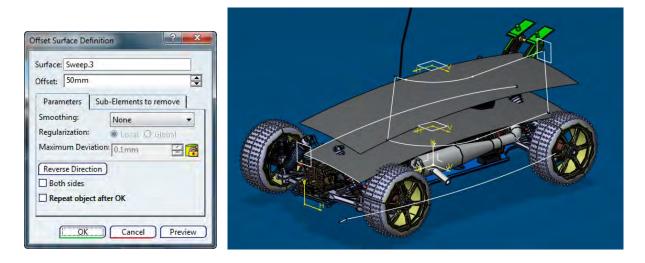
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	Subtype: With reference surface	
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D Position	profile Show parameters >>)	

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



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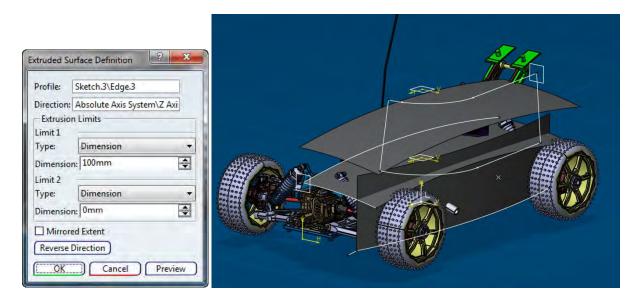
Using the Offset tool 🔊 select the swept surface to create a new surface offset 50mm above the original surface.



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

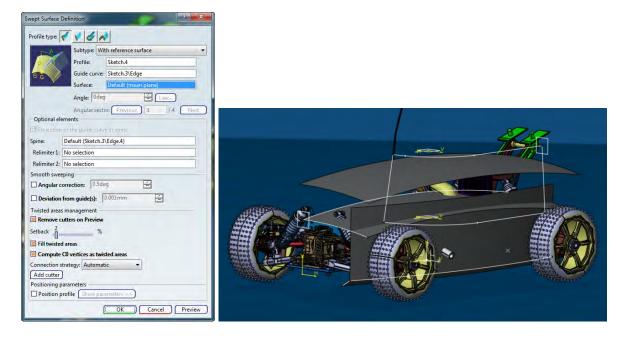


The Extrude tool is very helpful to create simple surfaces by only using only one element. Direction can be added by clicking on the desired arrow in the reference axis, or by using a previously created item to indicate direction.



Sketch.3

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.

Iulti-Sections Surface Definition	
No. Section Supports Continuity Closing Point 1 Sketch.6 2 Sketch.5	
Guides Spine Coupling Relimitation Canoni, No. Guide Supports Continuity 1 Line.2 2 Line.1	
Replace Remove Add Continuity: Tangent Smooth parameters Angular correction: 0.5deg	
Deviation: 0.001mm	



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

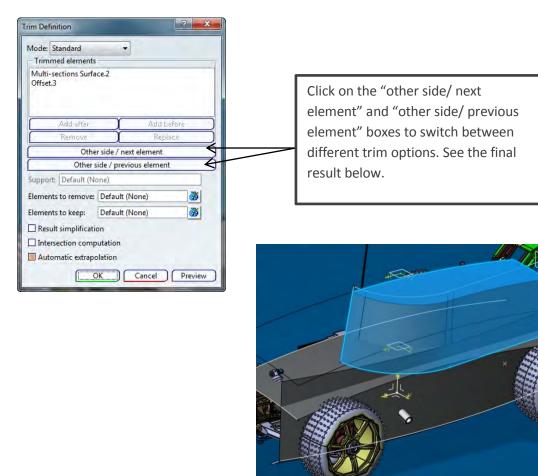
Trimming, splitting and joining

All the created surfaces are still separate entities and to create a coherent single surface all the surfaces must be trimmed for 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.

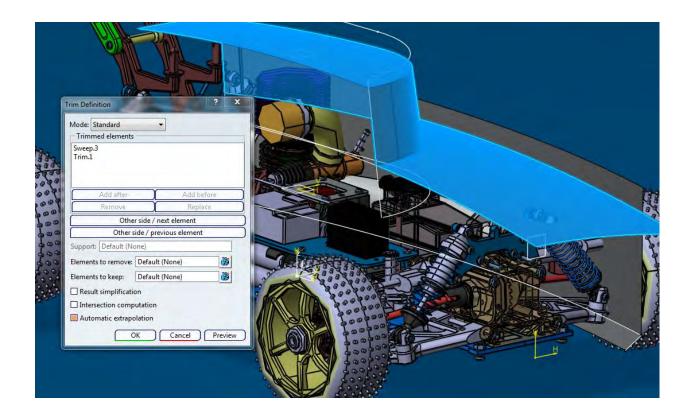
The first surfaces to trim are that of the cabin roof and side.

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)

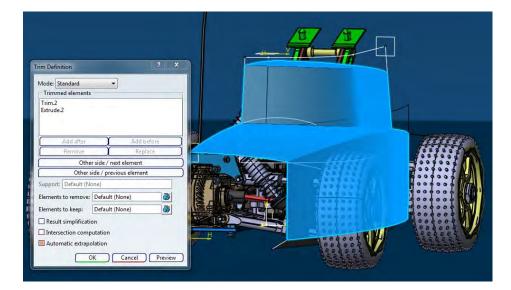


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In a similar fashion, trim the cabin and the first sweep you created. Notice that the cabin trim is now one element. As mentioned earlier, the trim function automatically joins the result.



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.

Image: Second state of the second s	Join Definition		? ×
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Check tangency Check connexity Check manifold Simplify the result Ignore erroneous elements Merging distance 0.001mm	12		
NUGOBO NI	Check tangence Simplify the res I Ignore erroneou Merging distance	y Check sult us elements	connexity Check manifold

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 varia	able 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.



) Edge() to filet
	Variable Radius Fillet D Support: Extremities: Radius: Edge(s) to fillet:	Join.1 Smooth	Edge(s) to keep: No selection
You can edit the Radius by double clicking on it (Same for the back radius)	Selection mode: Variation Variation: Conic parameter: Trim ribbons Trim support	Tangency 2 elements Cubic	Limiting element(s): No selection

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.*)

	Edge Fillet Definition Support: Extremities:	EdgeFillet.1	Edge(s) to keep:	No selection		
	Radius: Object(s) to fillet:	12mm	Limiting element			
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	Trim ribbons Trim support	1.0			00	17
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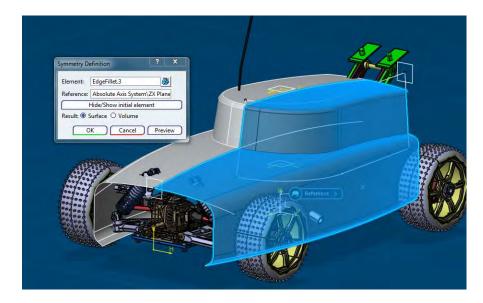
Using the edge fillet again, create a fillet with a radius of 5mm on the top of the canopy.

Edge Fillet Definition		© Object(s) to fillet »
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	<u> </u>	OK Cancel Preview

Symmetry and Fill

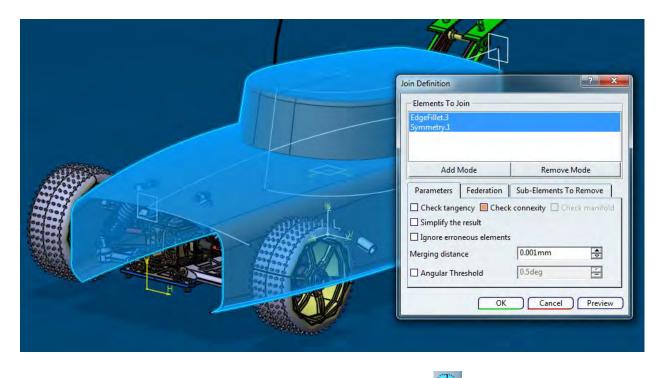
Now that the main body has been made, you can mirror it by using the symmetry tool.

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





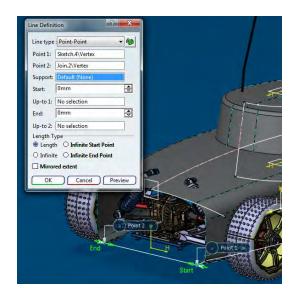
When the symmetry is completed you will notice that the two pieces are still separate. Use the join tool to join the two surfaces.



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



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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.

	Boundary Definiti	īon	? X	
	Propagation typ Surface edge: Limit1: Limit2:			Surfac
C Limit2		OK Cancel	Preview	

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.

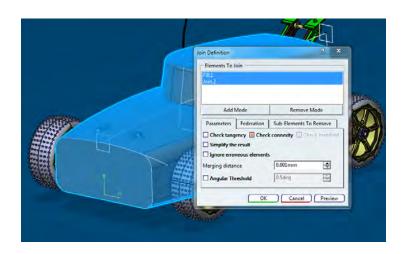
After the line and boundary creation, return to the surface geometrical set and make sure it is defined as the in work object.

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.

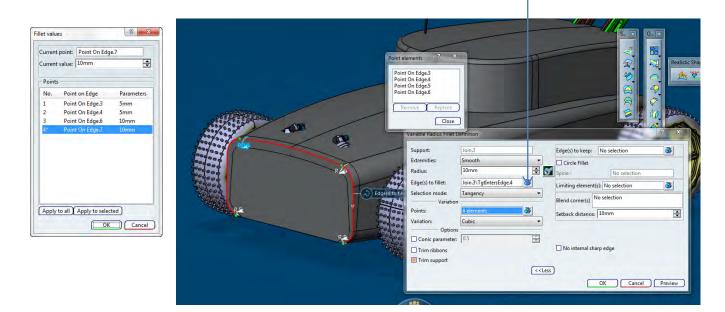
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You need to join the newly created nose surface to the main surface. Use the Join tool



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

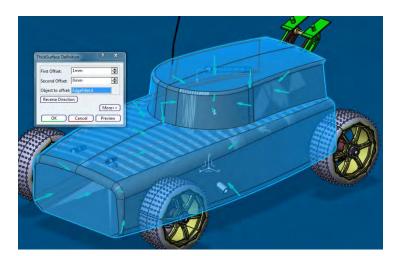
The next step will be to thicken the surface so that it actually becomes a part.

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



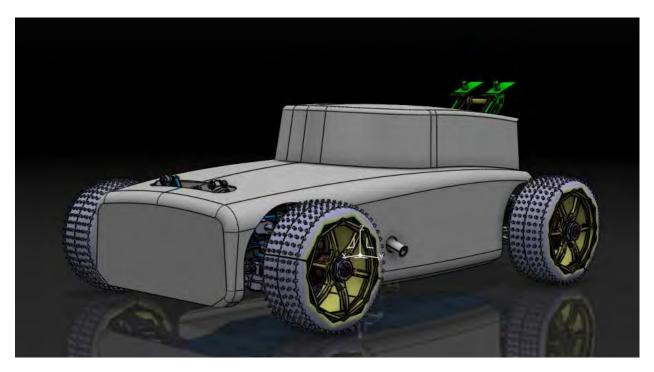
Once in the Part environment you need to convert the surfaces into a part. This can be

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







Exercises

