EXERCISE BOOK

Student Notes:

CATIA V5 Training
Exercises

2D Layout for 3D Design

Version 5 Release 19
August 2008

EDU_CAT_EN_LO1_FX_V5R19

Student Notes:

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EXERCISE BOOK

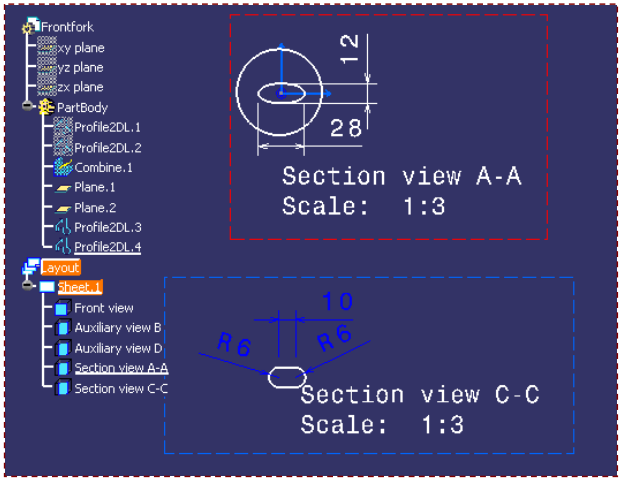
Student Notes:

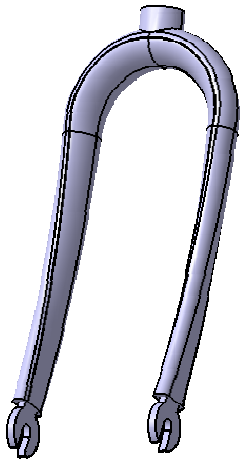
Front Fork

Recap Exercise

40 min.

In this exercise you will create the 2D geometry for the front fork.



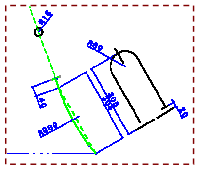


Student Notes:

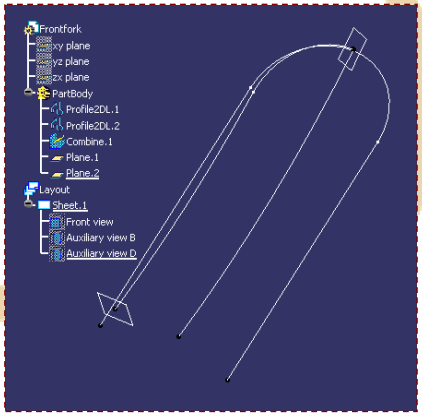
Do It Yourself

Open the "Front_fork_step0.CATPart"

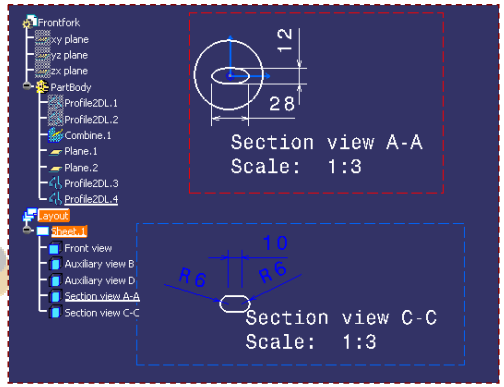
1 Open Front_fork_step0.CATPart



2 Extract the 3D profiles from the layout the front and the auxiliary view of the front. Combine these 2 profiles



3 Create two planes normal to this combine curve at its extremities. Create section views on these Planes. Define the geometry of these section views.



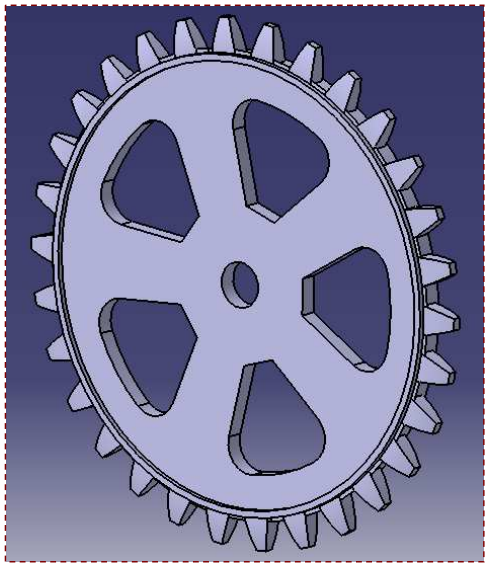
Student Notes:

Front Rear Sprocket

Recap Exercise

20 min

In this exercise you will create the following 3D part from a 2D layout

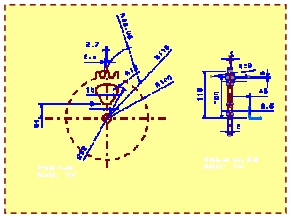


Student Notes:

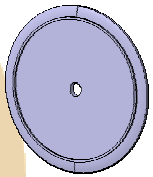
Design Process: Front Rear Sprocket Creation

You will be creating a Front Rear Sprocket using the following Design Process.

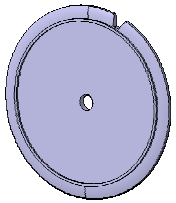
1. Open the CATPart.



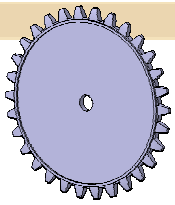
2. Extract the 3D profile (automatic propagation, no support) to create the shaft defining the sprocket



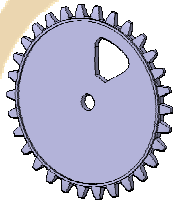
3. Extract the 3D profile (explicit definition on a plane defined by a planar surface of the shaft) to create the first tooth with a pocket



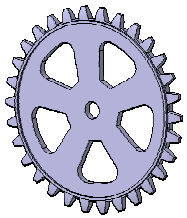
4. Create the other teeth using pattern capability



5. Create the 3D profile (explicit definition on a plane defined by a planar surface of the shaft) to create the sprocket pocket



6. Create the other pockets with pattern capability

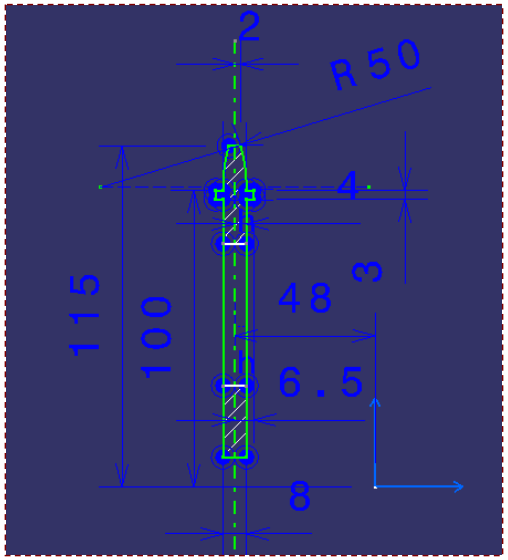


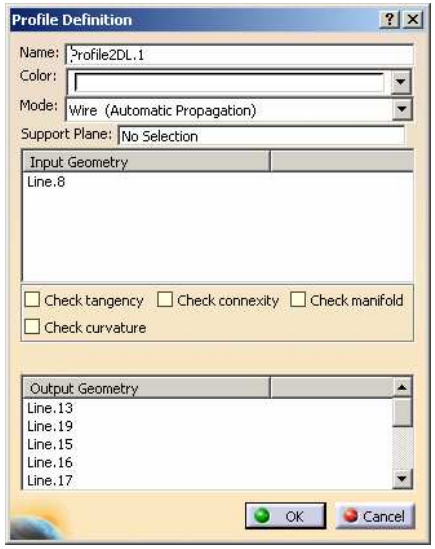
Student Notes:

Do It Yourself (1/3)

Open the "FrontRearSprocket_step1.CATPart"

- Extract the 3D profile of the sprocket shaft from the provided section view.
- Using the 3D Planes functionality, create two 3D planes with the help of the axes in the front view.
- Create an intersection of the two 3D planes.
- Create the shaft using the intersection as the shaft axis.

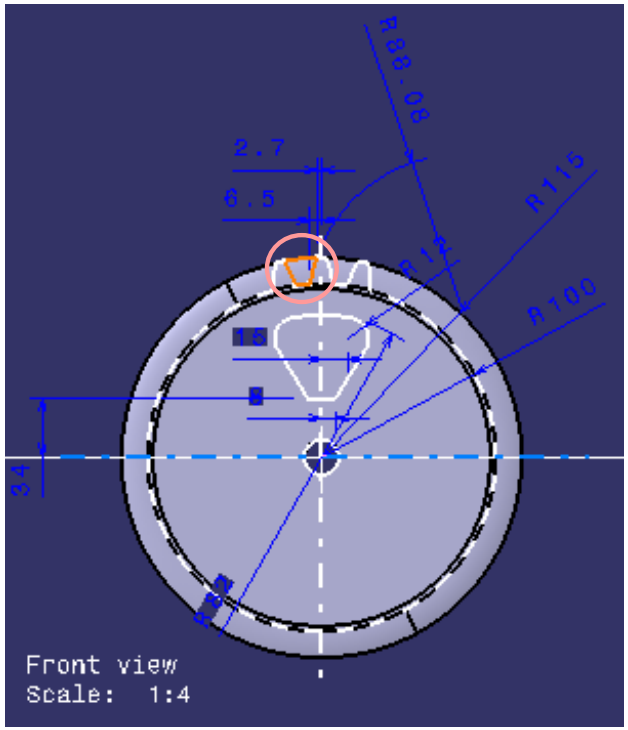


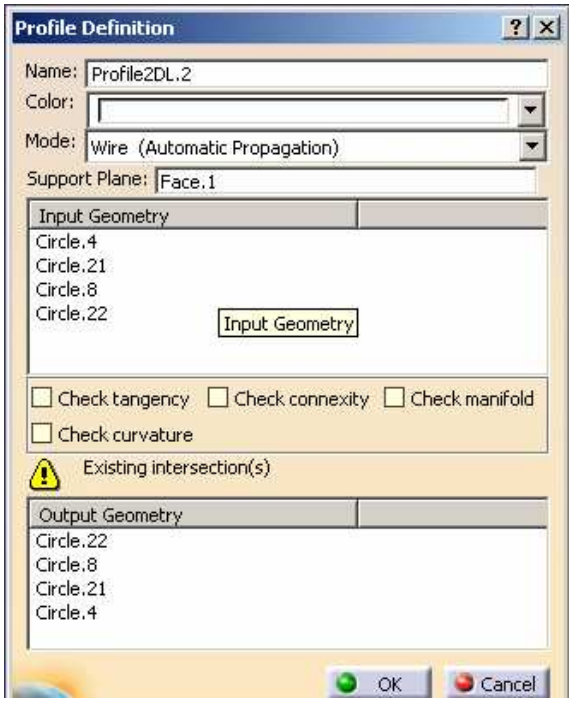


Student Notes:

Do It Yourself (2/3)

- Extract 3D profile on a planar face of the tooth.
- Create a Pocket using the created 3D profile.
- Pattern 30 instances of the Pocket around the complete crown. This will create 30 teeth on the Sprocket.

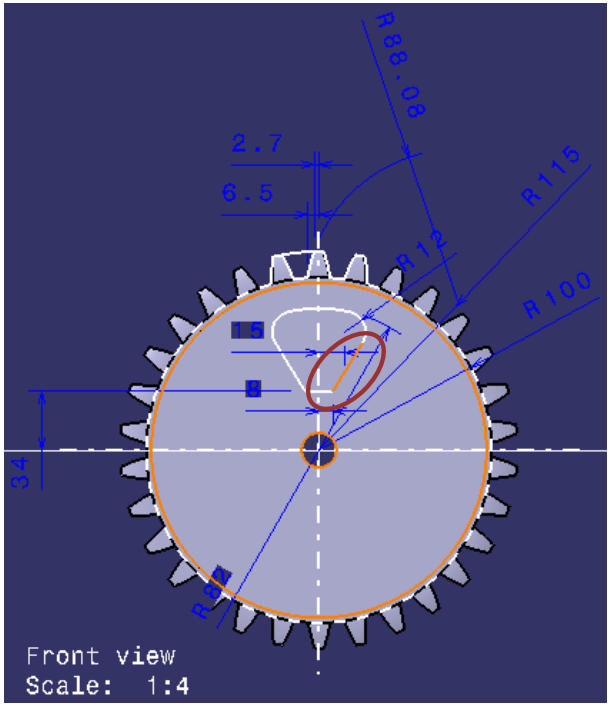


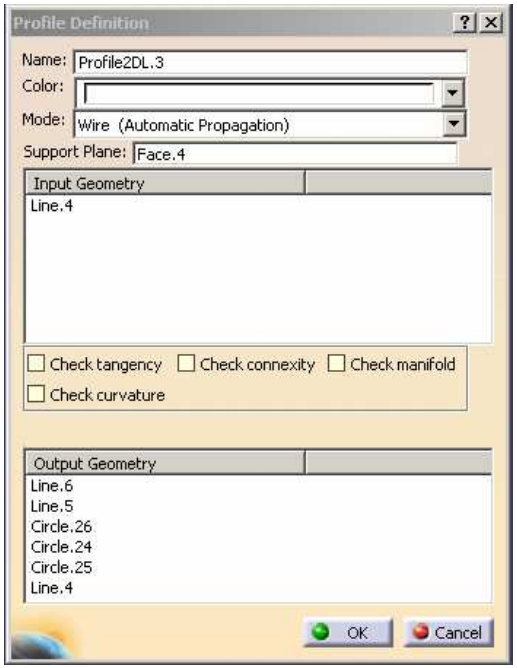


Student Notes:

Do It Yourself (3/3)

- Extract 3D profile on a planar face of the sprocket.
- Create a Pocket using the created profile.
- Pattern 4 instances of the Pocket around the complete crown.





You can compare your result with the end model "FrontRearSprocket_end.CATPart"

EXERCISE BOOK

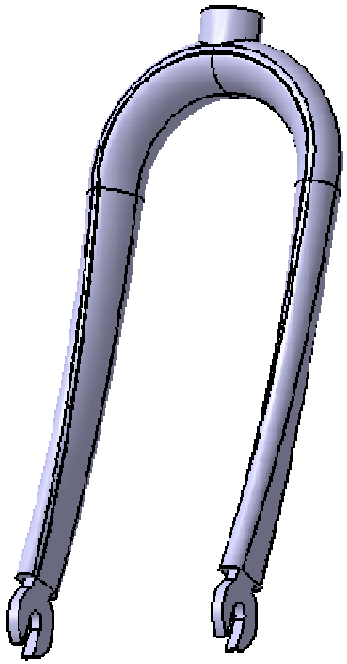
Student Notes:

Front Fork 3D Design

Recap Exercise

20 min.

In this step you will create the following 3D part from a 2D layout

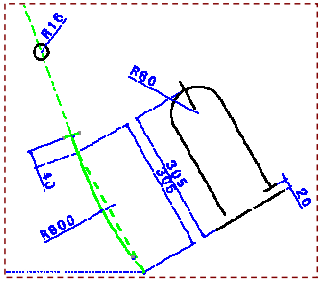


Student Notes:

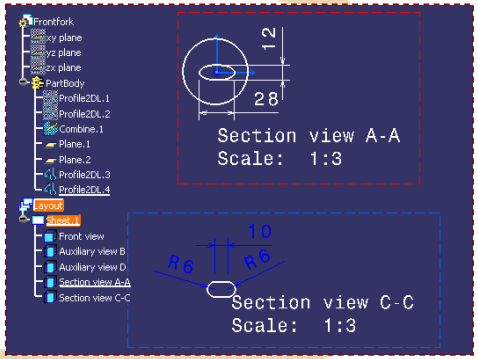
Do It Yourself

Open the "Front_fork_step0.CATPart"

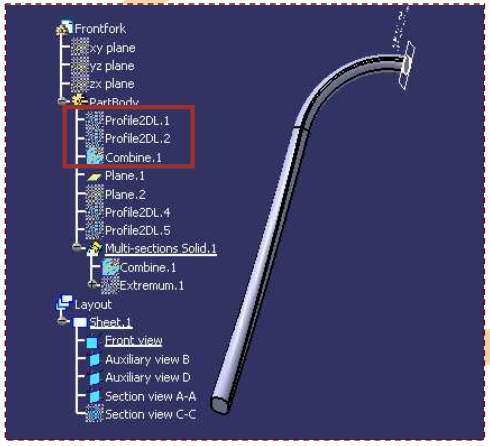
1 Open the Front_fork_step0.CATPart



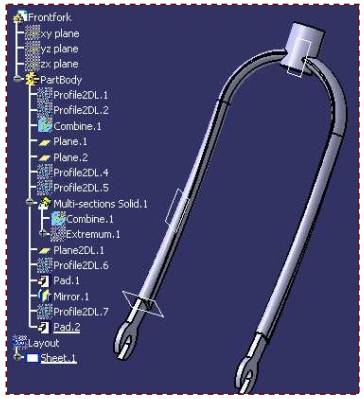
2 Create the Section View A-A and Section View C-C as shown.



3 Create 3D profiles of Section views A-A and C-C. Create the shape of a fork with these 3D profiles and their combine curve using the Multi-sections Solid.



4 Complete the fork design using the Front view and the Auxiliary view D



You can compare your result with "Front_fork_step5.CATPart"

EXERCISE BOOK

Student Notes:

Engine Assembly

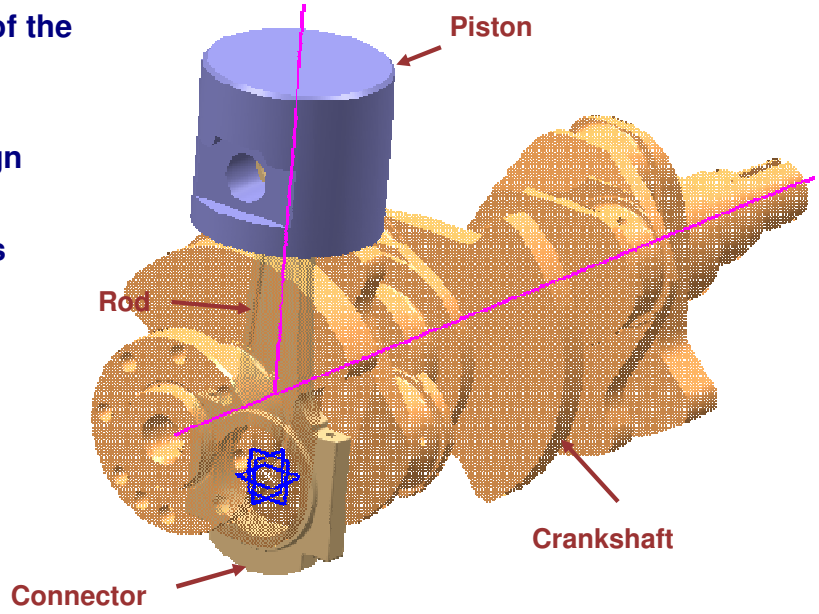
Master Exercise Presentation

85 min

In this exercise you will design the components of the Piston Assembly. The piston and the connector are to be designed in the context of the Piston Assembly.

You will design these components using the following steps in the 2D Layout for 3D Design workbench:

- **Defining 2D profiles using 2D Layout views**
- **Creating 3D elements by exporting the required 2D profiles**
- **Completing the 3D design**



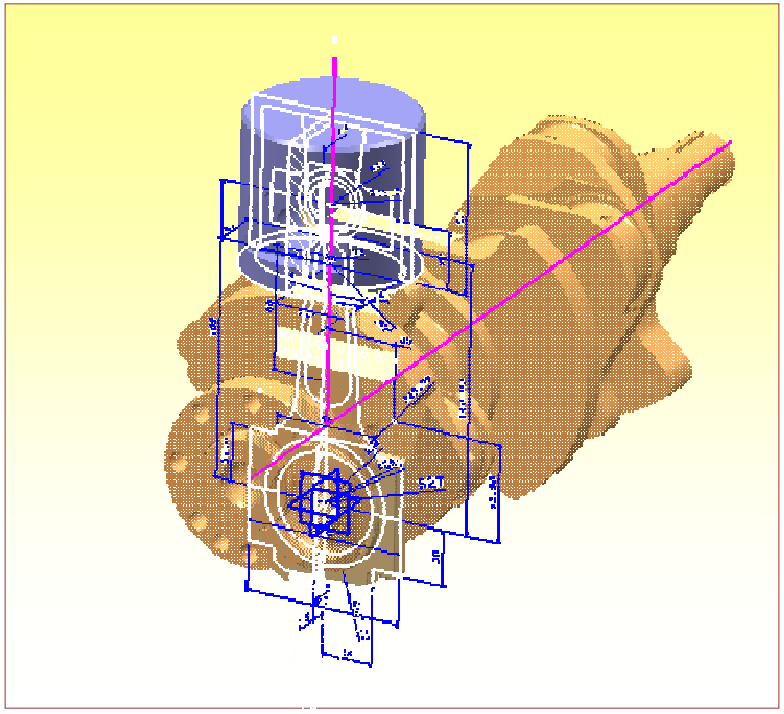
Engine Assembly

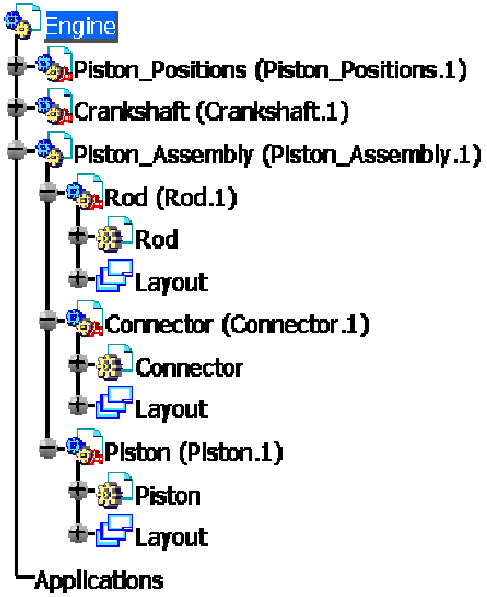
EXERCISE BOOK

Student Notes:

Engine Assembly – Design Intent

In this exercise, you will design the components of the Piston_Assembly in a 2D environment. You will create 2D layout views of the Connector and Piston.



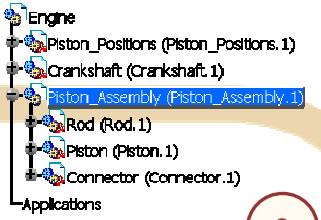


EXERCISE BOOK

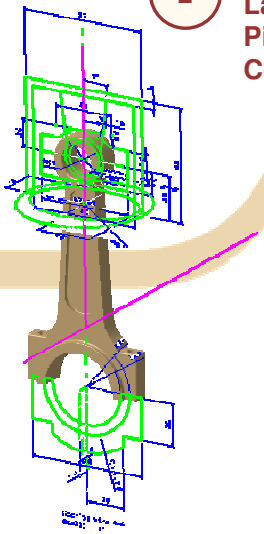
Student Notes:

Engine Assembly - Design Process

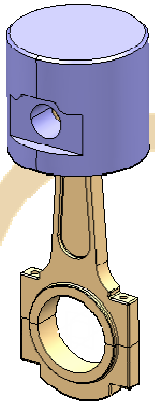
1 Edit Product Piston_Assembly and insert new parts: Piston and Connector



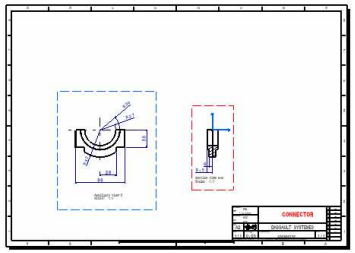
2 Design 2D Layout views for Piston and Connector



3 Complete 3D Design of Piston and connector



4 Create Drawing from 2D Layouts



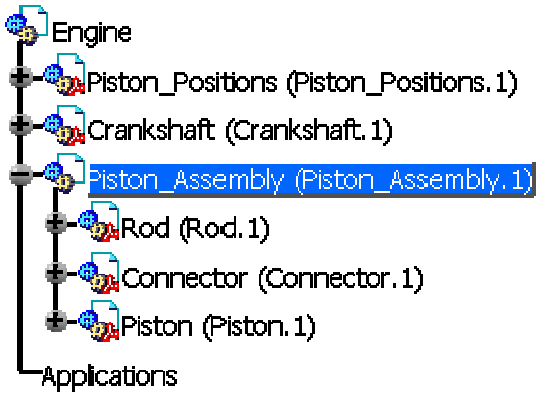
Student Notes:

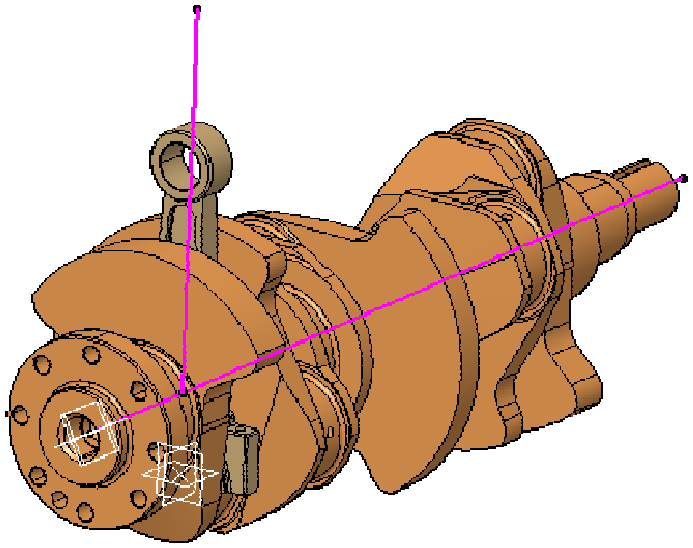
Engine Assembly

Step 1: Create New Parts

5 min

In this step you will create new parts: Piston and Connector in the Piston Assembly.





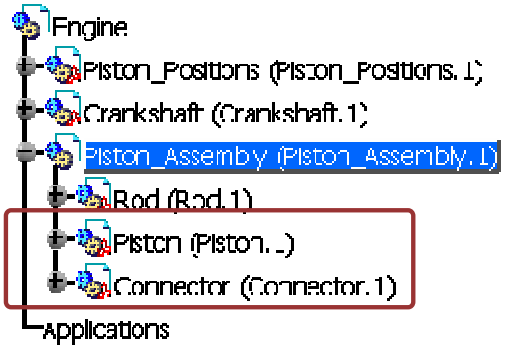
EXERCISE BOOK

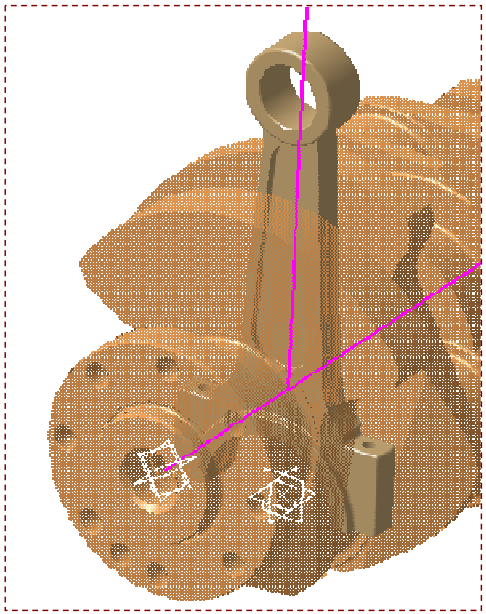
Student Notes:

Do It Yourself

Document used: LO1Engine_Step1.CATProduct

- Edit 'Piston Assembly' inside the root product.
- Insert the following new CATParts:
  - 'Piston'
  - 'Connector'
- Save the assembly using 'Save Management' as "LO1Engine_Step2.CATProduct".
- Save the new CATParts.





Engine Assembly

EXERCISE BOOK

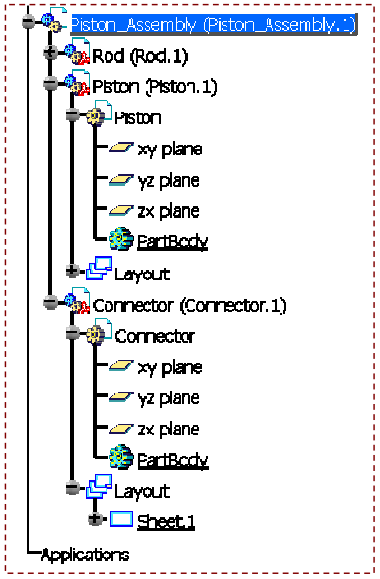
Student Notes:

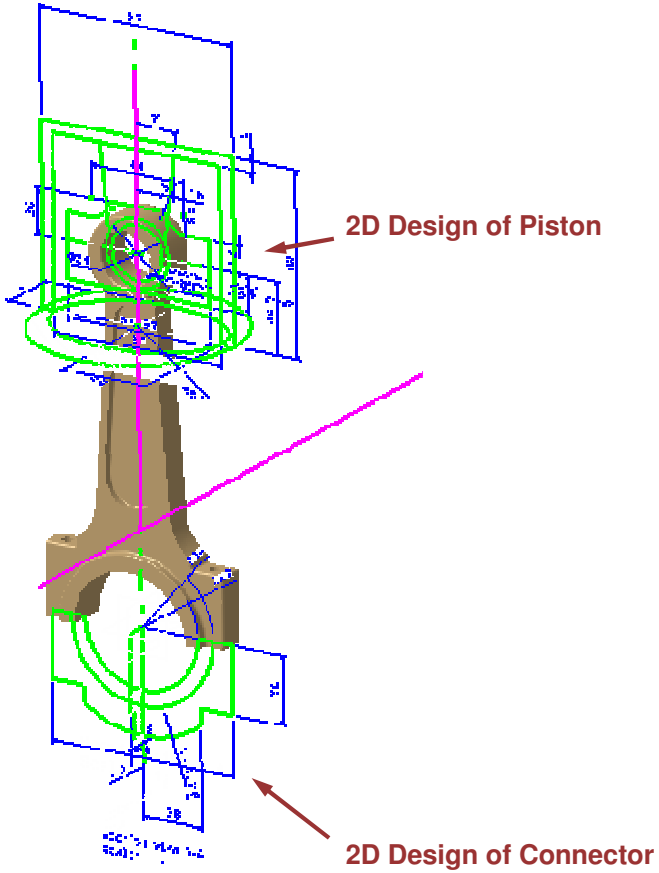
Engine Assembly

Step 2: Design 2D Layout Views

30 min

In this step, you will design the 2D Layout Views of the Piston and Connector parts using the 2D Layout for 3D Design workbench.





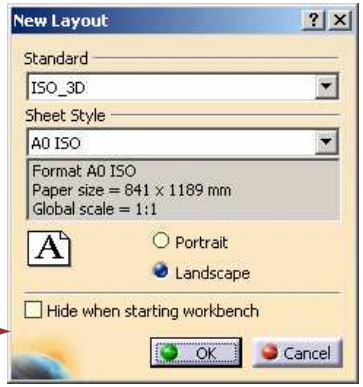
Student Notes:

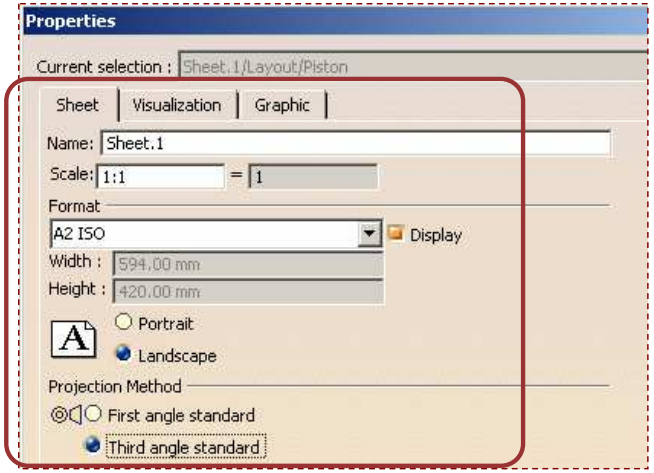
Do It Yourself (1/6)

Document used: LO1Engine_Step2.CATProduct

You will design the 2D views for the Connector.

- Edit the Connector part.
- Switch to '2D Layout for 3D Design' workbench.
- Keep the new layout creation parameters as shown.
- Modify the sheet properties and set the parameters as shown.





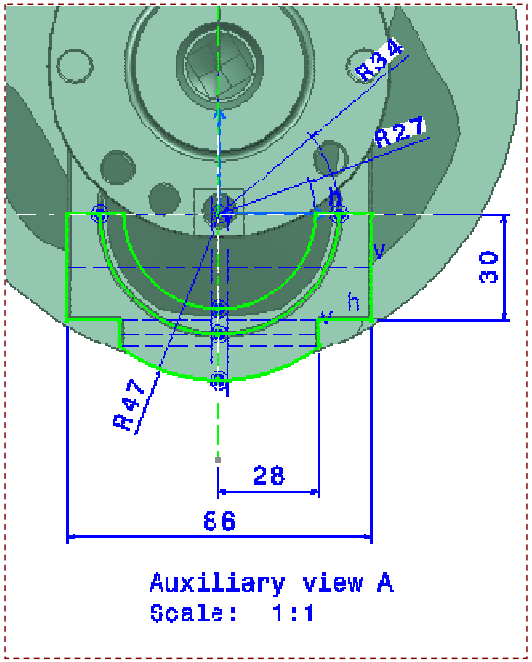
Sheet Properties Window

EXERCISE BOOK

Student Notes:

Do It Yourself (2/6)

- Create a new Auxiliary view using ‘New view From’ tool using yz plane as a reference plane.
- Create a 2D profile of the ‘Connector’ using the dimensions as shown.
- The inside diameter of the hole at the larger end of the Connector is equal to the shaft diameter of the Crank case.



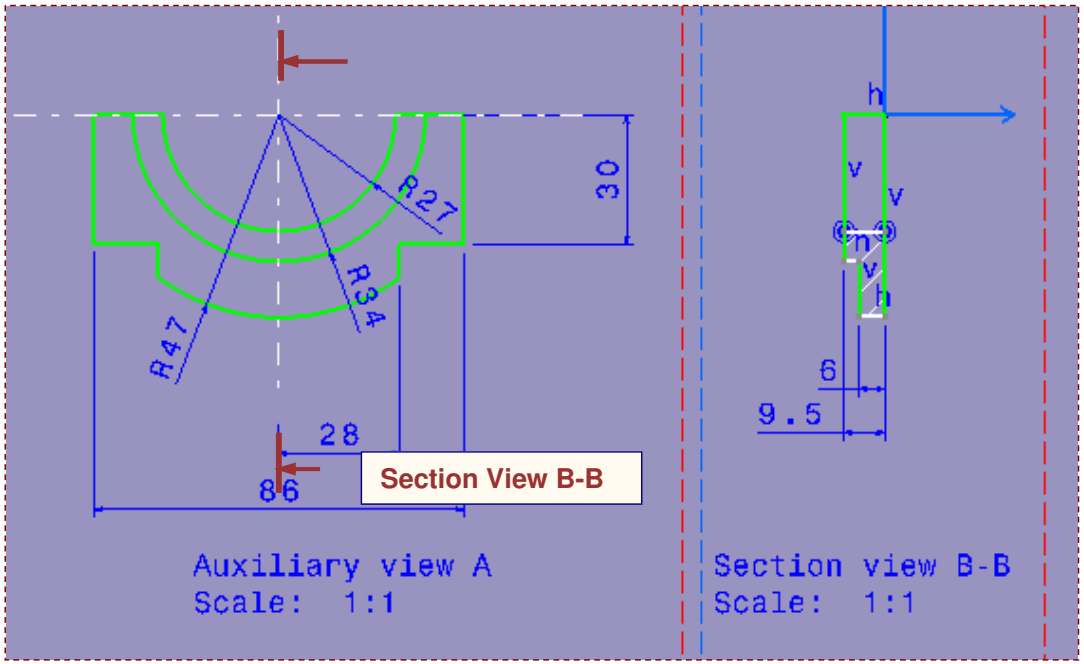
Auxiliary View A

EXERCISE BOOK

Student Notes:

Do It Yourself (3/6)

- Create a new section view as shown below.



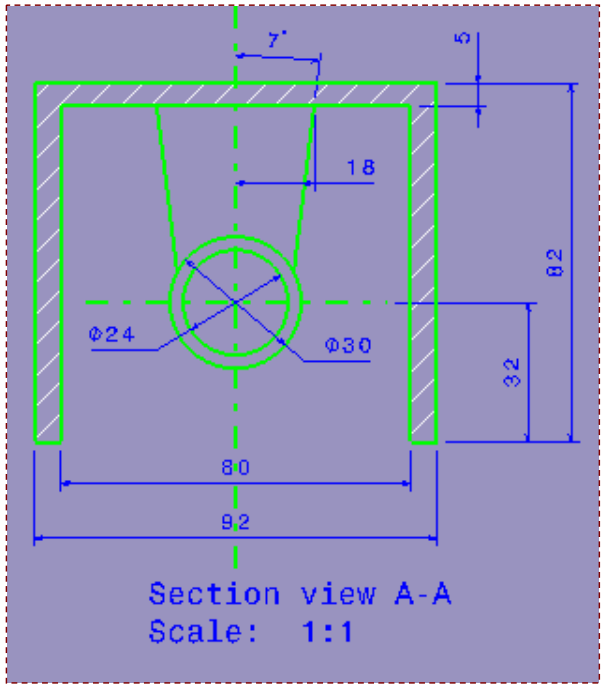
Section View B-B

Student Notes:

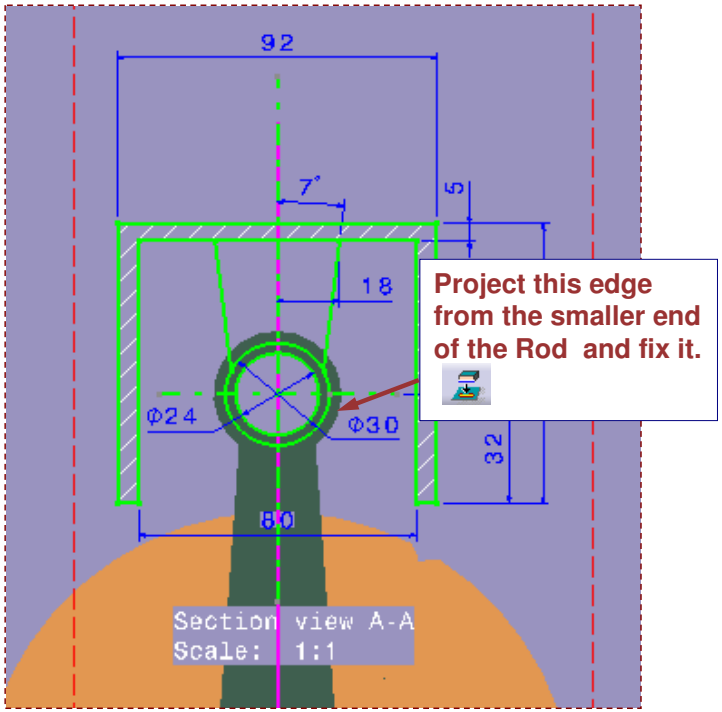
Do It Yourself (4/6)

You will now design the 2D views for the Piston.

- Edit the Piston part.
- Create a new auxiliary section view using 'New view From' tool using yz plane as a reference plane.



Section View A-A



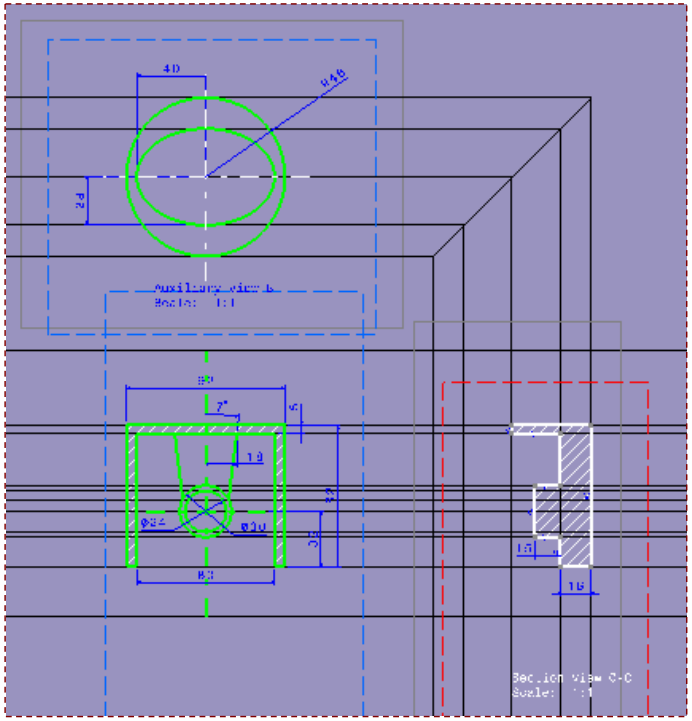
Project this edge from the smaller end of the Rod and fix it.

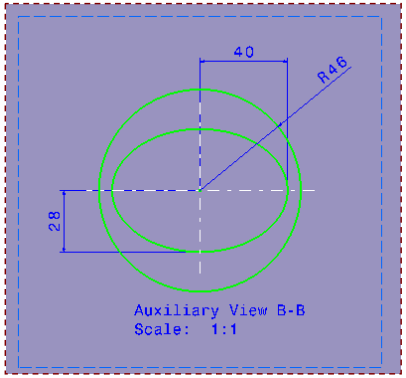
EXERCISE BOOK

Student Notes:

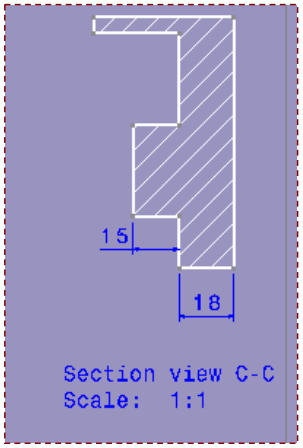
Do It Yourself (5/6)

- Create a new auxiliary section view using 'New Section/Auxiliary View' tool using the reference element as shown.
- Create an additional section view C-C using the folding lines as shown.





Auxiliary View B

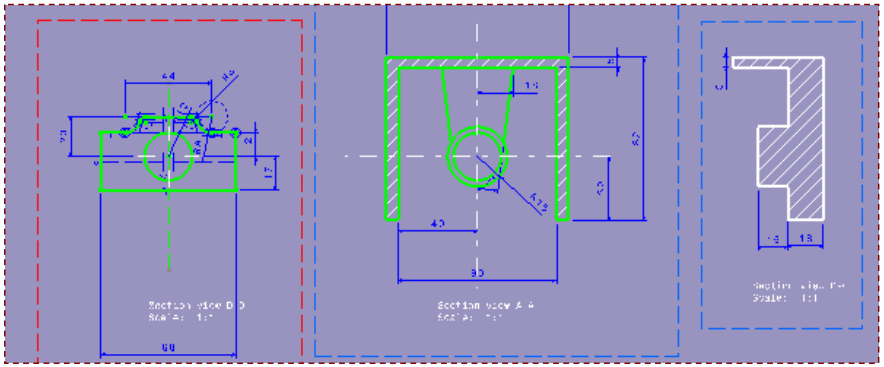


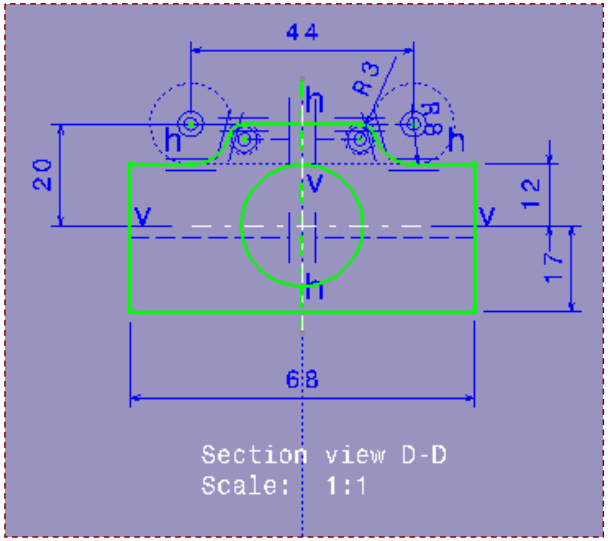
Section View C-C

Student Notes:

Do It Yourself (6/6)

- Create an additional auxiliary section view from Section View C-C.





Section View D-D

- Activate the root CATProduct and save the assembly as “LO1Engine_Step3.CATProduct”

EXERCISE BOOK

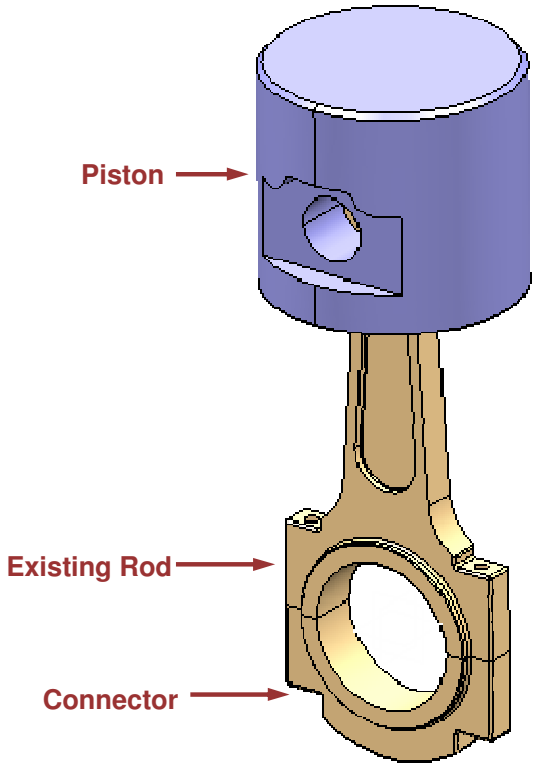
Student Notes:

Engine Assembly

Step 3: Create 3D Design

45 min

In this step, you will create the 3D design of Piston and Connector from their 2D views.

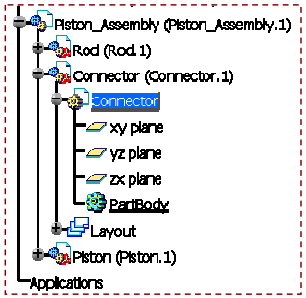


Student Notes:

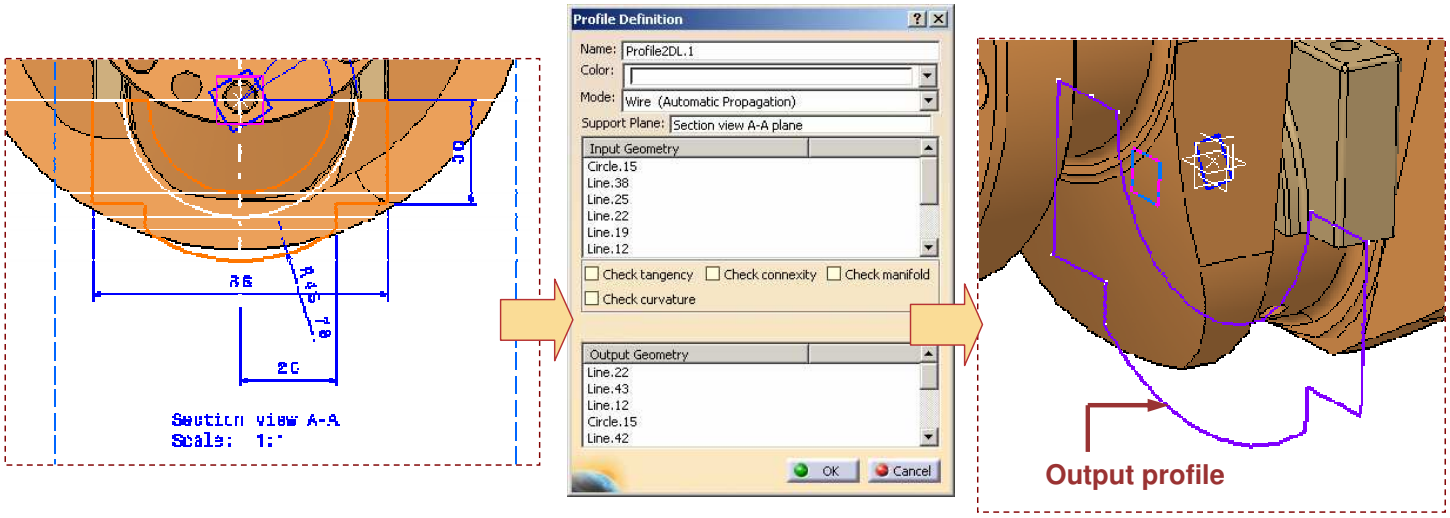
Do It Yourself (1/29)

Document used: LO1Engine_Step3.CATProduct

- Edit Connector part.



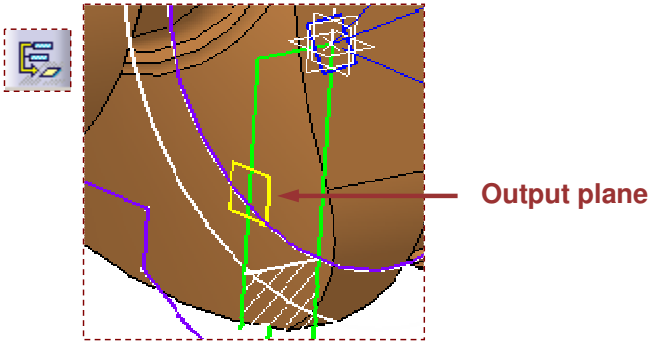
- Create the 3D profile from the Section view B-B using '3D Profile' tool.



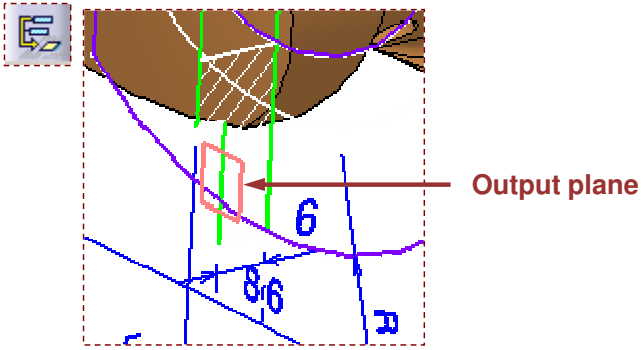
Student Notes:

Do It Yourself (2/29)

- Create a plane using '3D Plane' tool and passing through the vertical line in the Section view B-B as shown.



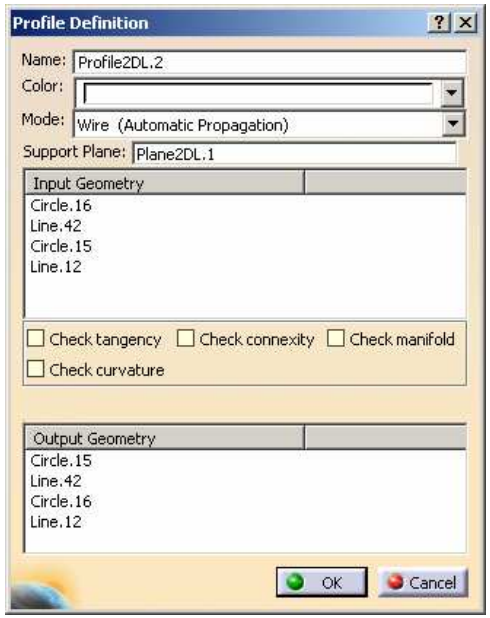
- Create another plane using '3D Plane' tool and passing through the vertical line in the Section view B-B as shown.

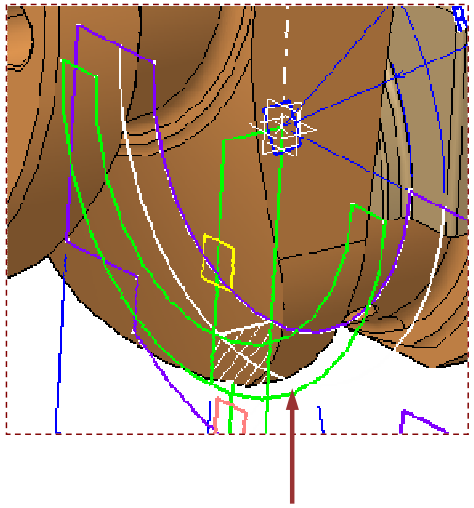


Student Notes:

Do It Yourself (3/29)

- **Output the 3D profile of the connector on Plane 2DL.1 from the Section view A-A using '3D Profile' tool.**



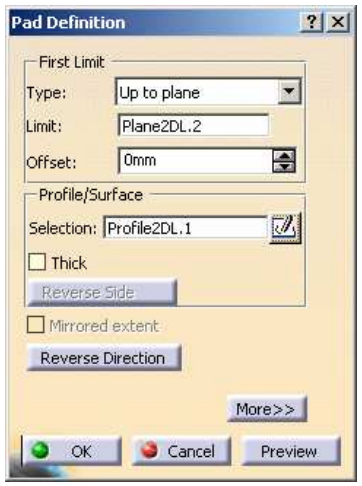


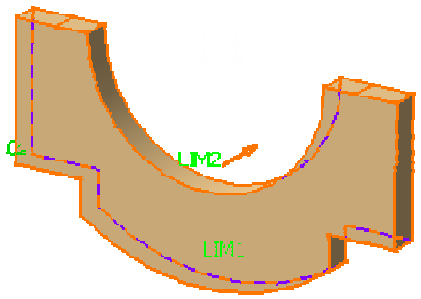
Output profile

Student Notes:

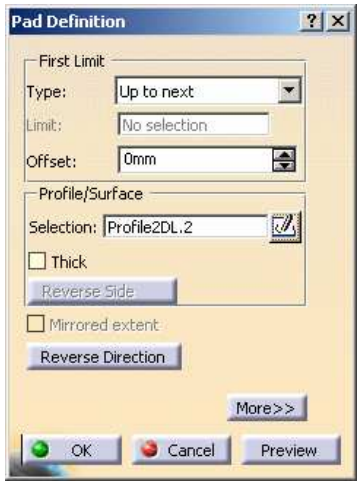
Do It Yourself (4/29)

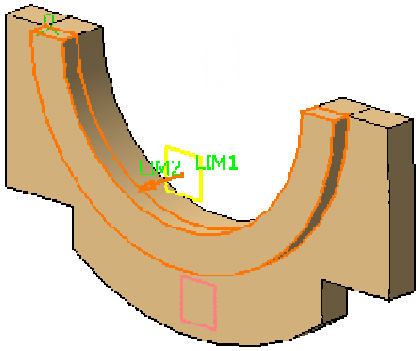
- Create a pad using following inputs:
  - Use 'Profile2DL.1' as input profile
  - Plane2DL.2 as limit plane





- Create a pad using following inputs:
  - Use 'Profile2DL.2' as input profile
  - Use 'Up to next' as limit type

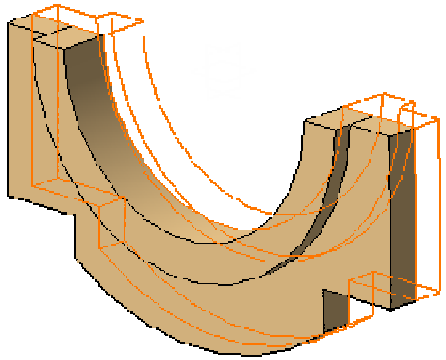




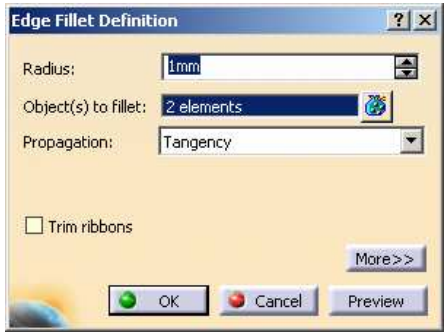
Student Notes:

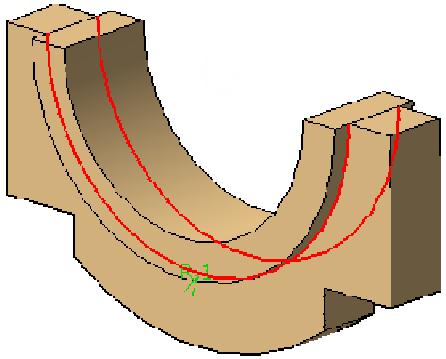
Do It Yourself (5/29)

- **Create a Mirror of using Current Solid as object to mirror and face of the pad as mirror plane.**



- **Create Edge fillets of 1mm as shown**

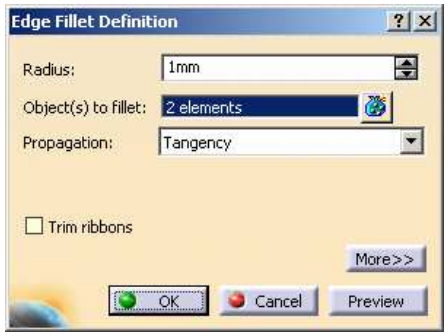


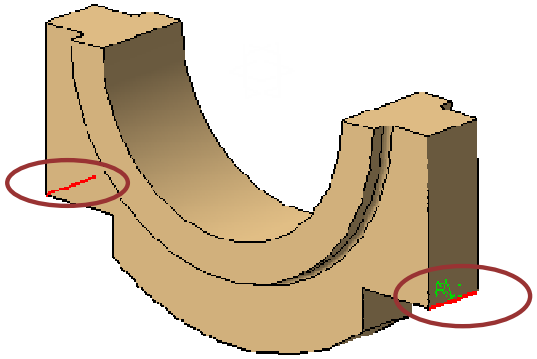


Student Notes:

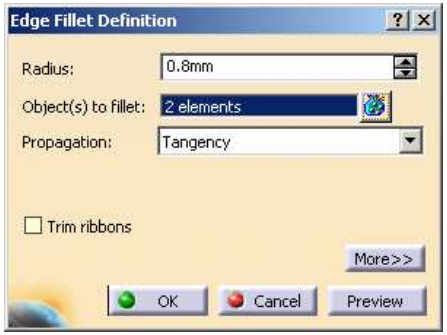
Do It Yourself (6/29)

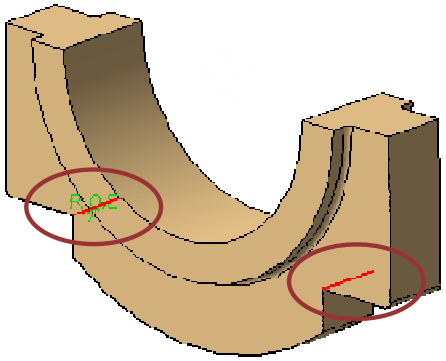
- **Create Edge fillets of 1mm as shown**





- **Create Edge fillets of 0.8mm as shown**

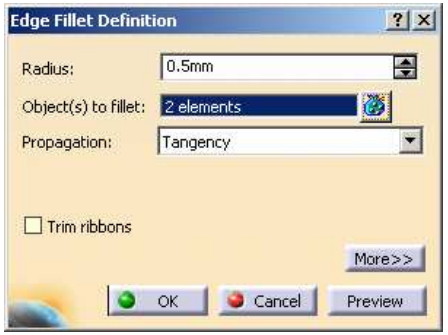


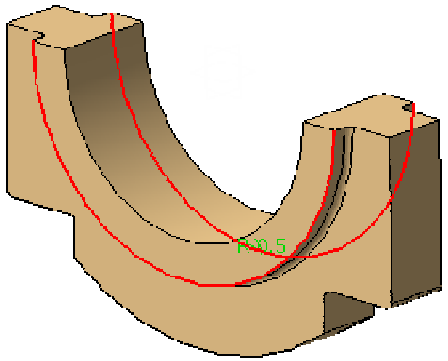


Student Notes:

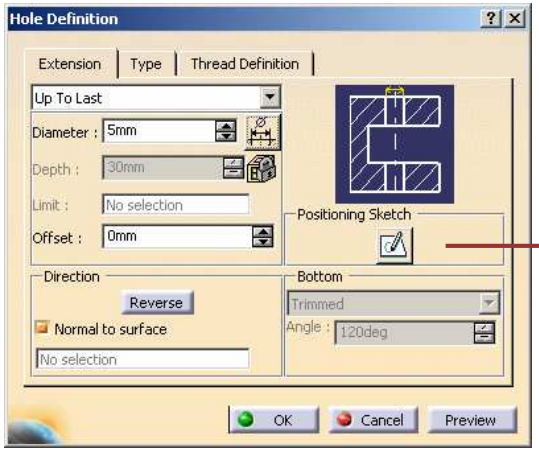
Do It Yourself (7/29)

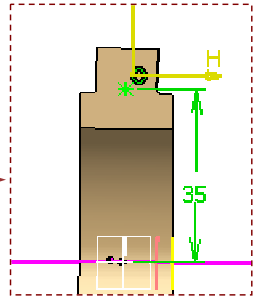
- **Create Edge fillets of 0.5mm as shown**

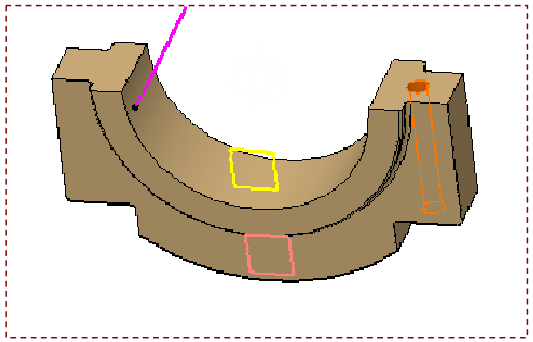




- **Create a through hole of Diameter 5mm and position as shown**



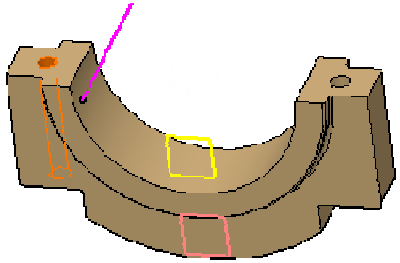




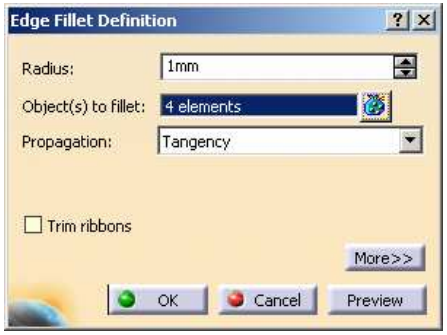
Student Notes:

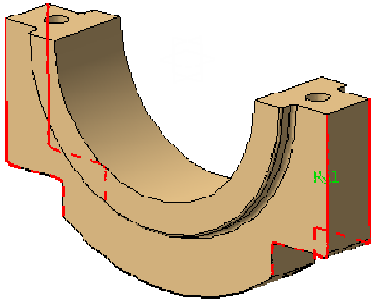
Do It Yourself (8/29)

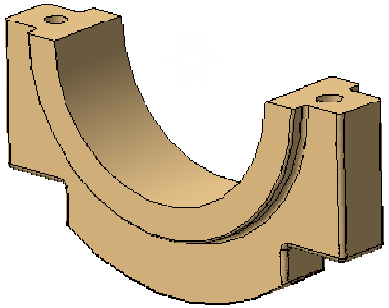
- **Create a mirror of the hole using zx plane as mirroring element**



- **Create Edge fillet of 1mm as shown**



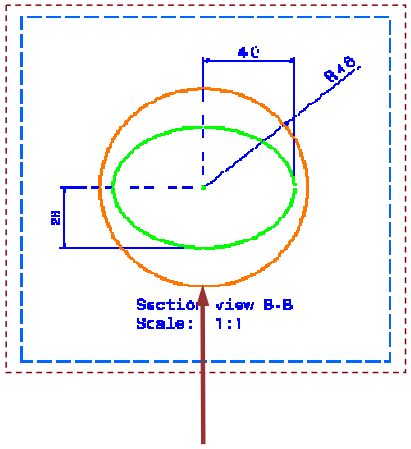




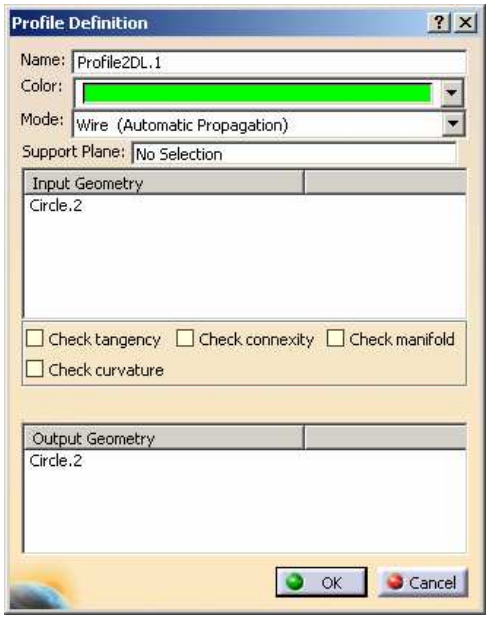
Student Notes:

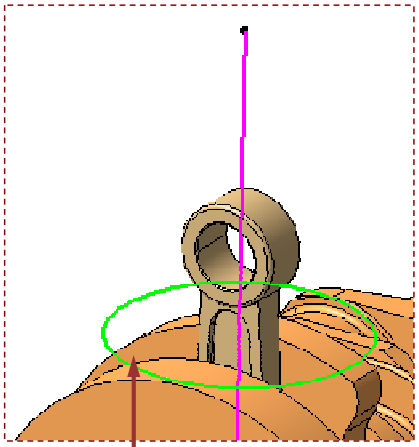
Do It Yourself (9/29)

- Edit Piston part.
- Create the 3D profile of the piston from the Auxiliary view B-B using '3D Profile' tool.



Output this profile



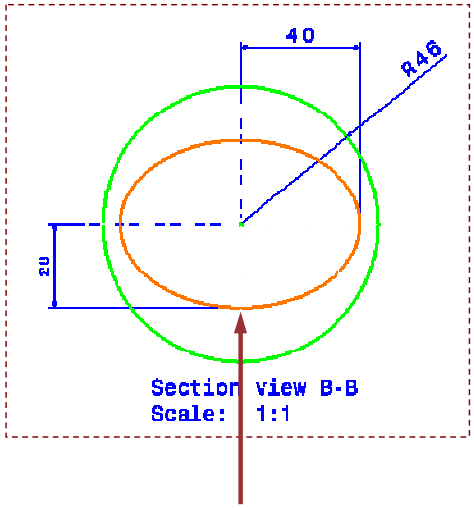


Output profile

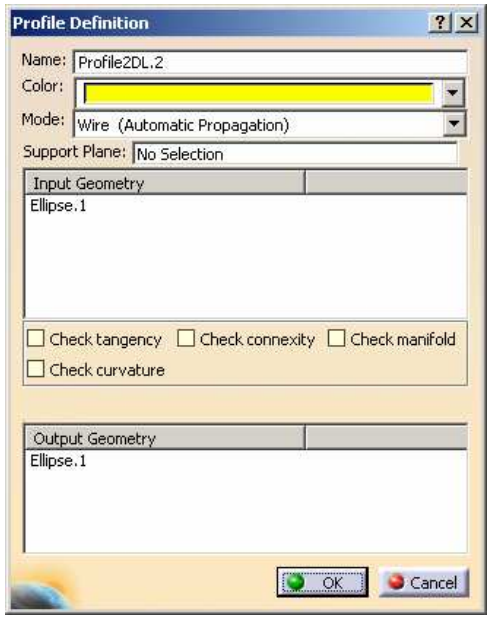
Student Notes:

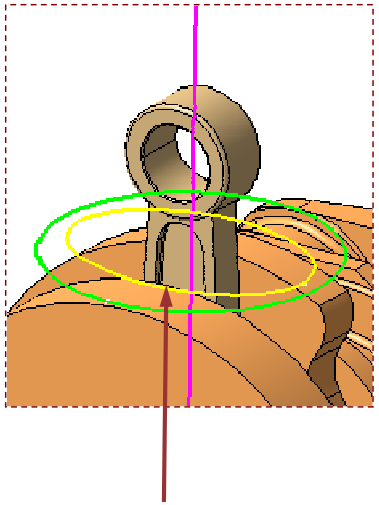
Do It Yourself (10/29)

- Create another 3D profile of the piston from the Auxiliary view B-B using '3D Profile' tool.



Output this profile





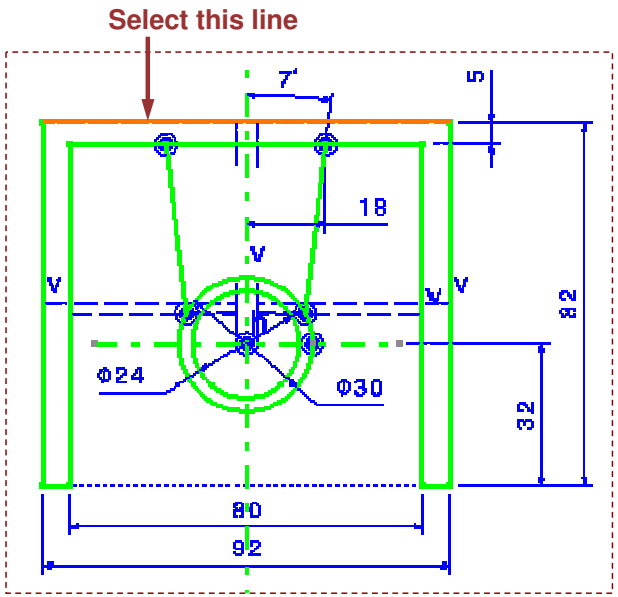
Output profile

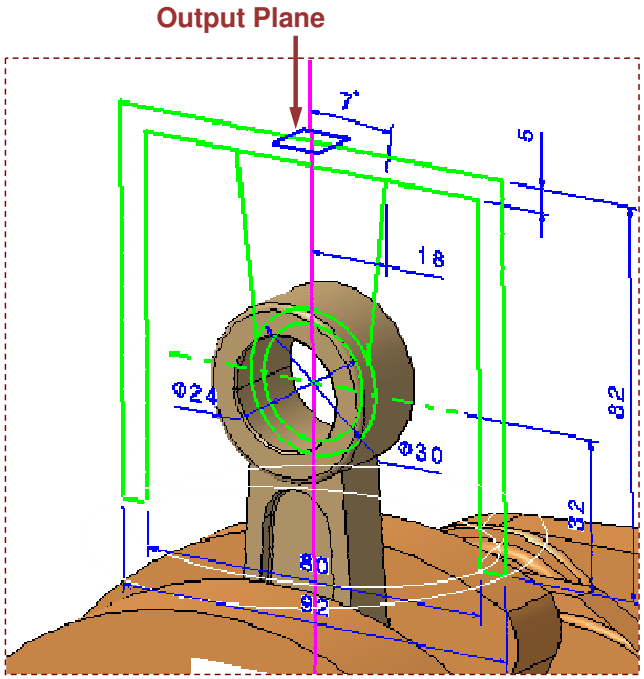
EXERCISE BOOK

Student Notes:

Do It Yourself (11/29)

- Create a plane using '3D Plane' tool and select the horizontal line in the Section view A-A as shown.

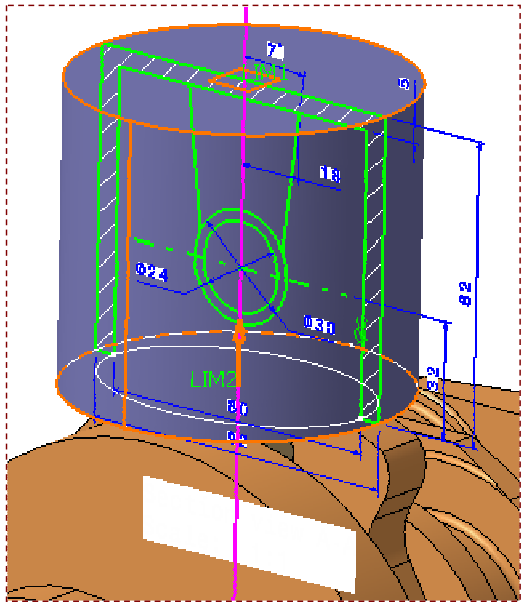


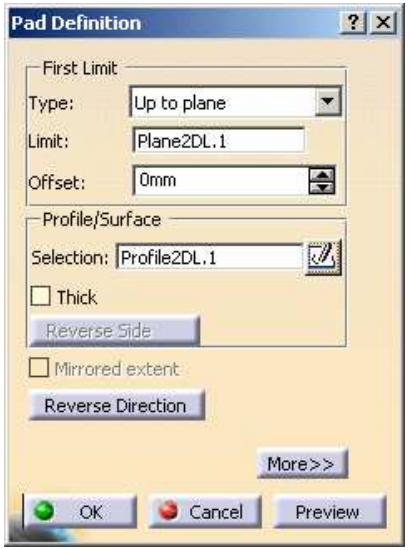


Student Notes:

Do It Yourself (12/29)

- Create a pad using following inputs:
  - Use ‘Plane2DL.1’ as limit plane
  - Use ‘Profile2DL.1’ as input profile



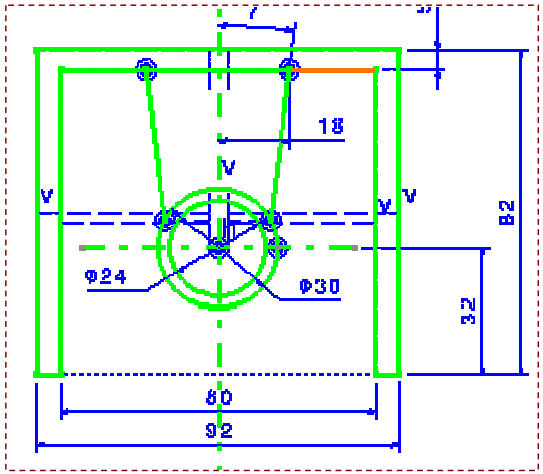


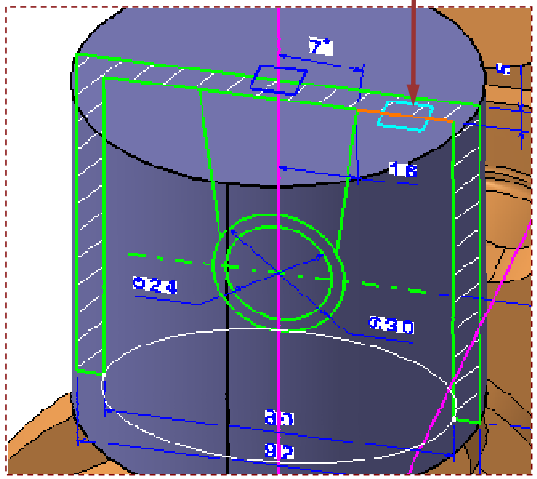
EXERCISE BOOK

Student Notes:

Do It Yourself (13/29)

- Create a plane using '3D Plane' tool and passing through the horizontal line in the Section view A-A as shown.

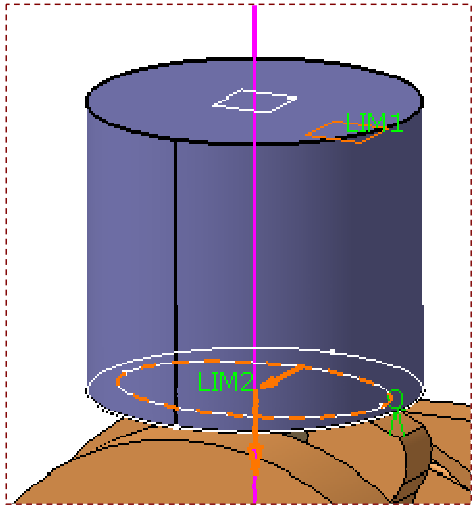


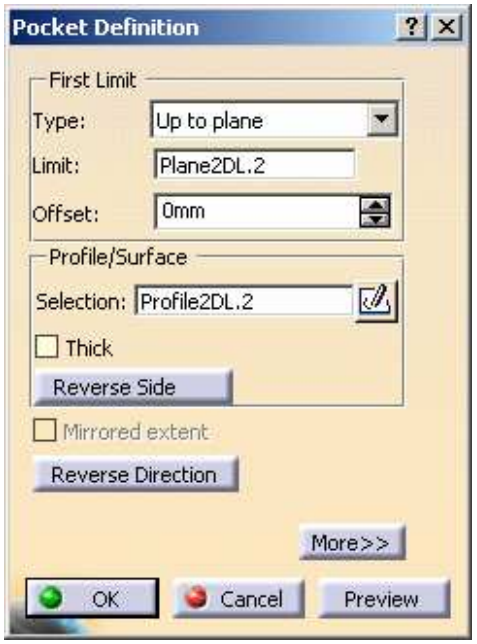


Student Notes:

Do It Yourself (14/29)

- Create a pocket using following inputs:
  - Use 'Plane2DL.2' as limit plane
  - Use 'Profile2DL.2' as input profile

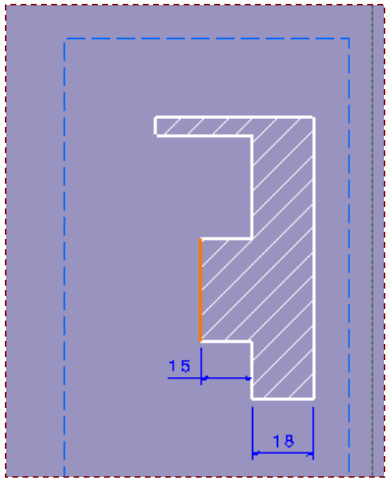




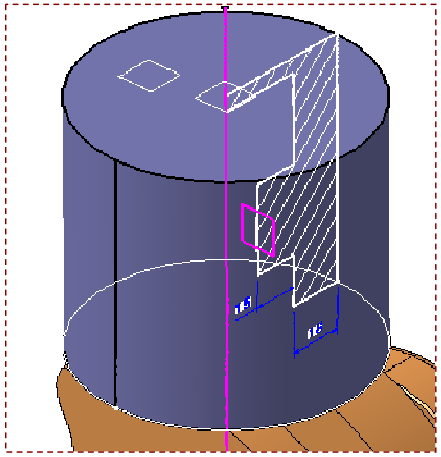
Student Notes:

Do It Yourself (15/29)

- Insert a new Body in the 'Piston' and activate this body.
- Create a plane using '3D Plane' tool and passing through the vertical line in the Section view C-C as shown.



Select this line

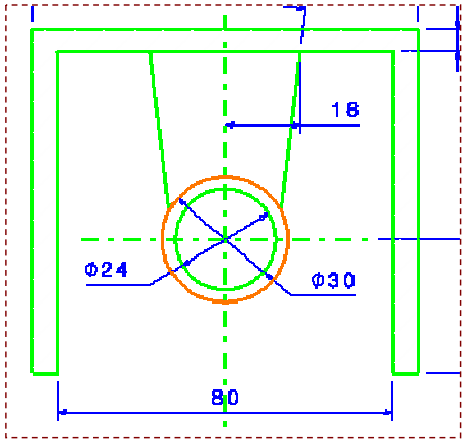


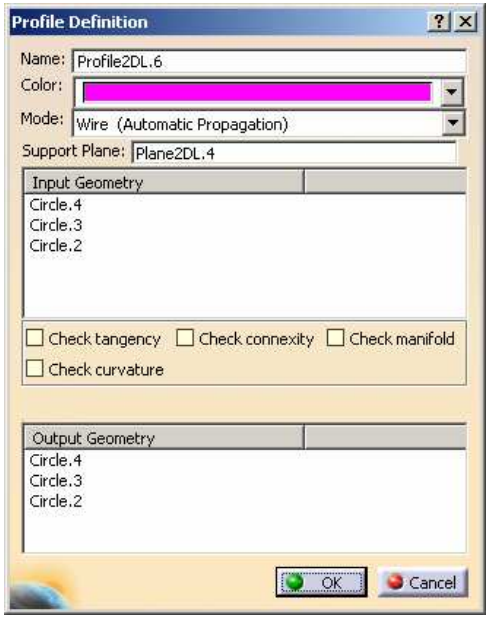
Output Plane

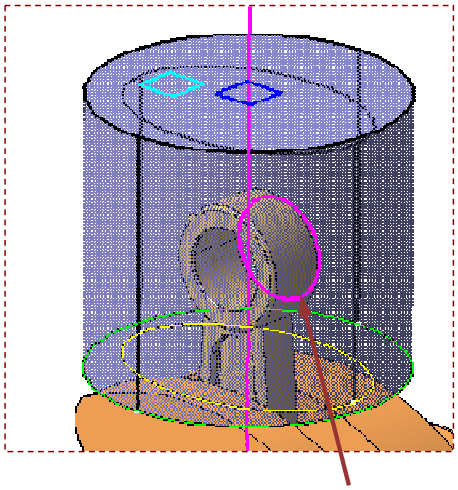
Student Notes:

Do It Yourself (16/29)

- Create another 3D profile of the piston from the Section view A-A using '3D Profile' tool.
- Use 'Plane2DL.4' as Support Plane.





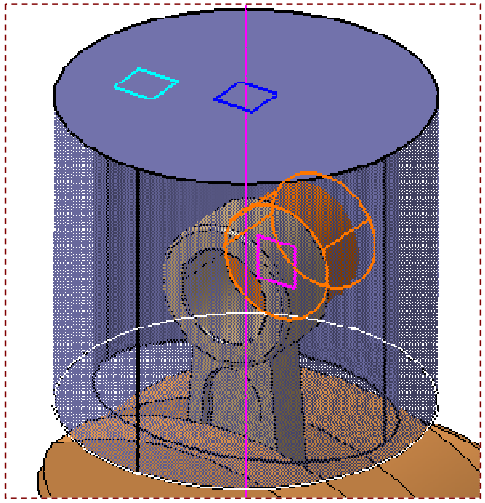


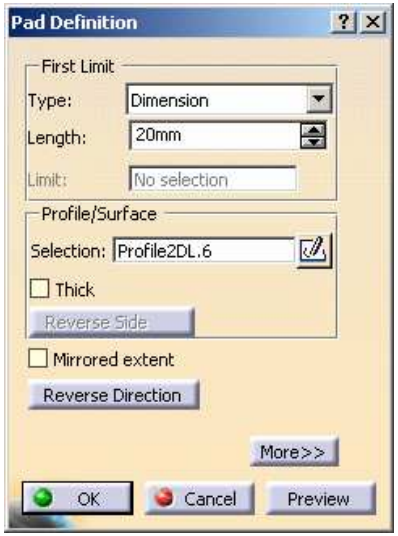
Output profile

Student Notes:

Do It Yourself (17/29)

- Create a pad using following inputs:
  - Use 'Profile2DL.6' as input profile

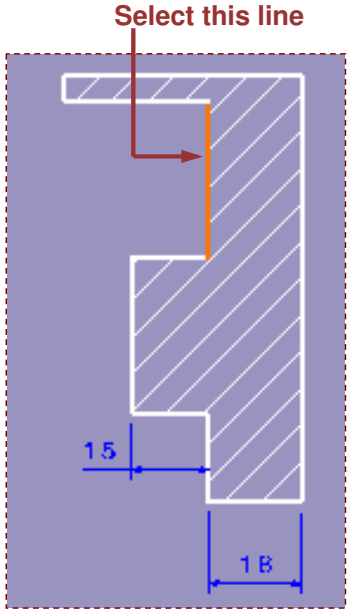


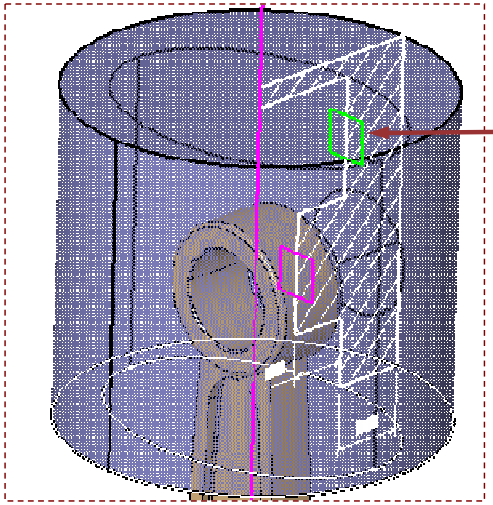


Student Notes:

Do It Yourself (18/29)

- Create a plane using '3D Plane' tool and passing through the vertical line in the Section view C-C as shown.

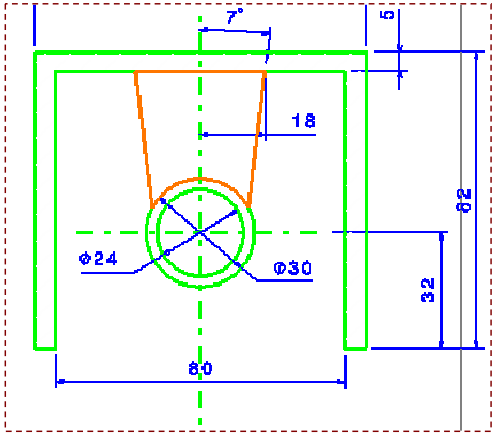


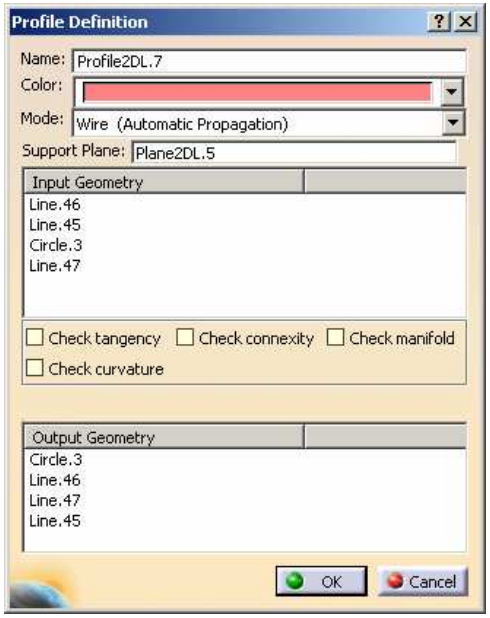


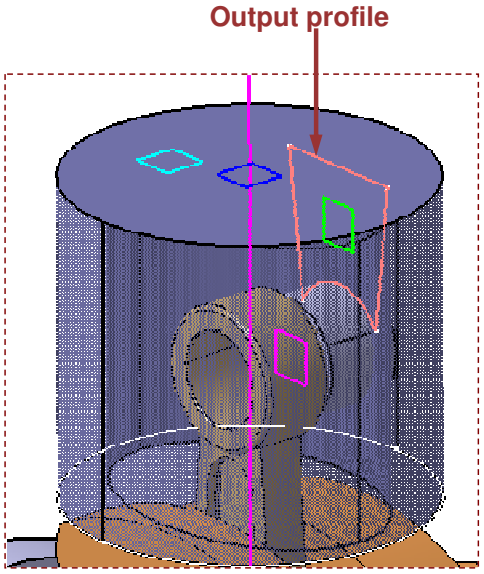
Student Notes:

Do It Yourself (19/29)

- Create a 3D profile of the piston from the Section view A-A using '3D Profile' tool.
- Use 'Plane2DL.5' as Support Plane.



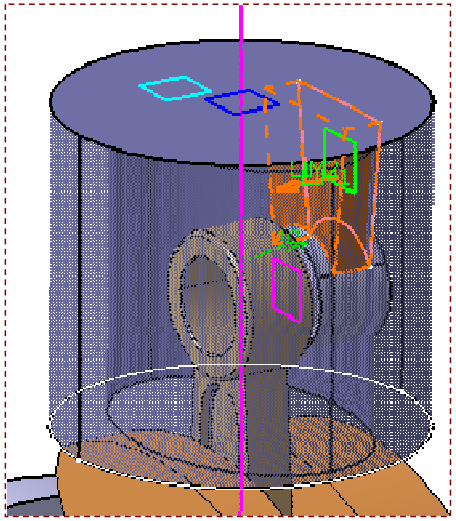


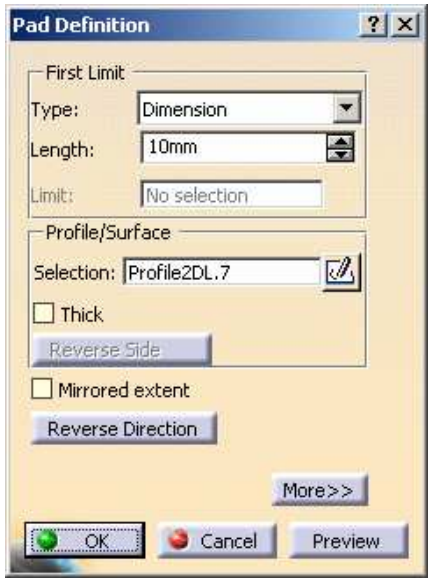


Student Notes:

Do It Yourself (20/29)

- Create a pad using following inputs:
  - Use 'Profile2DL.7' as input profile
  - Direction as shown

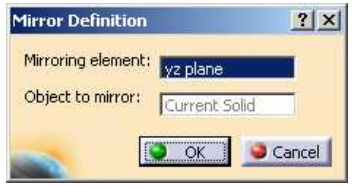


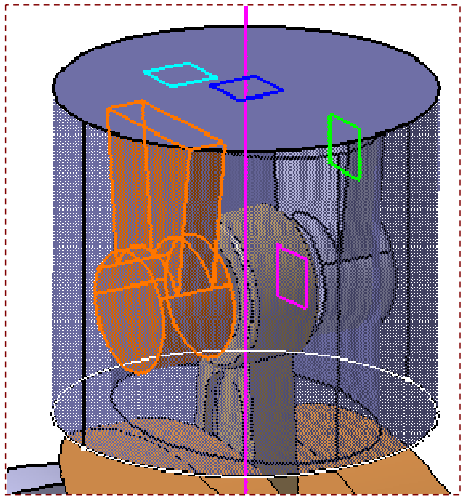


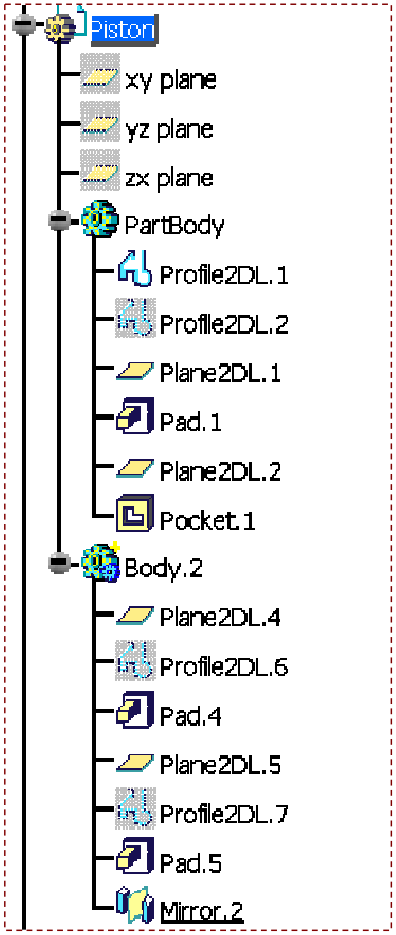
Student Notes:

Do It Yourself (21/29)

- **Create a Mirror of Body.2 using yz plane as mirroring element**



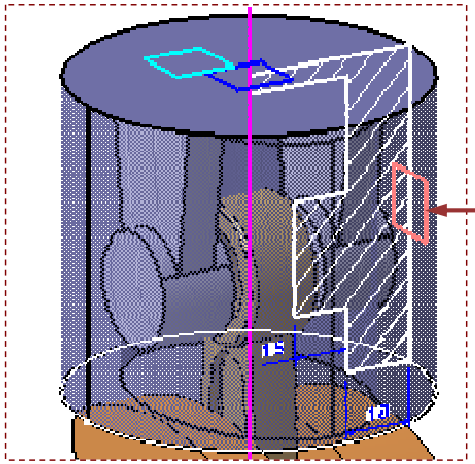




Student Notes:

Do It Yourself (22/29)

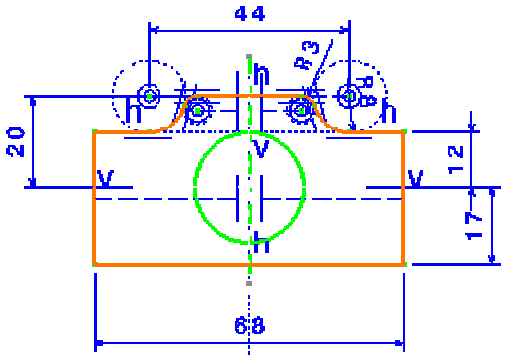
- Insert a new Body in the ‘Piston’ and activate this body.
- Create a plane using ‘3D Plane’ tool and passing through the vertical line in the Section view C-C as shown.

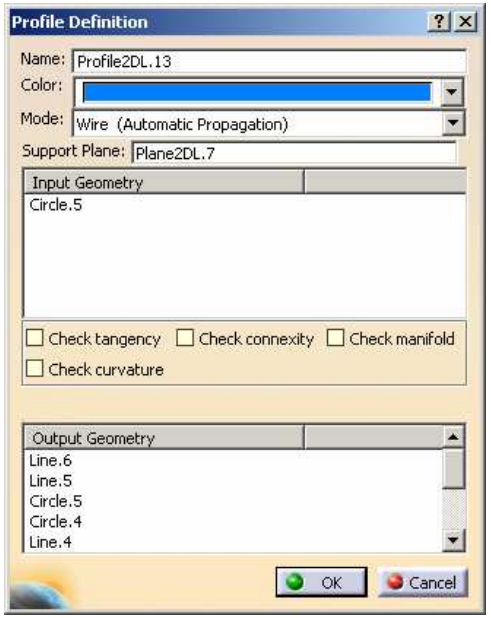


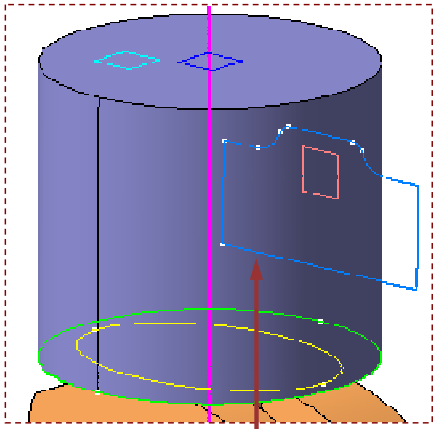
Student Notes:

Do It Yourself (23/29)

- Create a 3D profile of the piston from the Auxiliary view D-D using '3D Profile' tool.
- Use 'Plane2DL.7' as Support Plane.





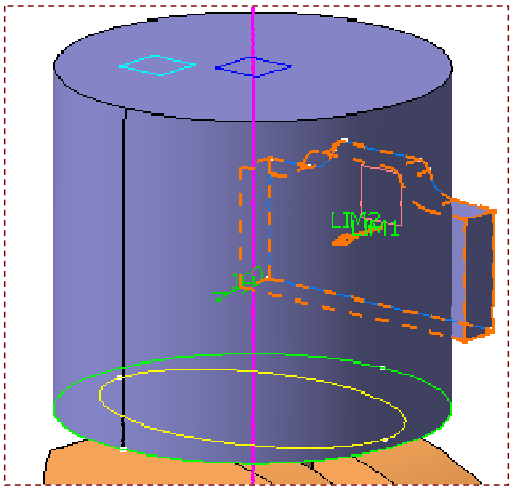


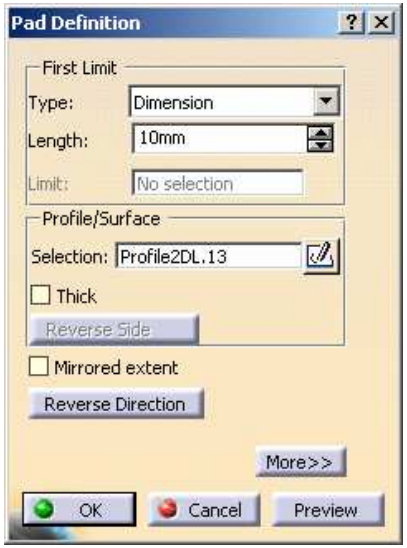
Output profile

Student Notes:

Do It Yourself (24/29)

- Create a pad using following inputs:
  - Use 'Profile2DL.13' as input profile
  - Direction as shown

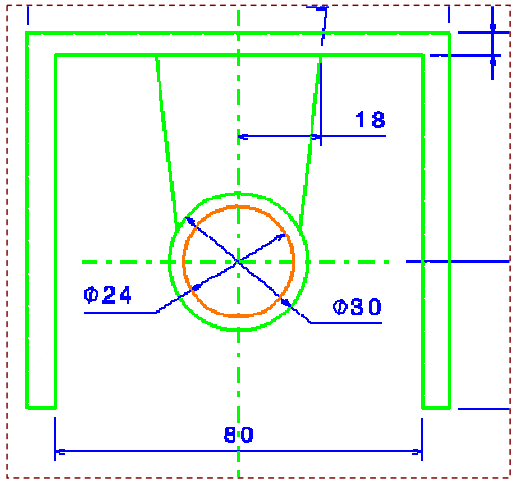


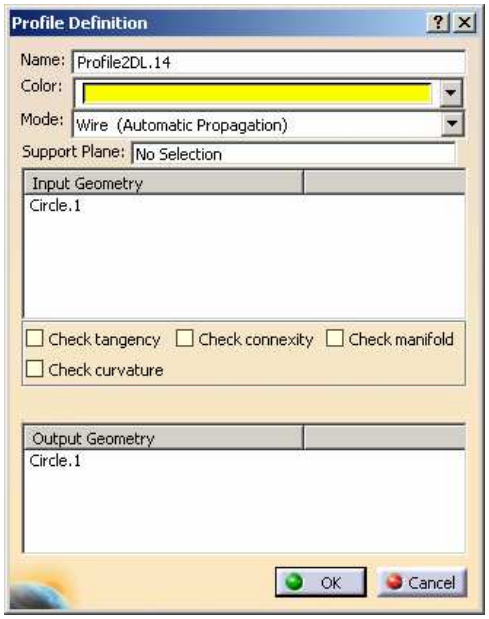


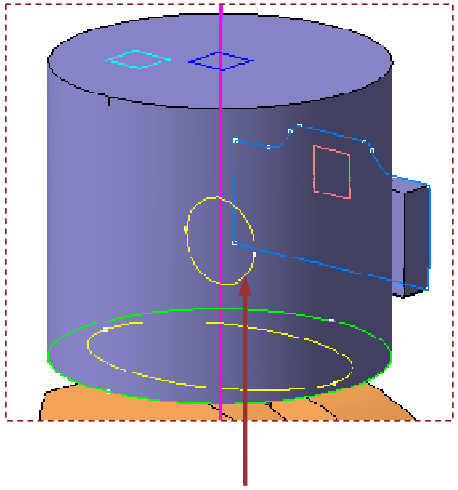
Student Notes:

Do It Yourself (25/29)

- Create a 3D profile of the piston from the Section view A-A as shown.





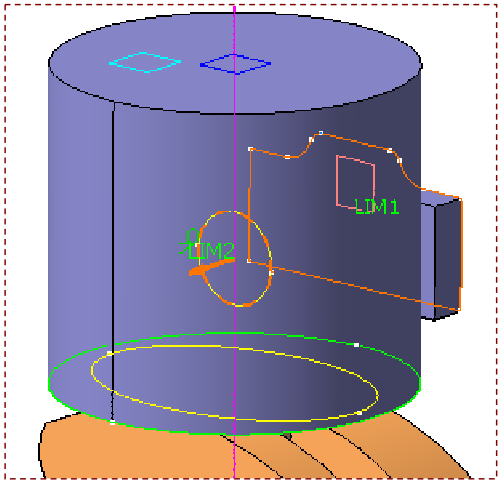


Output profile

Student Notes:

Do It Yourself (26/29)

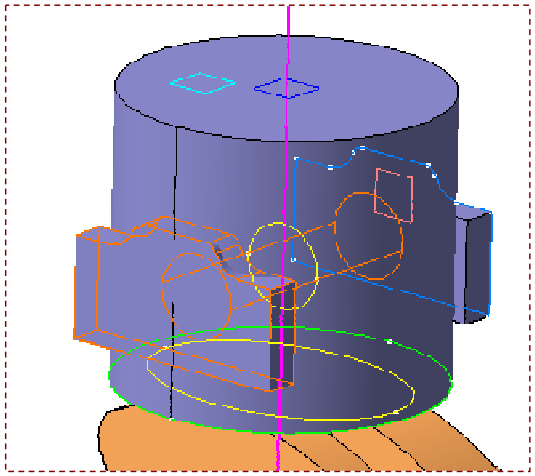
- Create a pad using following inputs:
  - Use 'Profile2DL.14' as input profile
  - Face of Pad.8 as limit

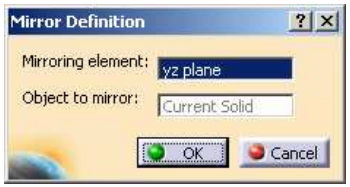


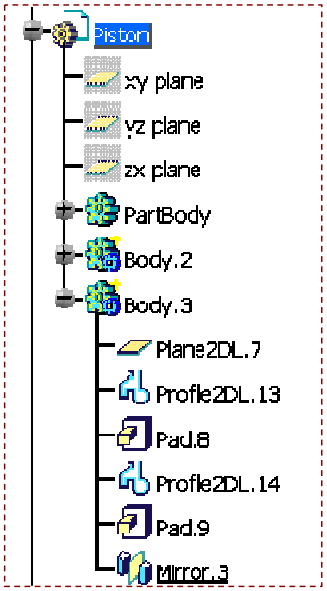
Student Notes:

Do It Yourself (27/29)

- **Create a Mirror of Body.3 using yz plane as a mirror plane.**



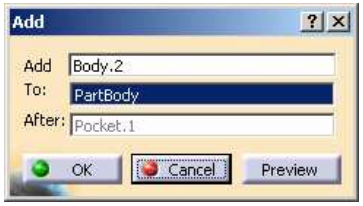




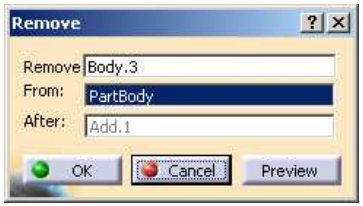
Student Notes:

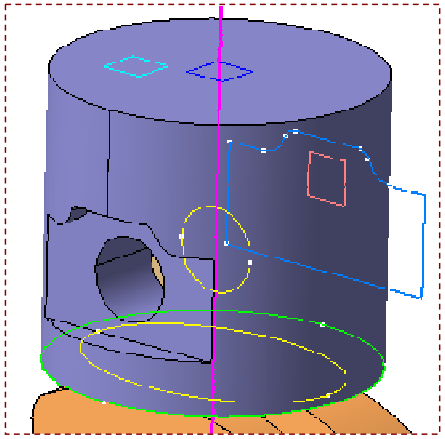
Do It Yourself (28/29)

- Activate PartBody.
- Perform a Boolean operation ‘Add’ and select ‘Body.2’ as a body to be added to PartBody.



- Perform a Boolean operation ‘Remove’ and select ‘Body.3’ as a body to be removed from PartBody.

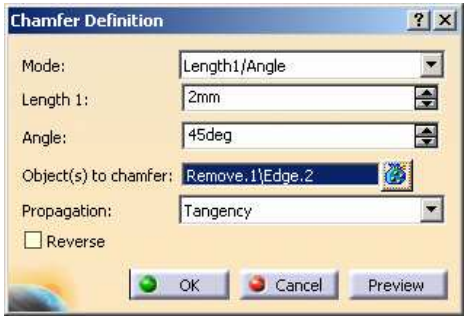


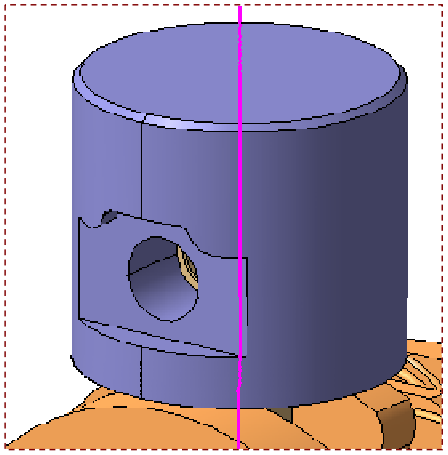


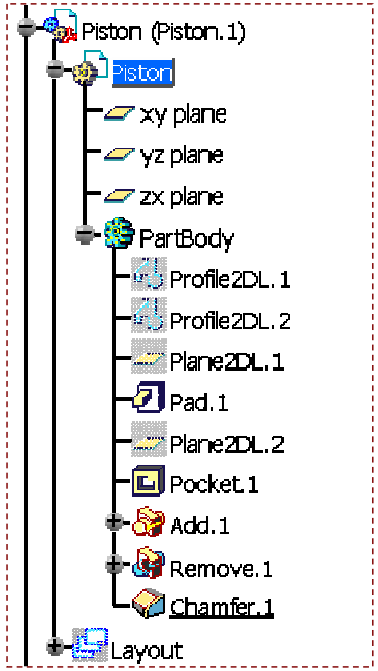
Student Notes:

Do It Yourself (29/29)

- Add a 2mm x 45 deg chamfer on the top edge of the Piston as shown.







- Activate the root CATProduct and save the assembly as LO1Engine_Step4.CATProduct

EXERCISE BOOK

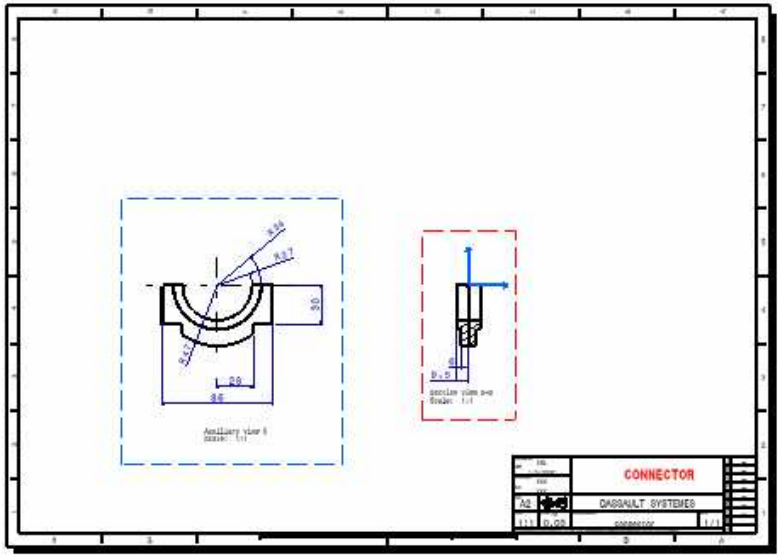
Student Notes:

Engine Assembly

Step 4: Create Drawing Views

5 min

In this step, you will create drawing views of the Connector and Piston from the 2D views.

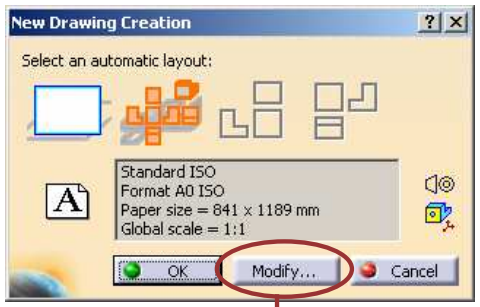


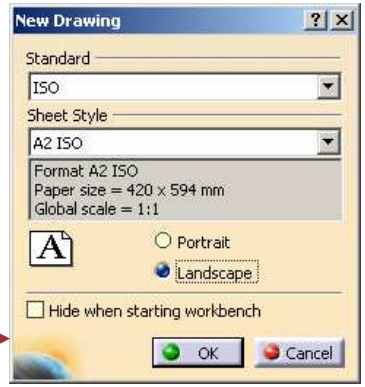
Student Notes:

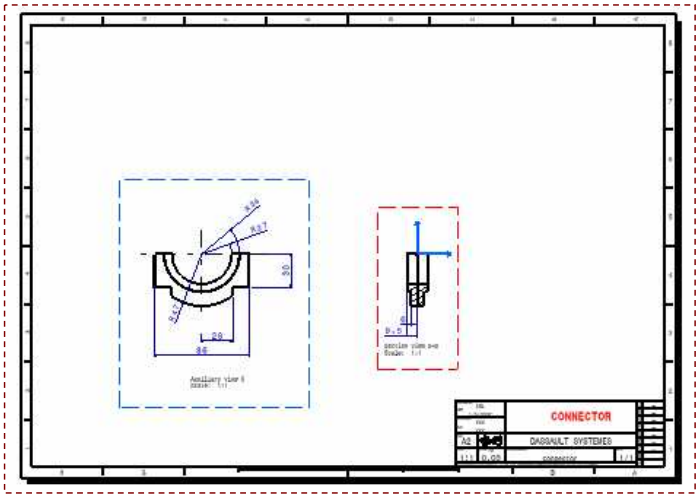
Do It Yourself

Document used: LO1Engine_Step4.CATProduct

- Edit Connector part. Switch to 2D Layout for 3D Design workbench.
- Switch to Drafting workbench and create a new Drawing with the sheet parameters and the layout as shown.
- Save the Drawing as Connector.CATDrawing.







Student Notes:

Tyre Assembly

Additional Exercise

60 min.

In this exercise you will modify the design changing the tire diameter from 32mm to 25mm.

Student Notes:

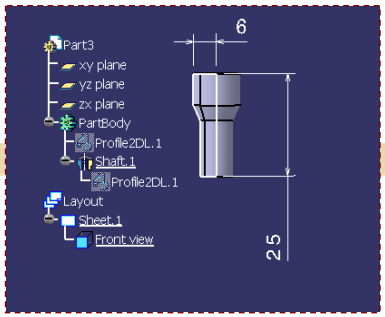
Do It Yourself

Open the “Tyre_assembly.CATProduct”

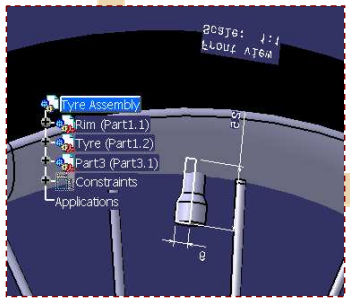
1 Open the Tire_assembly.CATProduct

2 Change the tire and rim diameter from 32mm to 25mm

3 Create a new layout CATPart to design the air valve. Use the provided catalog to design the valve. Instantiate the 2D component, explode it, create the 3D profile and the shaft



4 Insert the valve in the assembly and position it



5 Complete the layout dress-up and extract the associated CATDrawing

Open the “Tyre_assembly_step3.CATProduct” and compare your end result.