The Institution of Engineering and Technology

IET SMP

CATIA

FINAL PROJECT

SUBMITED BY –

ABHISHEK YADAV

(CIVIL ENGINEERING)

(1ST YEAR) (171CV202)

Acknowledgement

I would like to express my special thanks to IET club and it's members at National Institute of Technology Karnatka, Surathkal who gave me the golden opportunity to learn about CATIA software in my summer break.

Video tutorials provided by the club were really useful and I came to know many new things from them.

I also want to thank the youtube channels like CAD CAM TUTORIAL , CAD Designs. I came to know a lot more from them.

That's all.

ABSTRACT

The purpose of study and project was to learn about 3 dimension modeling of the object using CATIA software. The project making involved using various features in CATIA v5 in order to stimulate the 3 dimensional object in CATIA working environment. These type of modeling leads to better understanding of the product to be designed in real life. My final project involved making of various parts of twin cylinder inline engine and assembling them properly.

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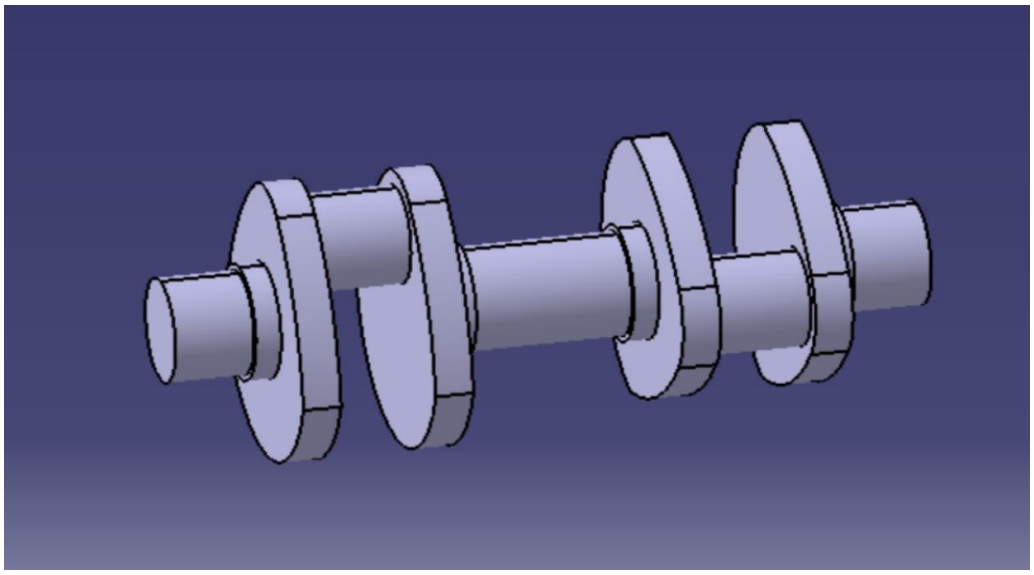
PROBLEM DEFINITION

Q.)

Create the piston cylinder assembly of Twin cylinder, inline engine.

- **Create each parts(Piston, Cylinder block, connecting rod, piston pin, cranck shaft)**
- **Assemble the parts correctly.**
- **Create a drawing file containing front, top and side views with necessary details. Also include a sectional view in the plane passing through the centre.**
- **Create a detailed report about your work.**

CREATING CRANCKSHAFT



Steps in creating cranckshaft :-

1. Chose yz plane and clicked on the sketch feature.
2. Created a circle of diameter 34 mm at origin using circle profile tool.
3. Using pad feature extended it upto 25 mm to make a cylinder.
4. Used sketch feature and clicked on the circular plane opposite to the plane on which the base circle of 34 mm was drawn.

5. Used “project 3D elements” tool and projected the circle of 34 mm on the plane. Using center of the circle drew another circle of 38 mm and extended it upto 8 mm along the cylinder.
6. Clicked on the surface plane of 38 mm circle and used sketch feature.
7. Using arc feature of profile tools drew an arc of 46 mm having end points which were horizontal to eachother having a distance of 88 mm between them.
8. Drew another circle of diameter 42 mm having center just 25 mm vertically above the center of the circle of 38 mm.
9. Used “Three point arc starting with limits” feature to construct an arc of radius 90 mm between the

end point of arc of 46 mm and the circle of 42 mm diameter.

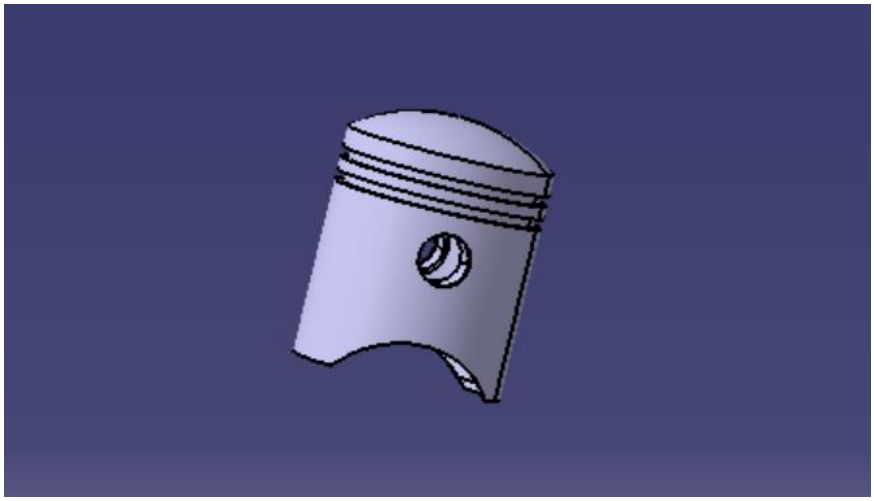
10. Made the arc tangent to the circle using tangency constraint .
11. Using “Quicktrim” feature removed the unnecessary parts of the circle.
12. Used pad feature and extruded the shape to 12 mm along the surface.
13. Clicked on the extruded face and used sketch feature. Drew a circle of diameter 32 mm having center 25 mm above that of origin.
14. Extruded this circle upto 15 mm along the surface using pad feature.
15. Used the mirror feature to create half part of the cranckshaft. Used the circular plane of 32 mm face as the mirroring element.

16. Used the sketch feature and clicked on the circular plane of 34 mm. Using “project 3D elements” feature and projected the circular part on the surface.
17. Using pad feature extruded the cylinder to 25 mm.
18. Again clicked on the obtained circular plane of 34 mm and drew a circle of radius 38 mm keeping the center same and extruded it upto 8 mm.
19. Clicked on the circular plane obtained of 38 mm and used sketch feature.
20. Using “project 3D elements” projected the shape obtained in step 11.

21. Using rotation feature made this shape at an angle of 90° to the previous one.
22. Using pad feature extruded this shape upto 12mm.
23. Clicked on the face of the shape obtained and used sketch feature. Drew a circle of radius 32 mm with it's center 25 mm vertically above that of origin. Using sketch feature extruded it upto 30 mm.
24. Clicked on the the circular face of diameter 32 mm obtained and used sketch feature.
25. Using "project 3D elements" feature projected the shape obtained in step 21 on the selected plane. Using pad feature extruded it upto 12 mm.

26. Clicked on the face obtained and used sketch feature. Using origin as center drew a circle of 38 mm and extruded it upto 8 mm.
27. Clicked on the face obtained and used sketch feature. Using origin as center drew a circle of 34 mm and extruded it upto 25 mm.
28. In this way cranckshaft is created.

CREATING PISTON

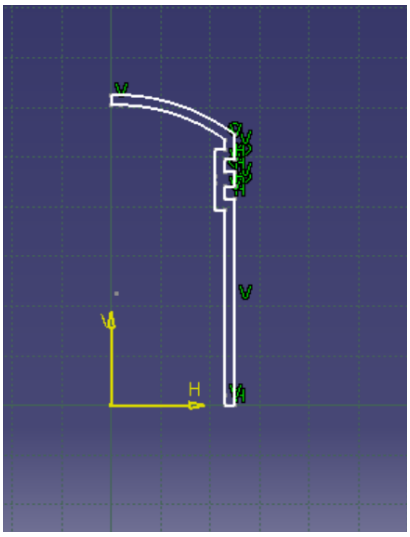


1. Chose yz plane. Drew a line vertically above from the origin of dimension 62.5 mm. Drew a line horizontally from the origin of dimension 25 mm.
2. From the end point of 25 mm line drew a line of dimension 41.5 vertically above from there.
3. From the end point obtained drew a line of dimension 2.5 mm towards left. From the end point

obtained drew a line of 2.5 mm vertically above from it.

4. Again drew a line of dimension 2.5 mm horizontally towards right from the end point obtained.
5. Drew a line of dimension 3 mm vertically above from the end point obtained.
6. Again repeated the block structure of 2.5 mm as done previously. Drew a line of dimension 5 mm from the end point obtained.
7. Used “Three point arc starting with limits” feature to draw an arc of radius 40 mm upto the end point of 62.5 mm line to get a closed curve.

8. Used the offset feature to produce offset of 2 mm inwards of the closed shape.
9. Used quick trim feature to trim unnecessary parts and produce the sketch as shown.

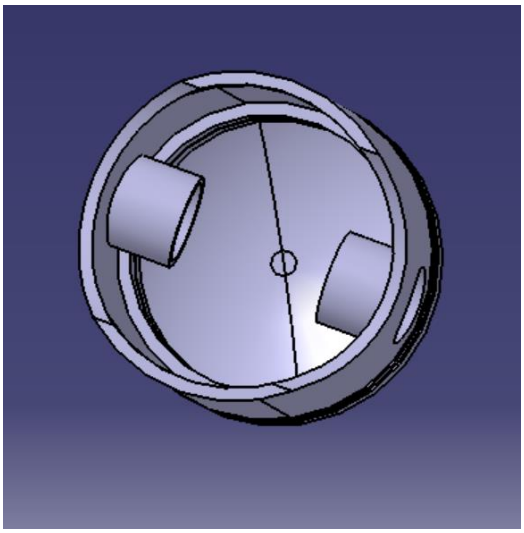


10. Using shaft feature about the y axis produced the body of the piston.
11. On the yz plane passing through origin created a segment of circle having 38 mm as length and 10 mm height with its center along y

axis. Used pocket feature to make the required cut.

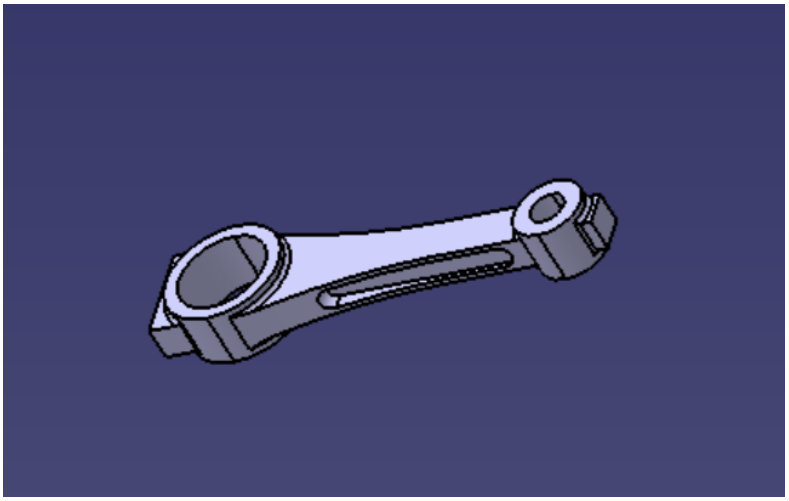
12. Again chose the yz plane and clicked on the sketch feature. Created a circle of diameter 12.5 mm with its center 30.5 mm vertically above that of origin point. Using pocket tool created the required hole in it.
13. Now created a plane at the distance 12 mm from the yz plane and parallel to it.
14. Using “project 3D elements” feature projected the circle of diameter 12.5 mm on the plane. Using the same center drew another circle of diameter 14.5 mm . Using the pad feature extruded the hollow thick cylinder upto the surface.

15. Using mirror feature created the same hollow cylinder opposite side using yz plane as mirroring element as shown below.



16. In this way piston was created.

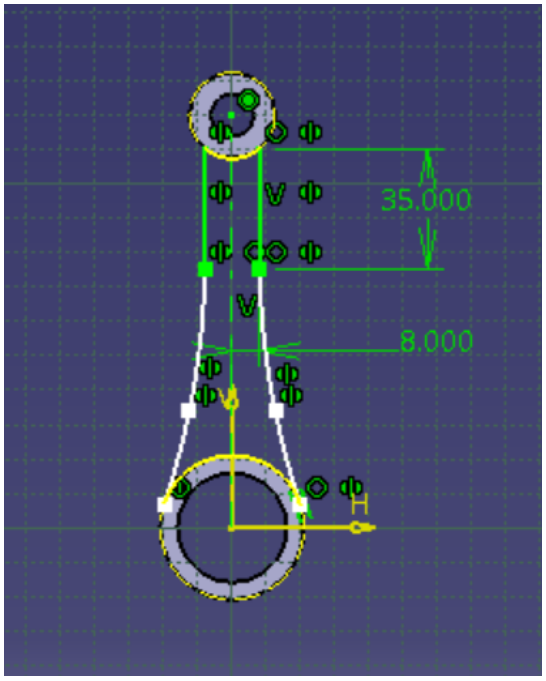
CREATING CRANCKROD



1. Chose yz plane and chose sketch feature. Created two concentric circles of diameter 32 mm and 42 mm using origin as center.

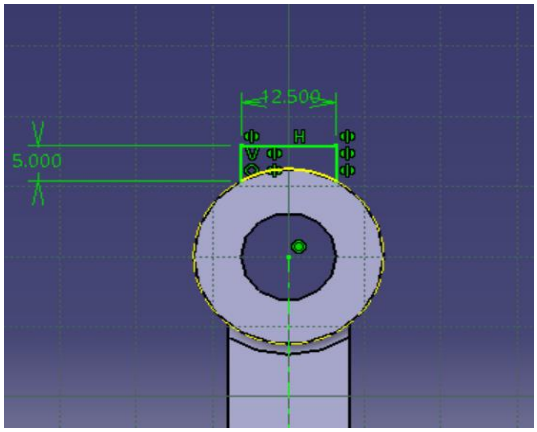
2. Also created two concentric circles of diameter 12.5 mm and 25 mm with the center of the circle at the distance 120 mm above the origin.
3. Extruded the above sketch using pad feature upto 10 mm while using mirrored extent feature.
4. Now clicked on the yz plane again and using “three point arc starting with limits” , “project 3D elements” , “quick trim” and “mirror feature” produced a

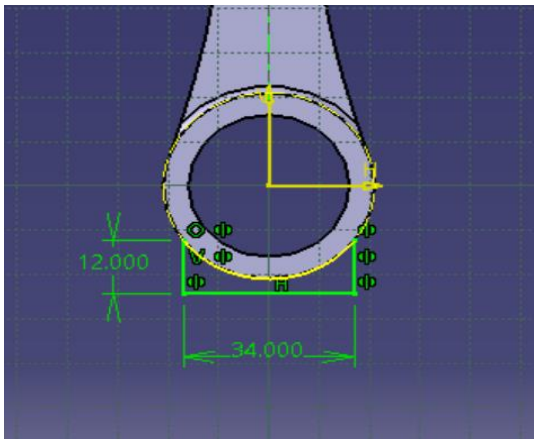
shape as shown below.



5. Using pad feature extruded the shape upto 6 mm while using mirrored extent feature.
6. Again clicked on the yz plane and used sketch feature. Using “project 3D element” , “profile feature” and “quick trim” feature

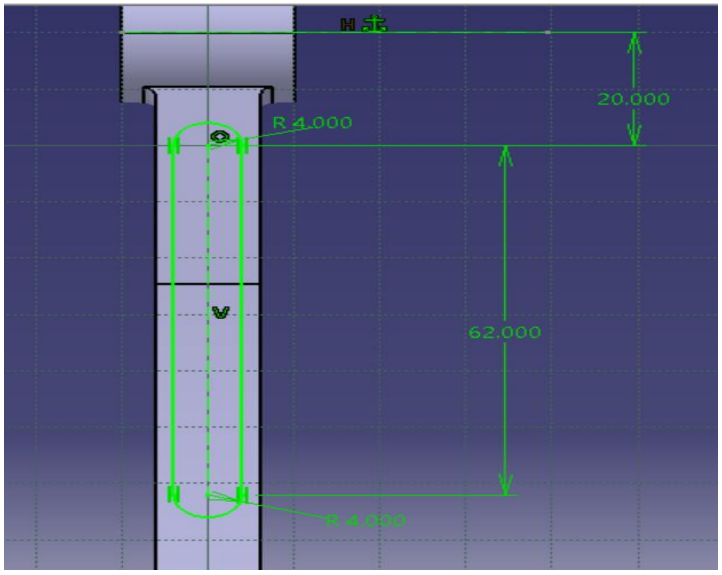
created shape as shown below.



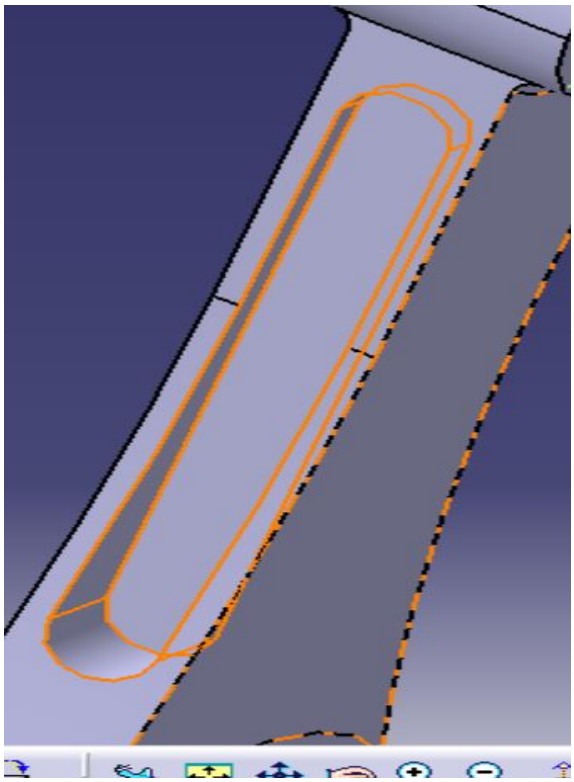


7. Using pad feature extruded the above shape upto 6mm while using mirrored extent feature.
8. Using plane feature created a plane at a distance 6 mm from zx plane and parallel to it.

9. Clicked on the obtained plane and used sketch feature. Using elongated hole profile drew a shape as given below.

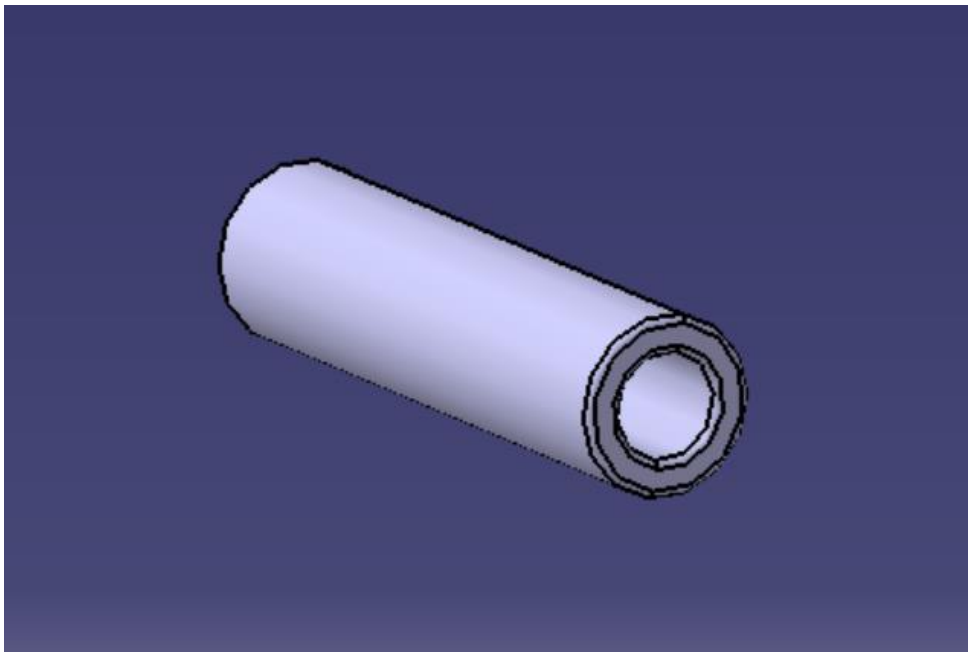


10. Using pocket feature created pocket as shown below.



11. Using mirror feature and using zx plane as mirroring element produced the same pocket on the opposite side.
12. Used edge fillet of 1.5 mm to produce a smooth cranckrod.
13. In this way cranckrod was created.

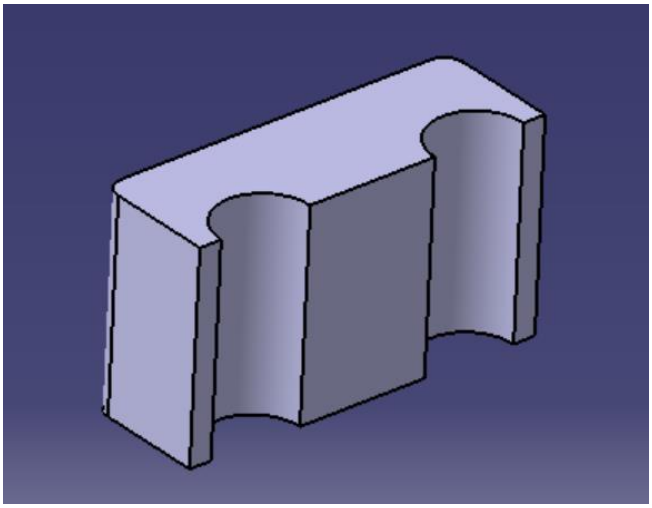
CREATING PISTONPIN



1. Chose yz plane and used sketch feature.
2. Drew two circles of diameter 12.5 mm and 7.5 mm using origin as center.
3. Using pad feature extruded the hollow thick cylinder upto 50 mm.

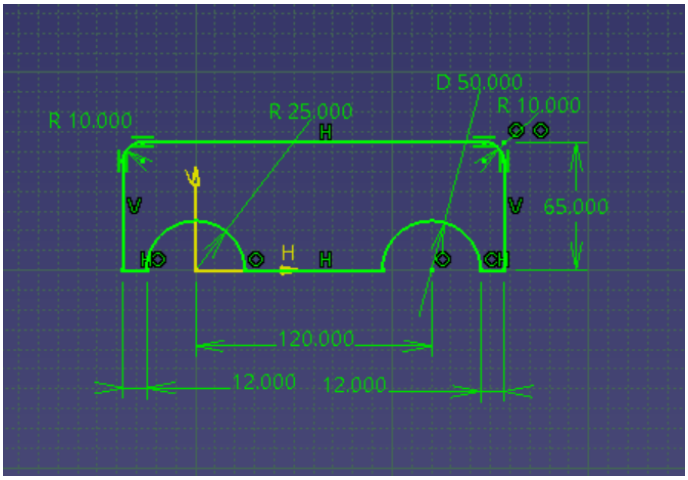
4. Used edge fillet of radius 0.5 mm to produce the complete pistonpin.
5. In this way pistonpin was created.

CREATING CYLINDER BLOCK



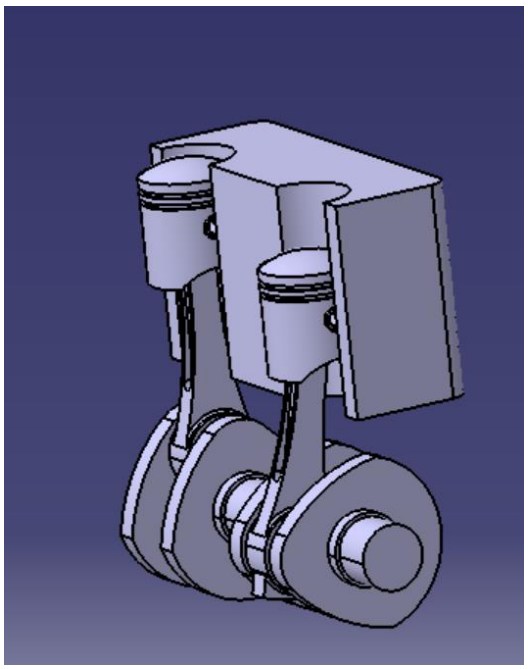
1. Chose yz plane and used sketch feature.
2. Created a circle of radius 25 mm using origin as center.
3. Created a circle of same dimension with its center at horizontal distance of 120 mm.
4. Using profile and corner feature created a sketch as

shown below.



5. Using pad feature extruded the obtained shape upto 110 mm.
6. In this way cylinder block was created.

ASSEMBLY OF TWIN CYLINDER INLINE ENGINE



1. Opened the product file and used existing component feature to bring out all the required part files to the work environment.
2. Using coincidence constraint aligned the bigger hole of first cranckrod to the respective shaft of the cranckshaft.

3. Using the offset constraint maintained cranckrod in the middle of the respective shaft.
4. Did the same thing for second cranckrod with the respective shaft of the cranckshaft.
5. Using the angle constraint maintained an angle of 180° between the cranckshaft and the first cranckrod.
6. Using the coincidence constraint aligned the smaller hole of the first cranckrod with hole of first piston.
7. Using offset constraint maintained the cranckrod at the middle of the piston.

8. Using coincidence constraint aligned the first piston pin with the piston.
9. Using line contact constraint and checking internal contact assembled the piston pin to knot respective piston and the cranckrod.
10. Using surface contact made the first piston to fix inside the cylinder block.
11. Using the offset feature made the first piston to its uppermost position.
12. Using coincidence constraint aligned second cranckrod with second piston and maintained the cranckrod in between the piston using offset constraint.

13. Using angle constrained maintained an angle of 0° between cranckshaft and cylinder block.
14. Now fixed the position of cylinder block using fix constraint.
15. Aligned the second piston pin to the second cranckrod using coincidence feature.
16. Using line contact constraint with internal contact checked assembled the second piston pin to knot second piston and cranckrod.
17. Using surface contact constraint fixed the piston inside the cylinder block.

18. In this way an assembly of twin cylinder inline engine was created.

RESULT

After following all the steps given we get a 3D model of two cylinder inline engine.