Departamento de Computação

PROJETO E MANUFATURA ASSISTIDOS POR COMPUTADOR 27260 A

AULA 04 – LAB07
FEATURE OPERATIONS

Departamento de Computação
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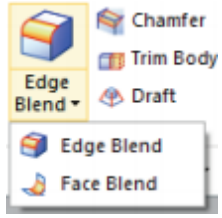
Aula 04

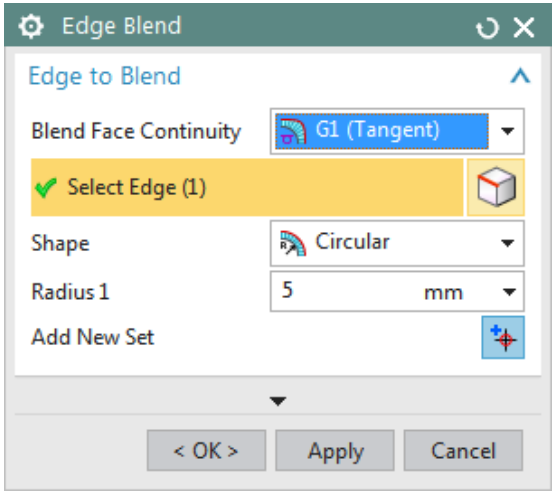
- Feature Operations are performed on the basic Form Features to smooth corners, create tapers, make threads, do instancing and unite or subtract certain solids from other solids. Some of the Feature Operations are explained below.

5.1 Edge Blend

An Edge Blend is a radius blend that is tangent to the blended faces. This feature modifies a solid body by rounding selected edges. This command can be found under **Insert →Detail Feature →Edge Blend.**

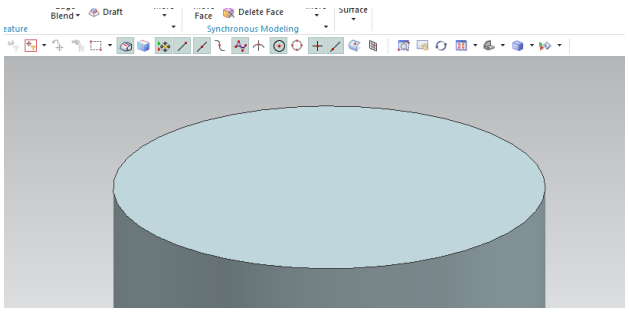


You can also click on its icon in the Feature Group. You need to select the edges to be blended and define the Radius of the Blend as shown below.

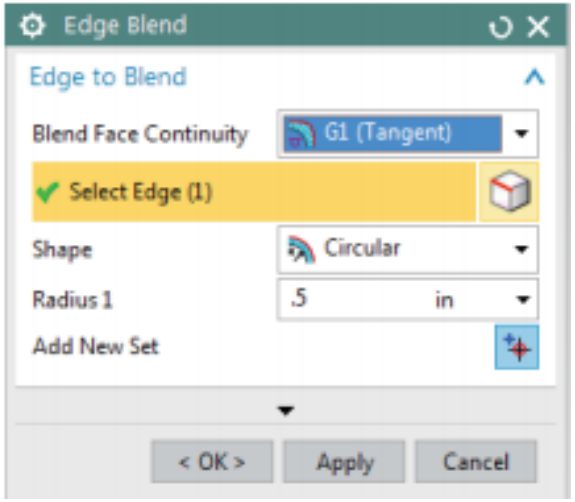


5.1 Edge Blend

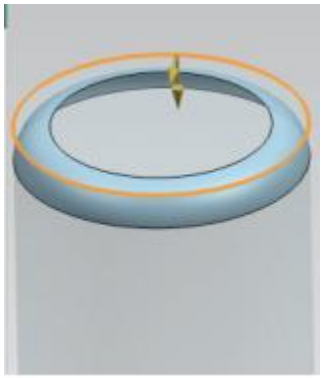
1. Choose **Insert →Design Feature →Cylinder**



2. **Insert →Edge Blend**.



5.1 Edge Blend



5.2 Chamfer

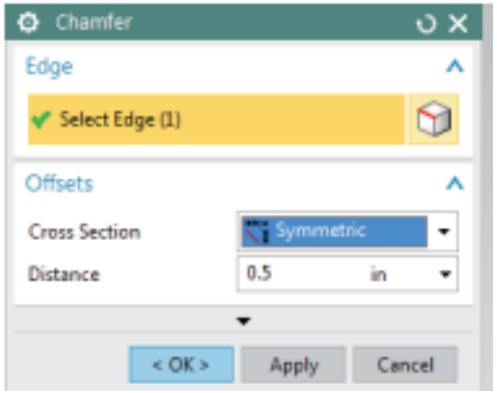
The Chamfer Function operates very similarly to the Blend Function by adding or subtracting material relative to whether the edge is an outside chamfer or an inside chamfer. This command can be found under **Insert →Detail Feature →Chamfer.**

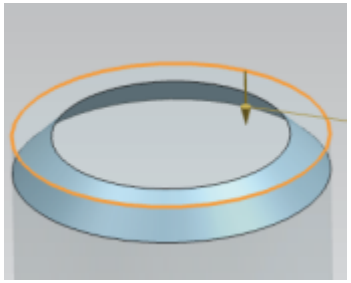
You can also click on its icon in the Feature Group. You need to select the edges to be chamfered and define the Distance of the Chamfer as shown below.

5.2 Chamfer

3. Choose Choose **Insert →Design Feature →Cylinder**

4**. Insert → Chanfer**



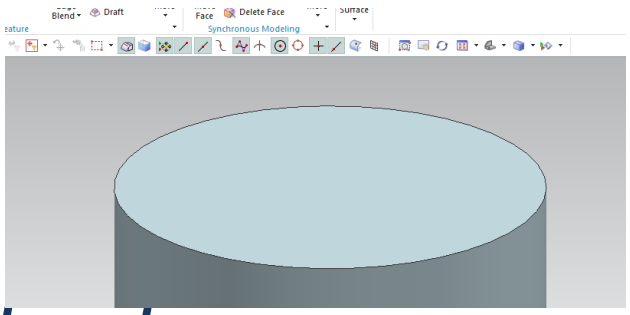


5.3 Thread

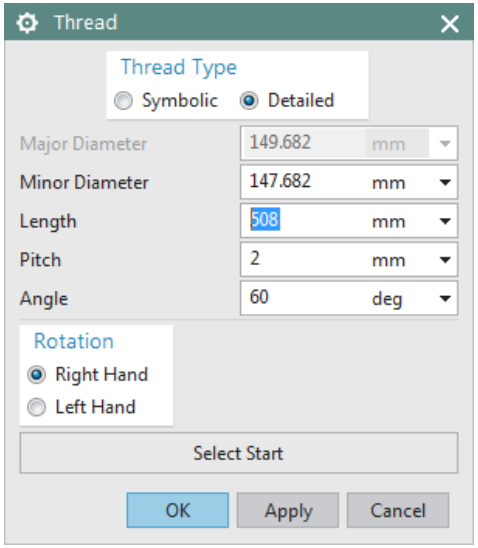
Threads can only be created on cylindrical faces. The *Thread Function* lets you create *Symbolic* or *Detailed* threads (on solid bodies) that are right or left handed, external or internal, on cylindrical faces such as *Holes*, *Bosses*, or *Cylinders*. It also lets you select the method of creating the threads such as cut, rolled, milled or ground. You can create different types of threads such as metric, unified, acme and so on. To use this command, go to *Insert →Design Feature →Thread*.

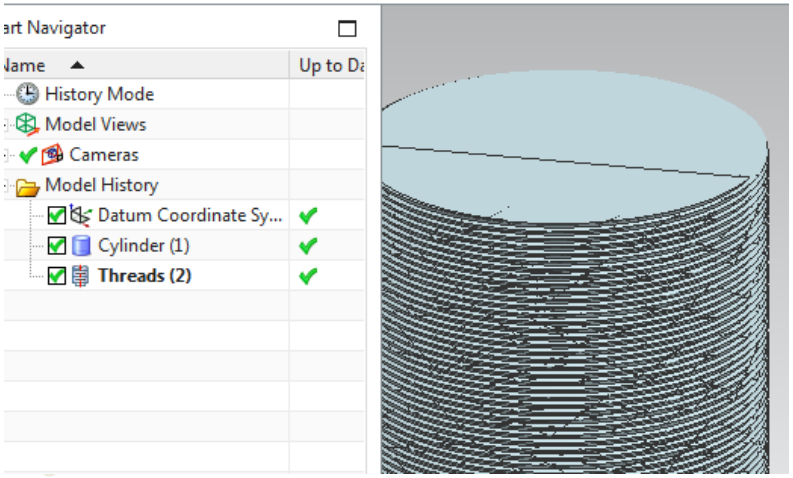
5.3 Thread

5. Choose **Insert →Design Feature →Cylinder**



5. ***Insert →Design Feature →Thread***.

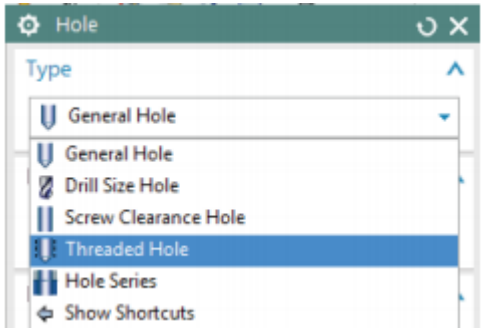




5.3 Thread

For Threaded Holes, it is recommended to use the Threaded Hole command instead of the Thread command:

7. **Insert →Design Feature →Hole**

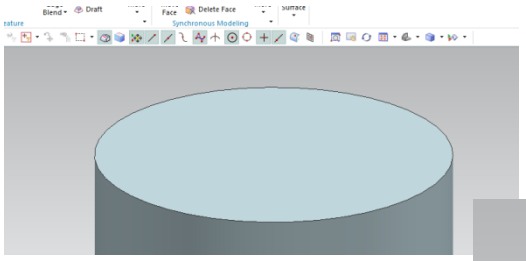


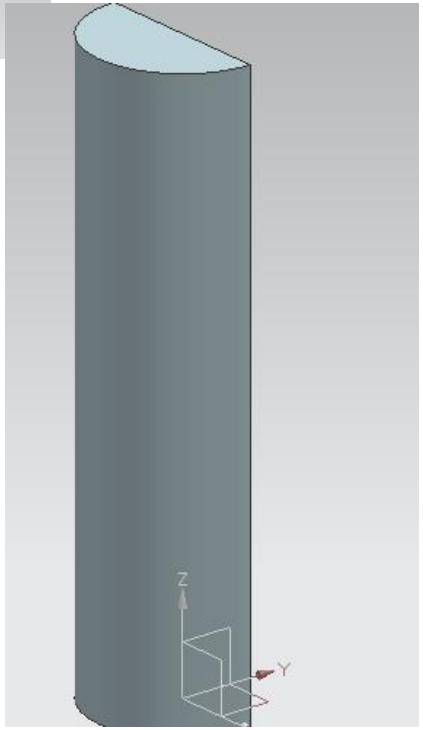
5.4 Trim Body

A solid body can be trimmed by a Sheet Body or a Datum Plane. You can use the Trim Body function to trim a solid body with a sheet body and at the same time retain parameters and associativity.

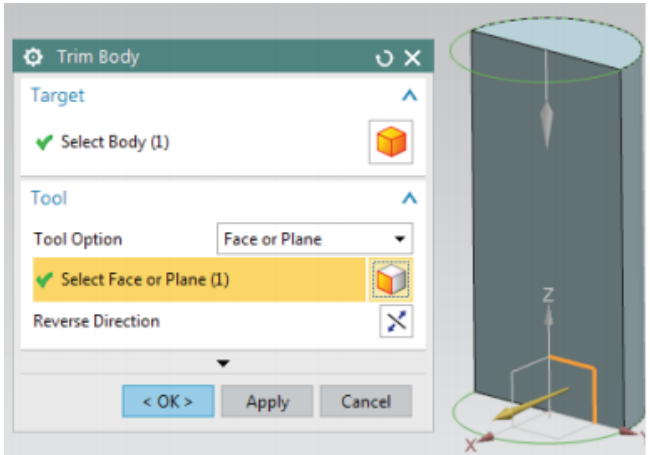
5.4 Trim Body

8. Choose **Insert →Design Feature →Cylinder**





9. ***Insert →Trim → Trim Body*** or click on its icon in the Feature Group.

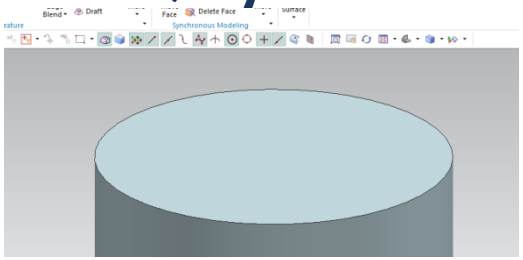


5.5 Split Body

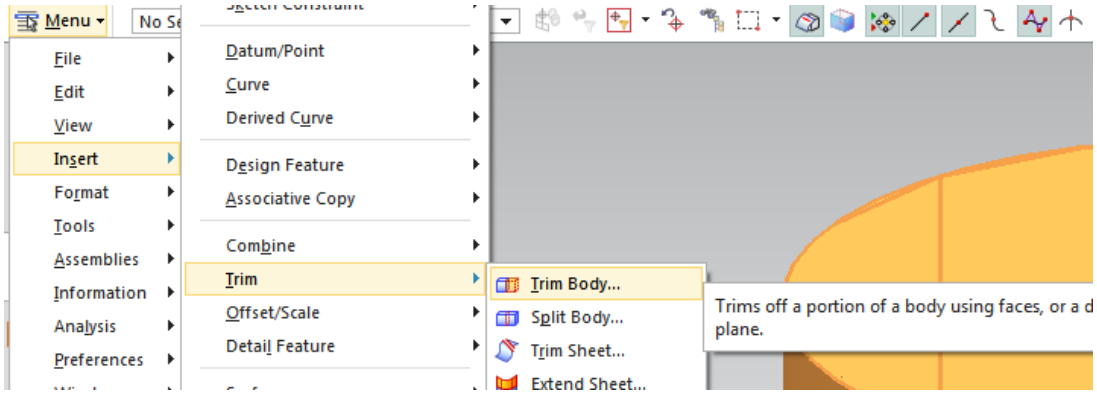
A solid body can be split into two similar to trimming it. It can be done by a plane or a sheet body.

5.5 Split Body

10. Choose **Insert →Design Feature →Cylinder**



11. ***Insert →Trim →Split Body*** *or click on its icon in the* ***Feature Group****.*



5.5 Split Body

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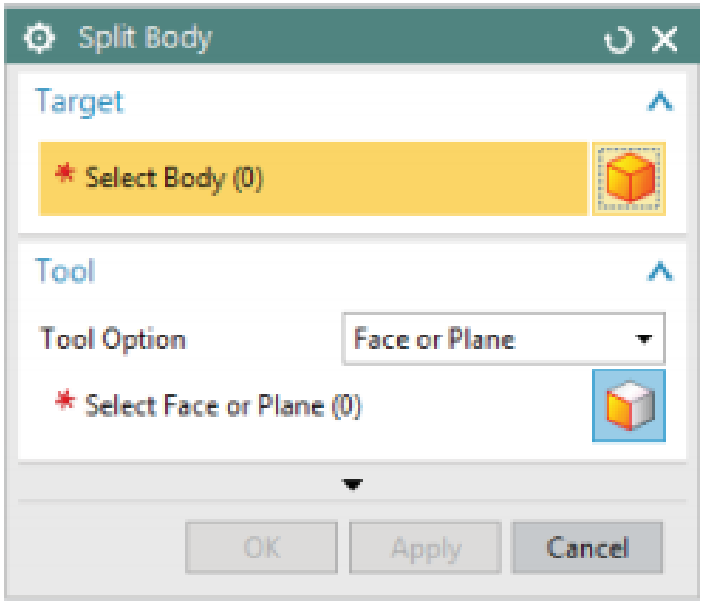
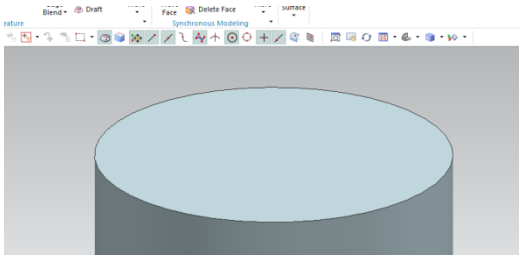


5.6 Mirror

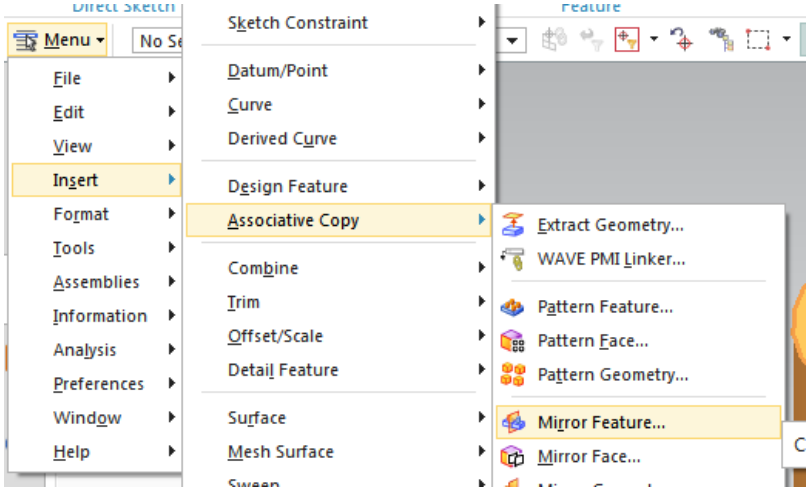
Mirror is a type of Associative Copy in which a solid body is created by mirroring the body with respect to a plane.

5.6 Mirror

12. Choose **Insert →Design Feature →Cylinder**



13. ***Insert →Associative Copy →Mirror Feature*** *or click on its icon in the* ***Feature Group****.*



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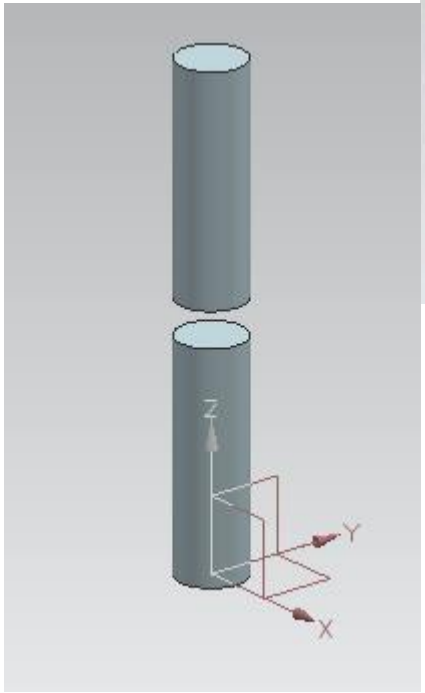
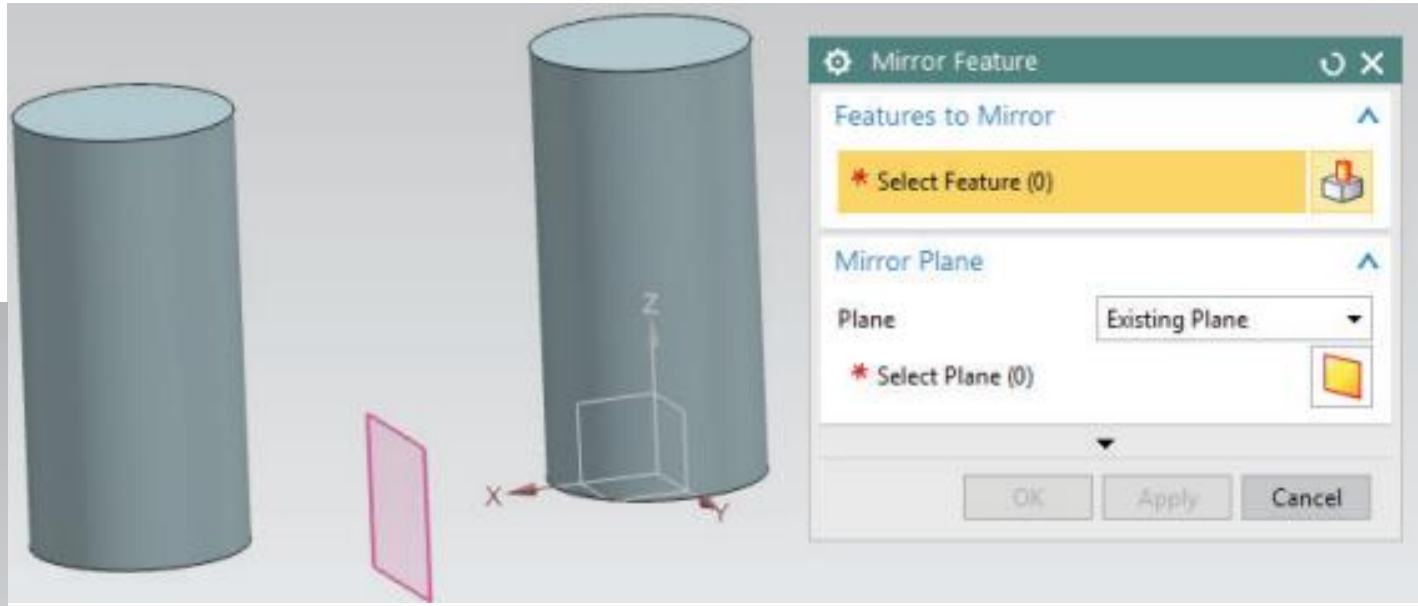
5.6 Mirror





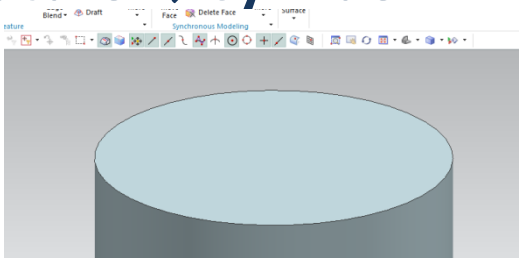
5.7 Pattern

A Design Feature or a Detail Feature can be made into dependent copies in the form of an Array.

It can be Linear, Circular, Polygon, Spiral, etc. This particularly helpful feature saves plenty of time and modeling when you have similar features. For example threads of a gear or holes on a mounting plate, etc.

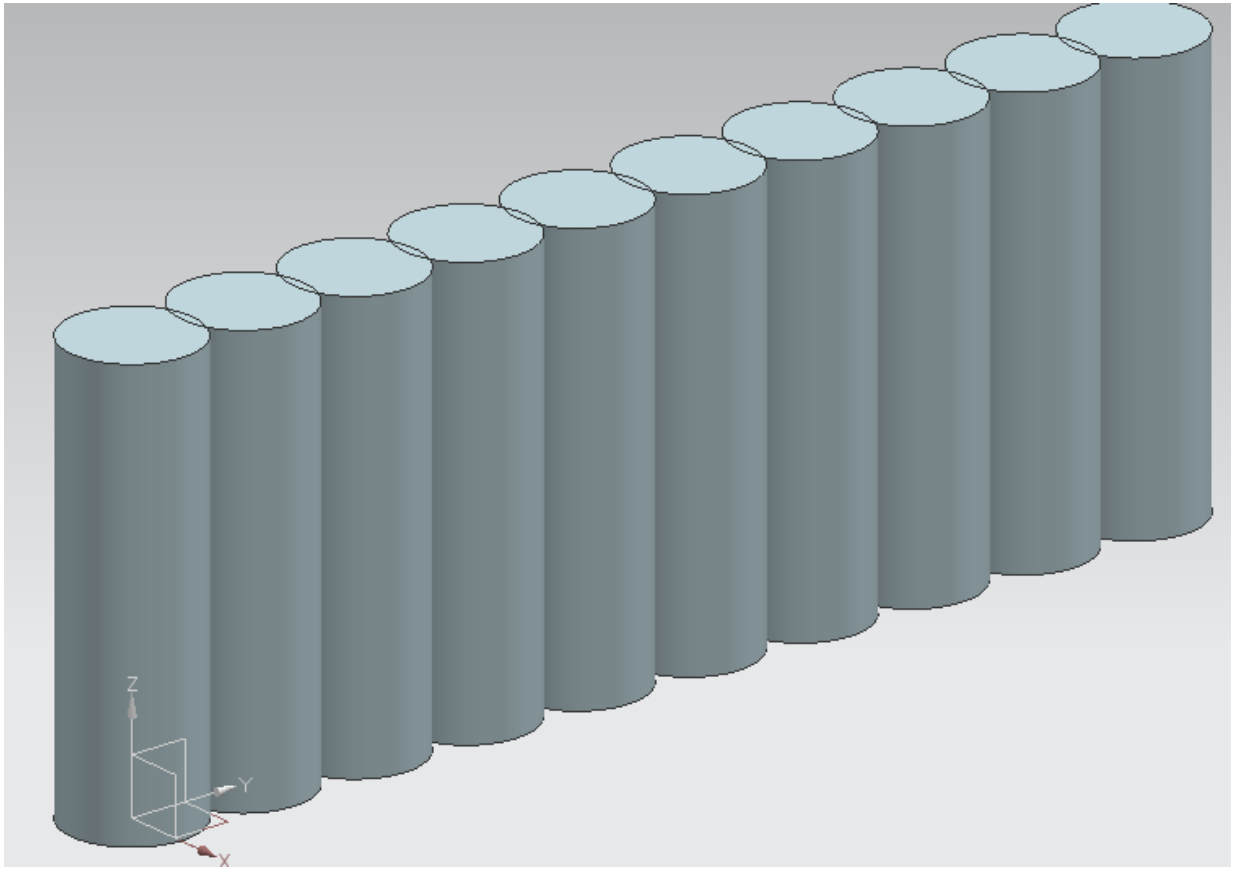
5.7 Pattern

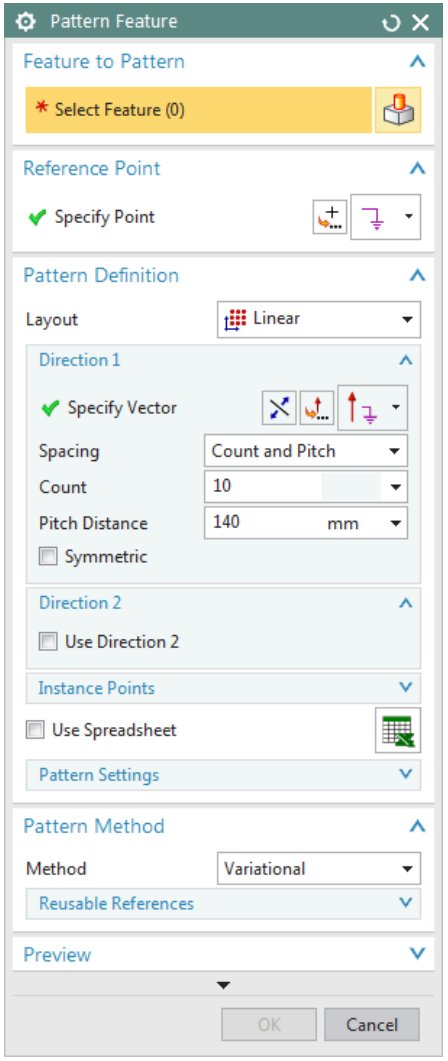
14. Choose **Insert →Design Feature →Cylinder**



15. ***Insert →Associative Copy →Pattern Feature*** *or click on its icon in the* ***Feature Group****.*

5.7 Pattern


Z
Y
X

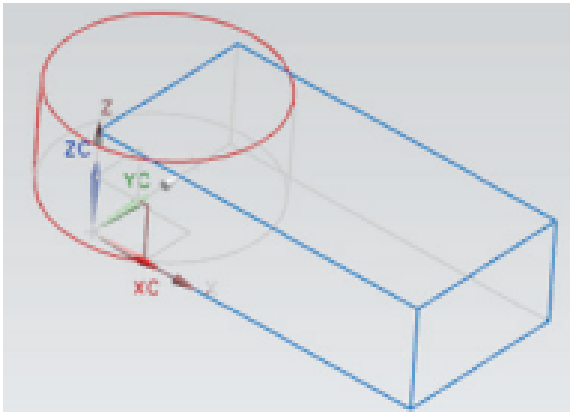

Pattern Feature
Feature to Pattern
Select Feature (0)
Reference Point
Specify Point
Pattern Definition
Layout
Linear
Direction 1
Specify Vector
Spacing
Count and Pitch
Count
10
Pitch Distance
140
mm
Symmetric
Direction 2
Use Direction 2
Instance Points
Use Spreadsheet
Pattern Settings
Pattern Method
Method
Variational
Reusable References
Preview
OK
Cancel

5.8 Boolean Operations

There are three types of *Boolean Operations: Unite, Subtract,* and *Intersect*. These options can be used when two or more solid bodies share the same model space in the part file. To use this command, go to *Insert* → *Combine* or click on their icons in the *Feature Group.*

5.8 Boolean Operations

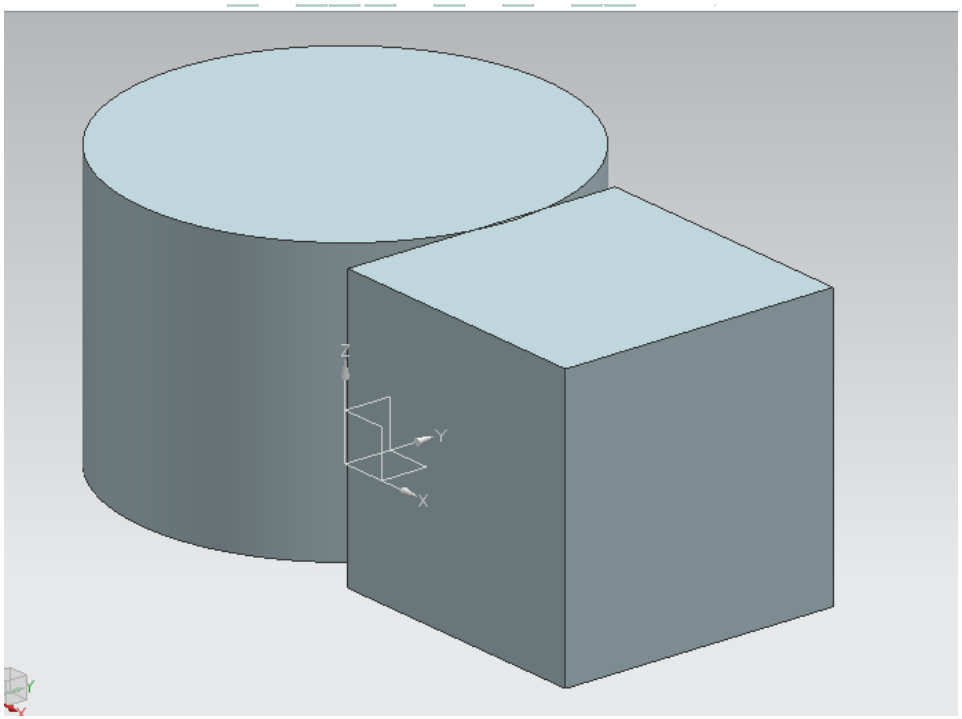
There are three types of *Boolean Operations: Unite, Subtract,* and *Intersect*. These options can be used when two or more solid bodies share the same model space in the part file. To use this command.



5.8 Boolean Operations

15. Choose **Insert →Design Feature →Cylinder**

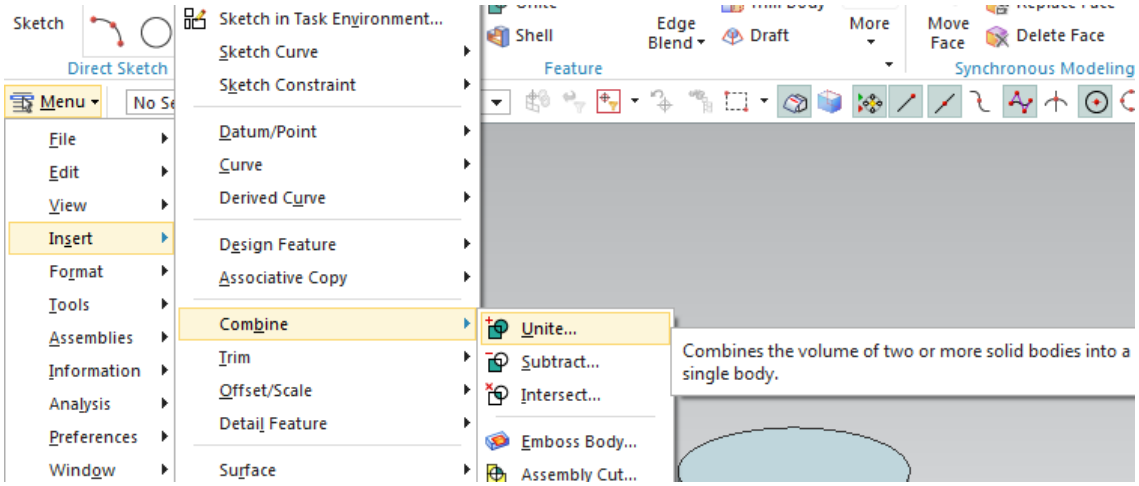
17. Choose **Insert →Design Feature → Block**



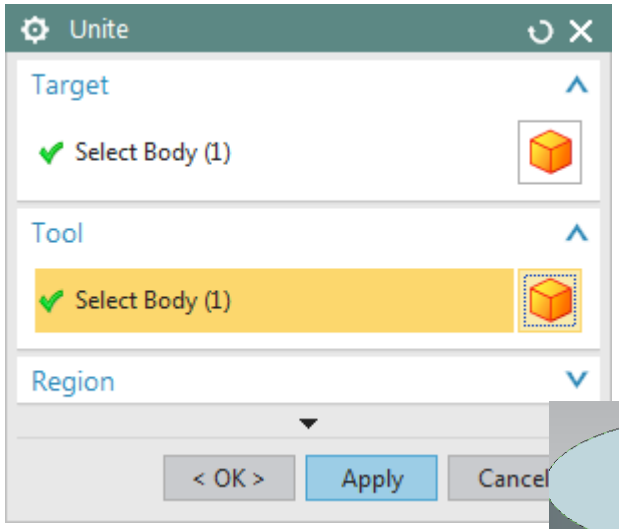
5.8.1 Unite

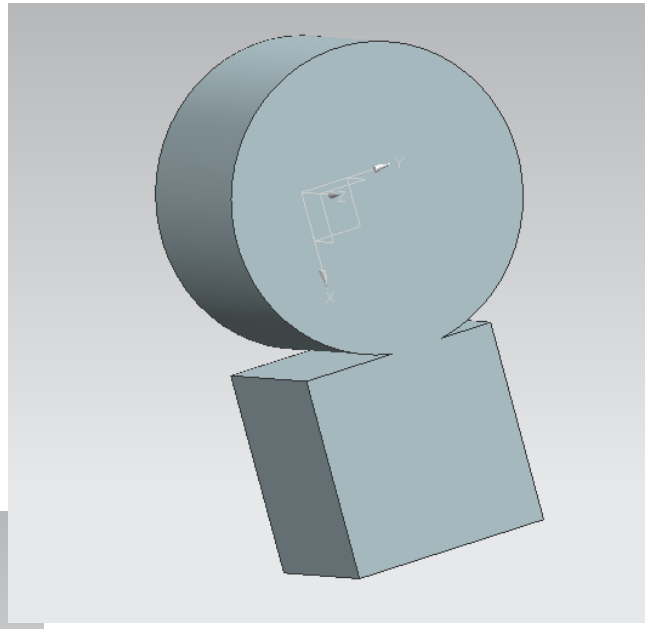
The unite command adds the Tool body with the Target body. For the above example, the output will be as follows if Unite option is used.

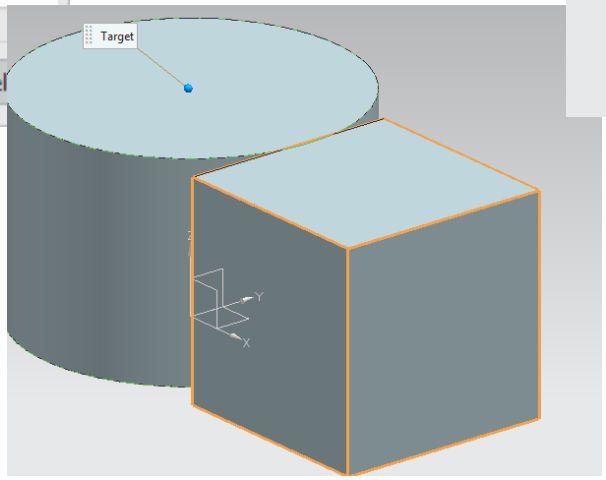
18. Choose **Insert →Combine→ Unite**



5.8.1 Unite


Unite
Target
Select Body (1)
Tool
Select Body (1)
Region
< OK >
Apply
Cancel

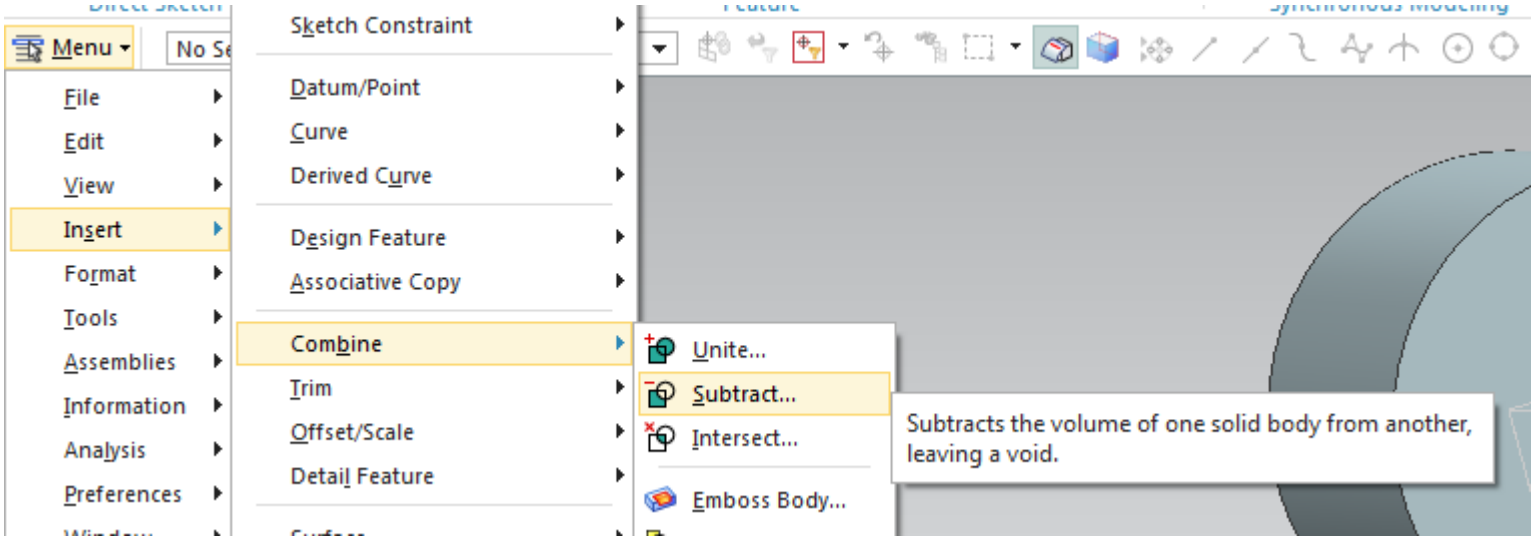



Target

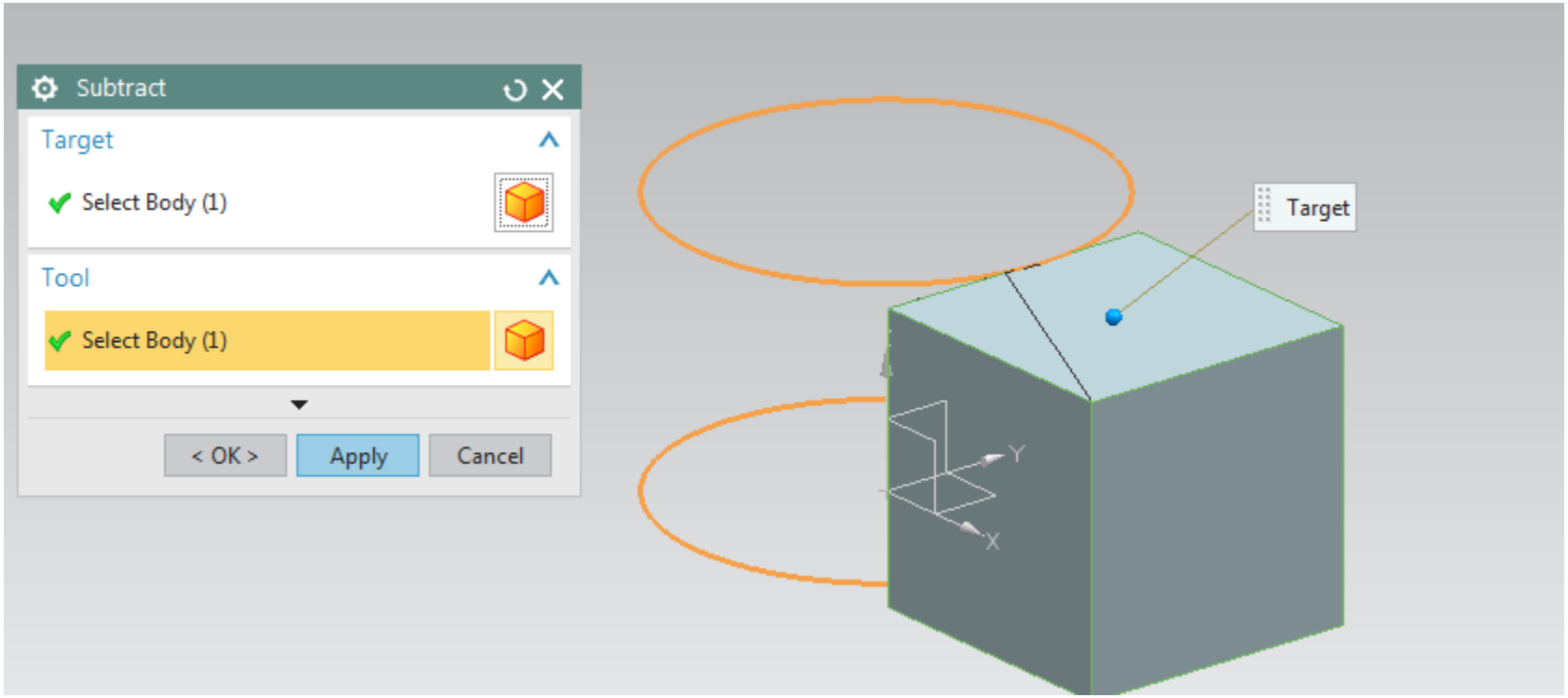
5.8.2 Subtract

When using the subtract option, the Tool Body is subtracted from the Target Body. The following would be the output if the Block is used as the Target and the Cylinder as the Tool.

19. Choose **Insert →Combine→ Subtract**



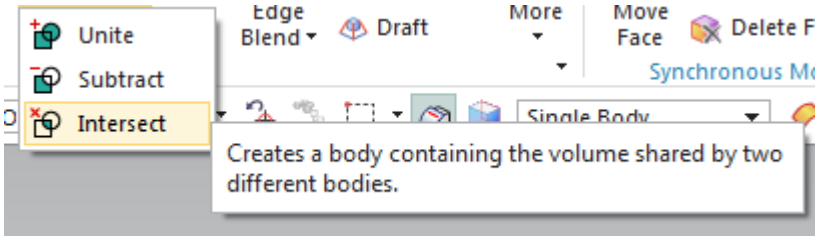
5.8.2 Subtract

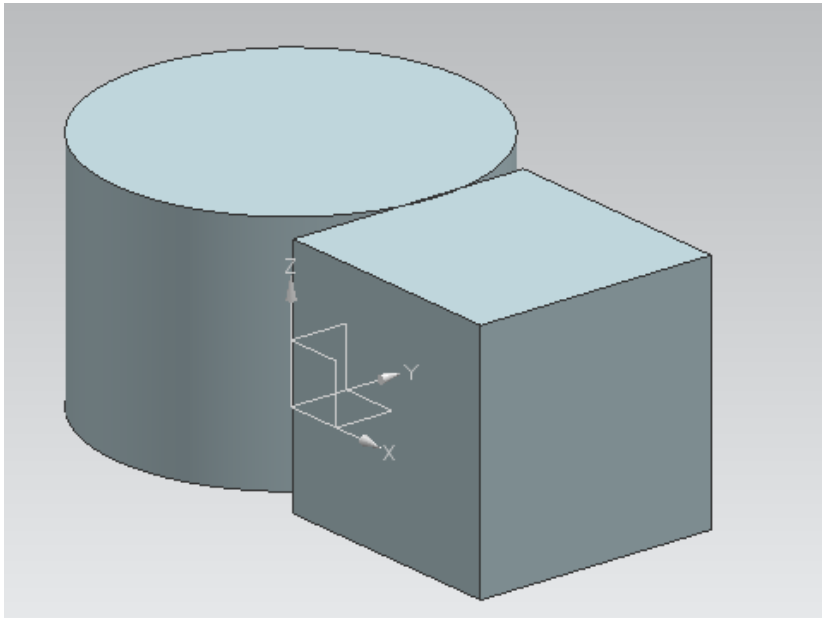


5.8.3 Intersect

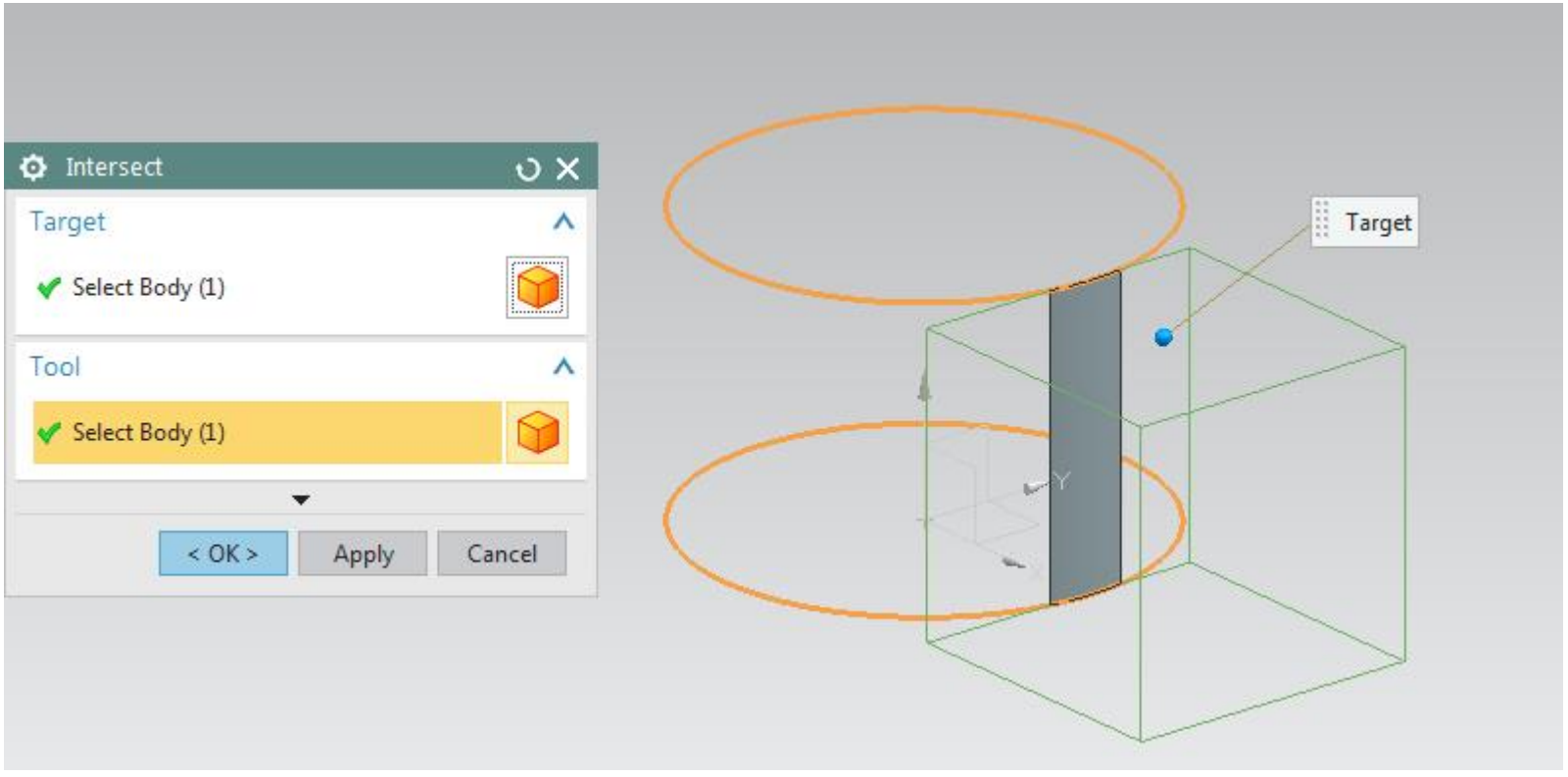
This command leaves the volume that is common to both the *Target Body* and the *Tool Body*.

20. Choose **Insert →Combine→ Intersect**





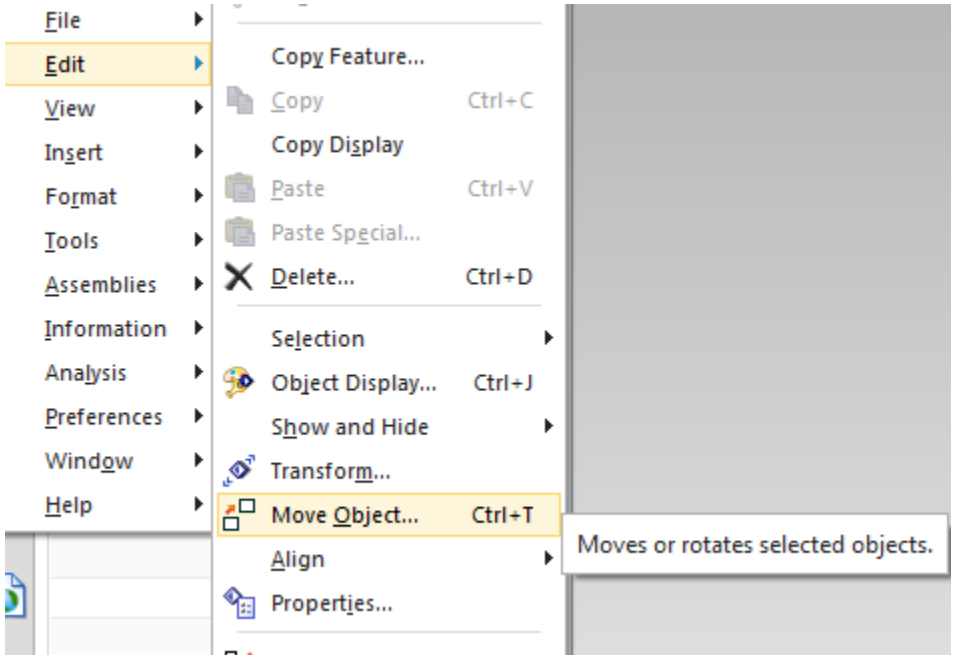
5.8.3 Intersect

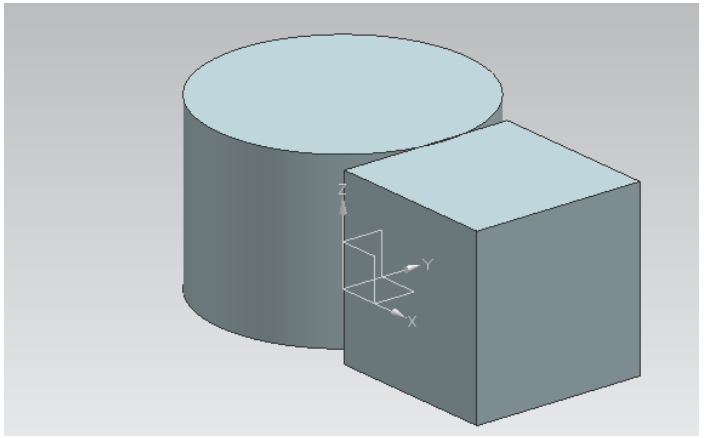


5.9 Move

If you want to Move an object with respect to a fixed entity.

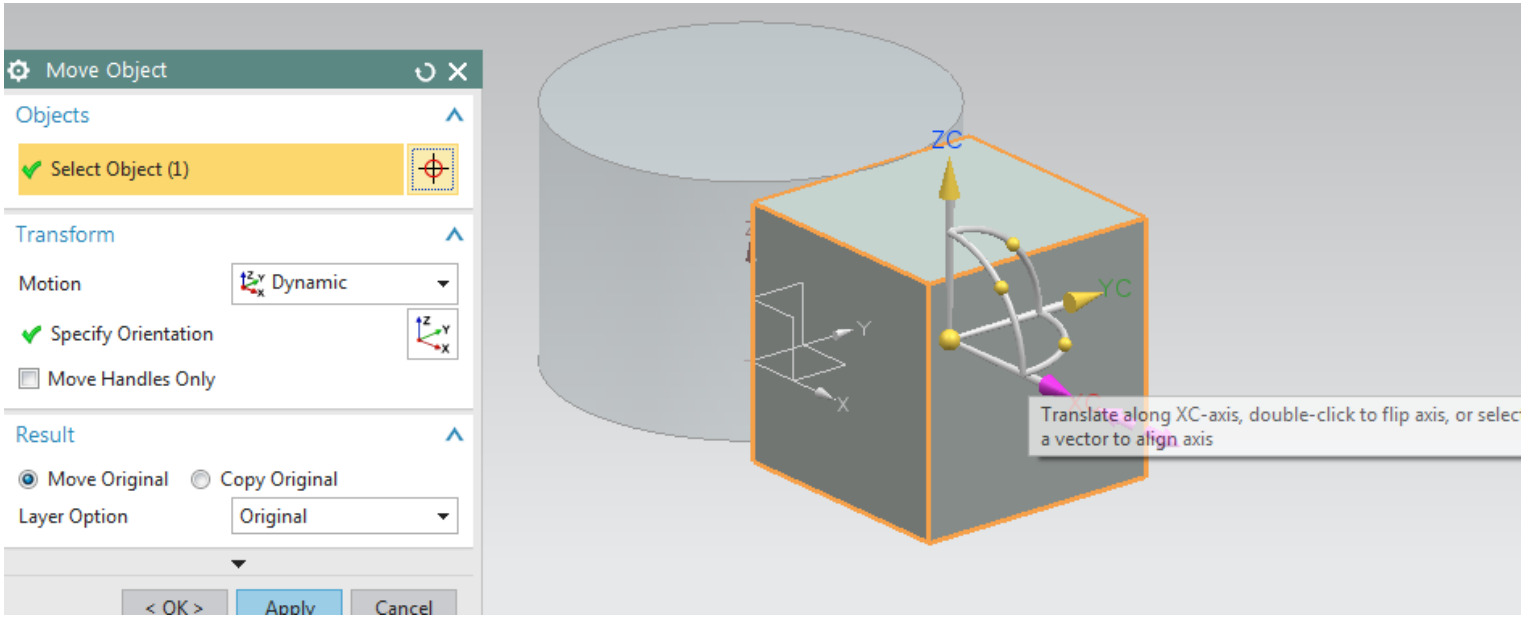
21. Click on **Edit →Move Object**





5.9 Move

You can select the type of motion from the Motion drop-down menu. The default option is Dynamic. With this you can move the object in any direction. There are several other ways of moving the object.



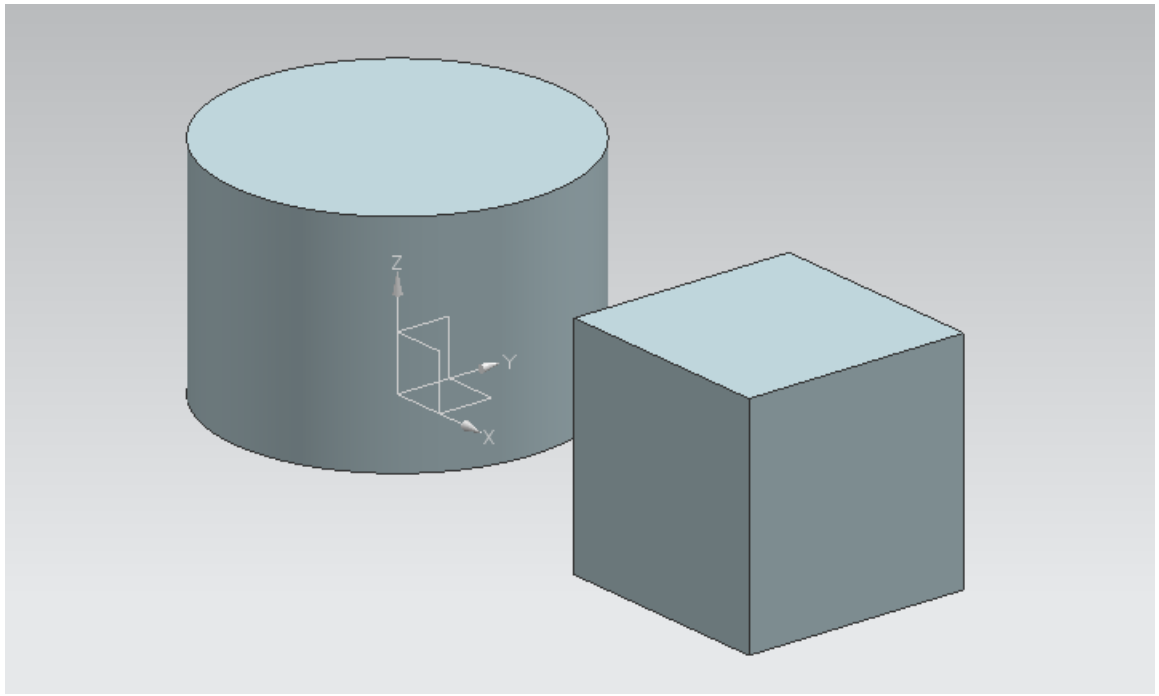
5.9 Move

If you choose Distance you can move the selected object in the X-Y-Z direction by the distance that you enter.

22. Click on **Specify Vector** and select the direction.
23. Type **5** in the **Distance** box. This will translate the cylinder a distance of 5 inches along X-Axis
24. Click **OK**

5.9 Move

As you can see, we have moved the cylinder in the X-direction. Similarly, we can also copy the cylinder by a specified distance or to a specified location by selecting the Copy Original option in the Result.



5.10 Hexagonal Screw

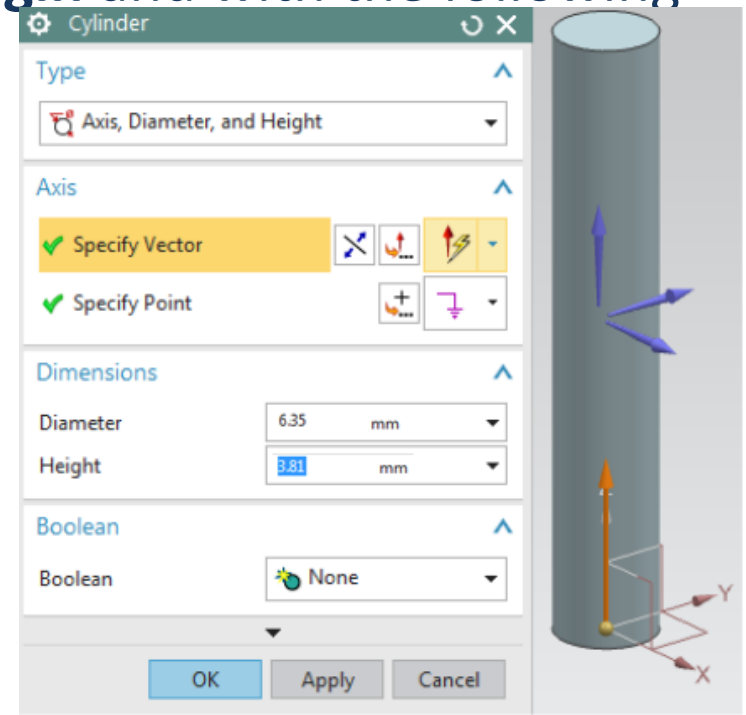
25.Create a new file and save it as **Impeller_hexa-bolt.prt**

25. Choose **Insert** →**Design Feature** → **Cylinder**

27. The cylinder should be pointing in the **Positive ZC-Direction** with the center set at the **Origin** and with the following dimensions:

Diameter = **6.35 mm**

Height = **3.81 mm**



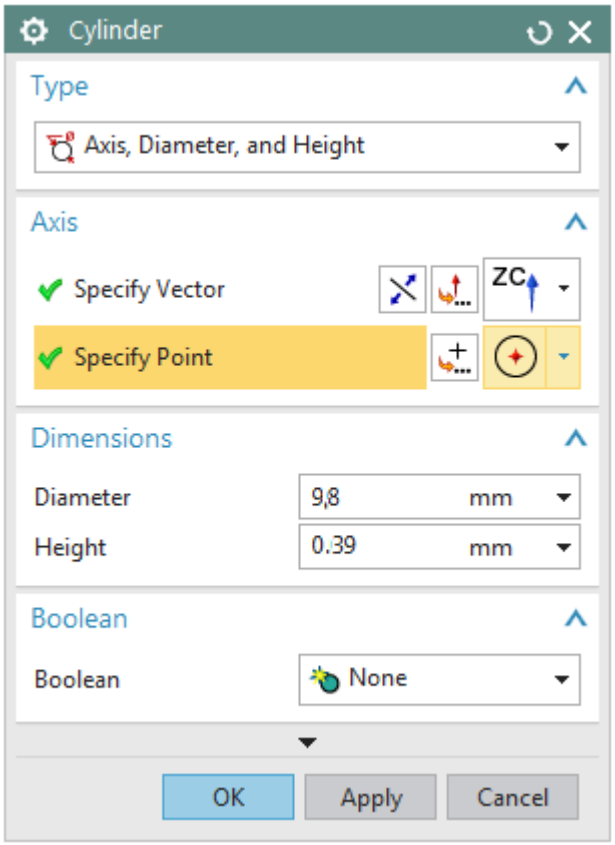
5.10 Hexagonal Screw

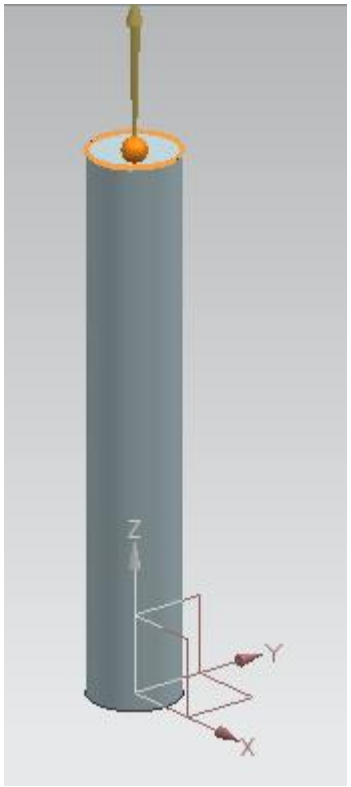
Now create a small step cylinder on top of the existing cylinder.

28. Create a **Cylinder** with the following dimensions:

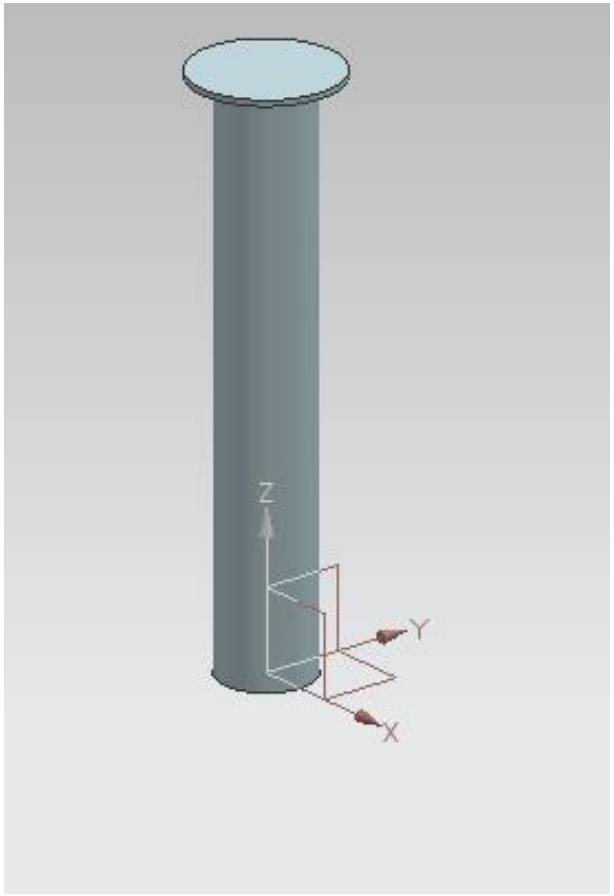
Diameter = **9.82 mm**

Height = **0.396 mm**



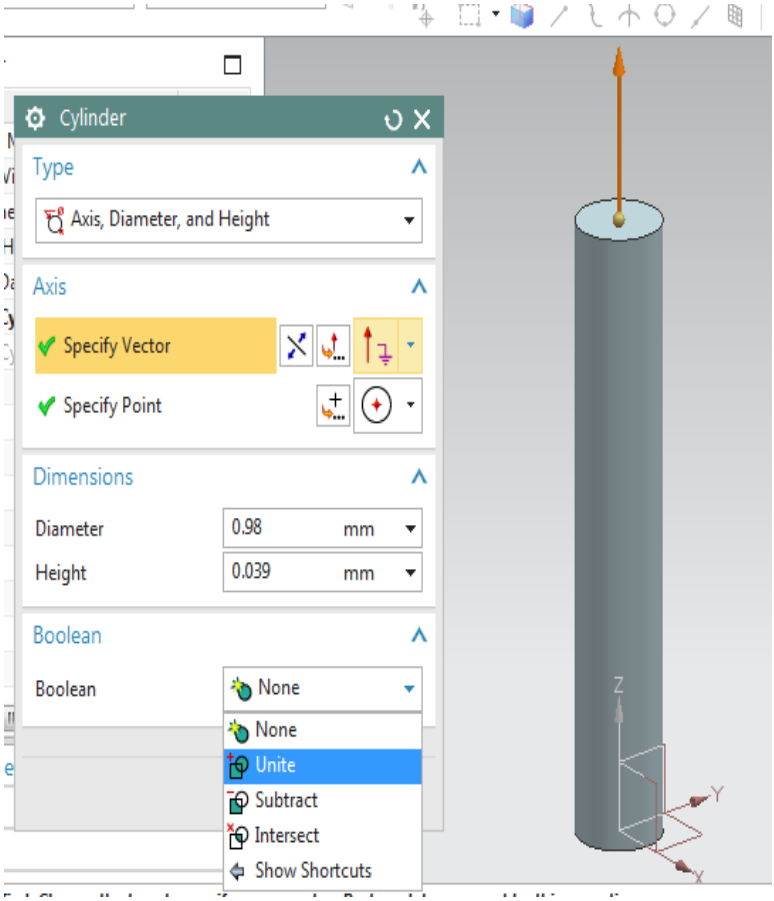


5.10 Hexagonal Screw



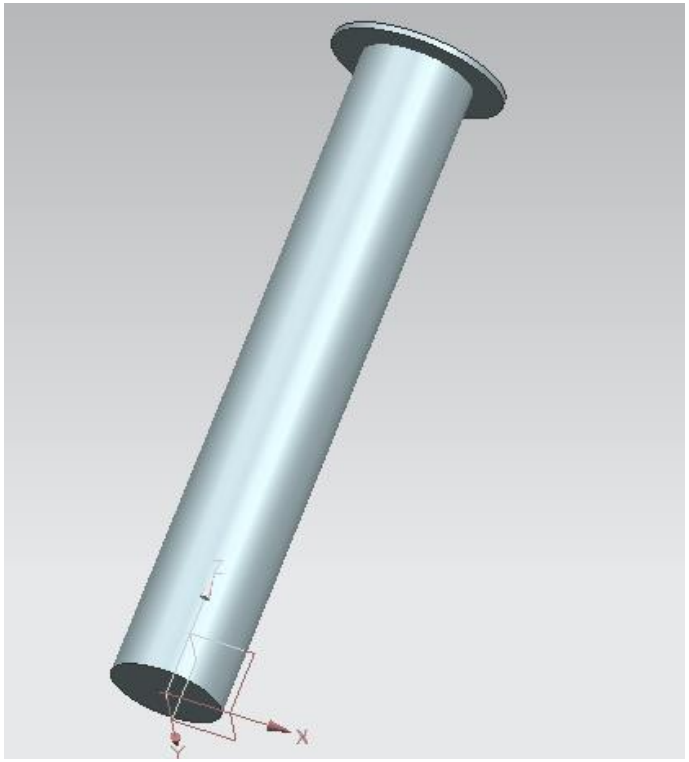
5.10 Hexagonal Screw

29. Click on the top face of the existing cylinder
30. On the **Point Constructor** window, choose the **Arc/Ellipse/Sphere Center** icon from the drop-down **Type** menu
31. Click **OK** to close the **Point Constructor** window
32. Under the **Boolean** drop-down menu, choose **Unite**



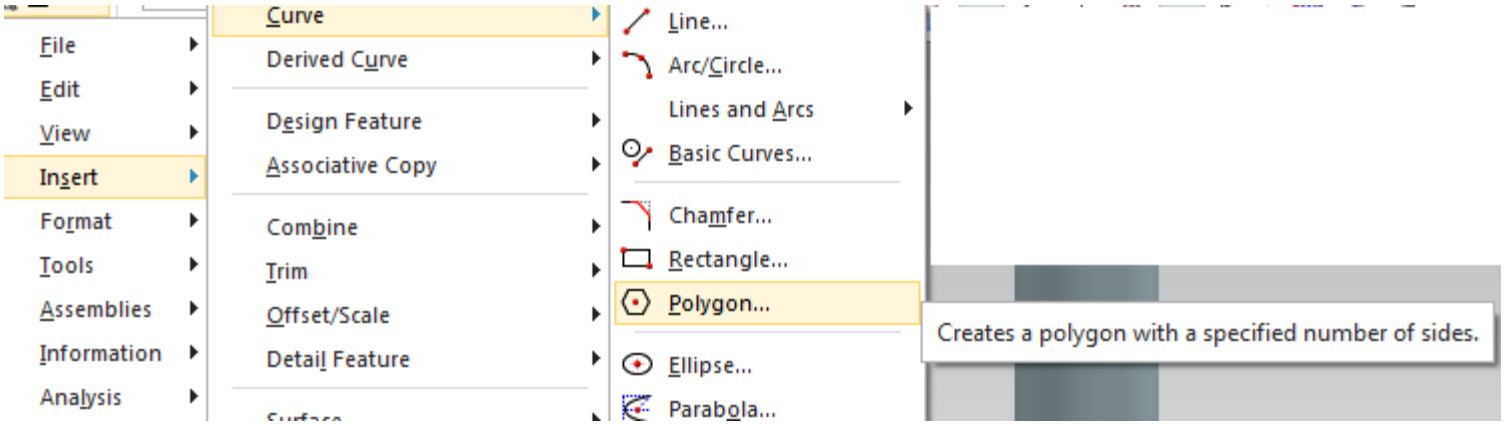
5.10 Hexagonal Screw

The two cylinders should look like the figure shown on the right.



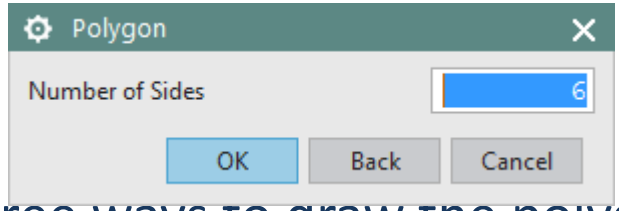
5.10 Hexagonal Screw

33. Choose **Insert →Curve →Polygon**



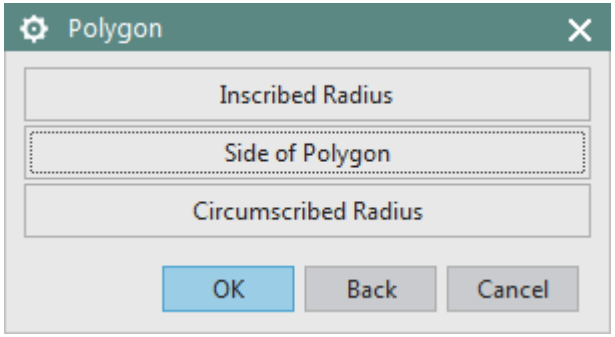
5.10 Hexagonal Screw

34. On the **Sides** window, type **6** for the **Number of Sides**



There are three ways to draw the polygon.

- *Inscribed Radius*
- *Circumscribed Radius*
- *Side of Polygon*



5.10 Hexagonal Screw

35. Choose **Side of Polygon** and enter the following dimensions:

Length = **6.2484 mm**

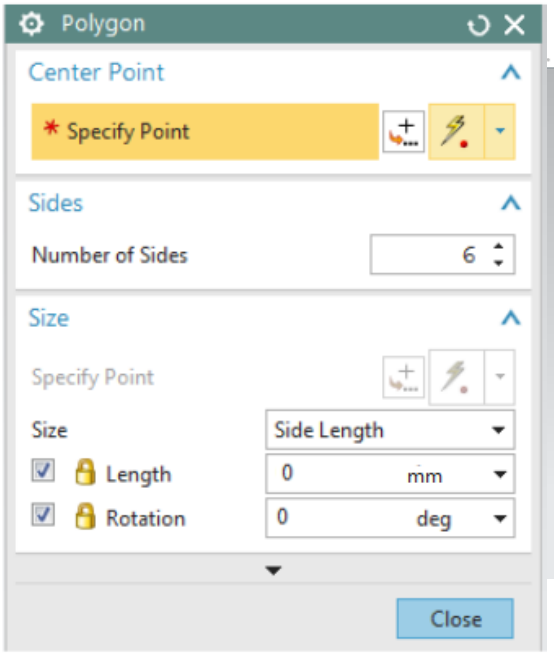
Rotation = **0.00** degree

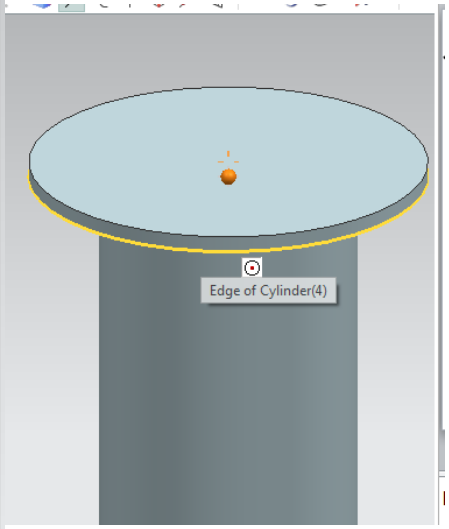
35. Click **OK**

37. In **Point Location**,

choose the Edge of Cylinder

38. Click **OK**

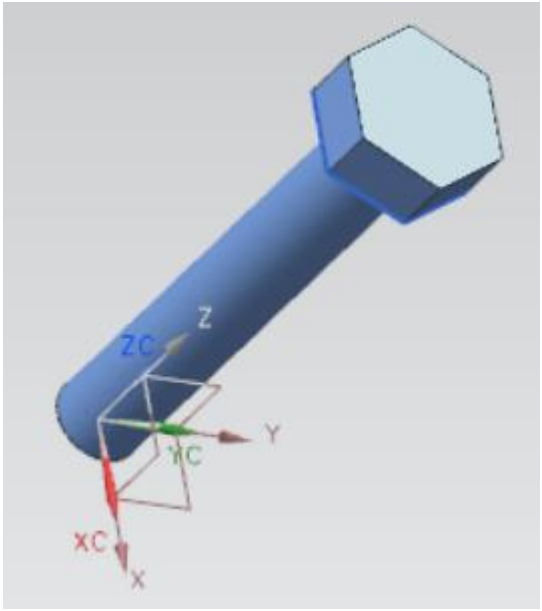




5.10 Hexagonal Screw

Now we will extrude this polygon.

39. Choose **Insert** →**Design Feature** →**Extrude**
40. Choose the **Hexagon** to be extruded
41. Enter the **End Distance** as 4.76504



5.10 Hexagonal Screw

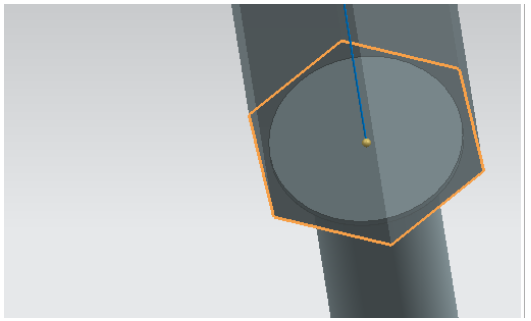
42. On top of the cylinder that has a diameter Of 9.8298 mm, insert another cylinder with the following dimensions.

Diameter = 9.8298 mm

Height = 4.7625 mm

You will only be able to see this cylinder when the model is in Static Wireframe since the cylinder is inside the hexagon head.

The model will look like the following.



5.10 Hexagonal Screw

43. We will now use the feature operation Intersect.

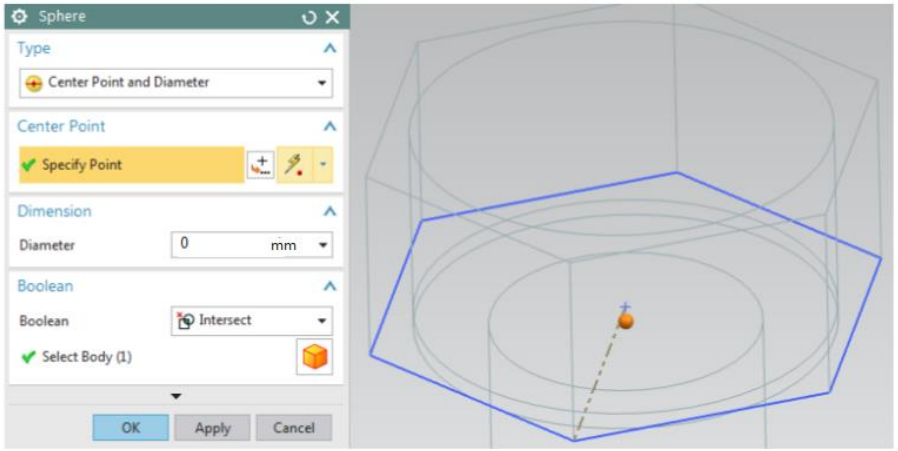
44. Choose Insert →Design Feature →Sphere

45. Choose Center Point and Diameter

46. Select the bottom of the last cylinder drawn (which is inside the hexagon head and has a diameter of 9.8298 mm and a height of 4,7625mm)

47. Give **13.97** as the **Diameter**

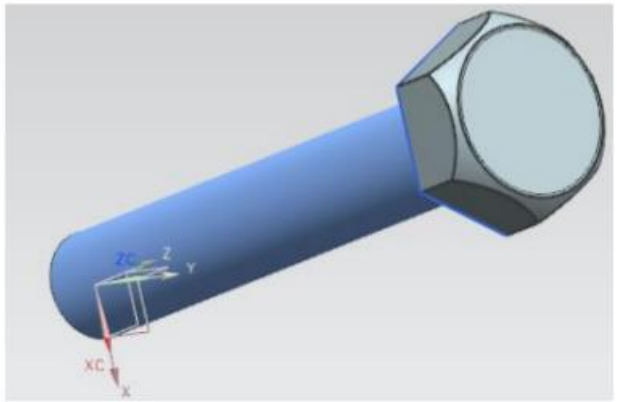
48. Choose **Intersect** in the Boolean dialog box



5.10 Hexagonal Screw

49. It will ask you to select the Target Solid
50. Choose the hexagonal head
51. Click **OK**

This will give you the hexagonal bolt.

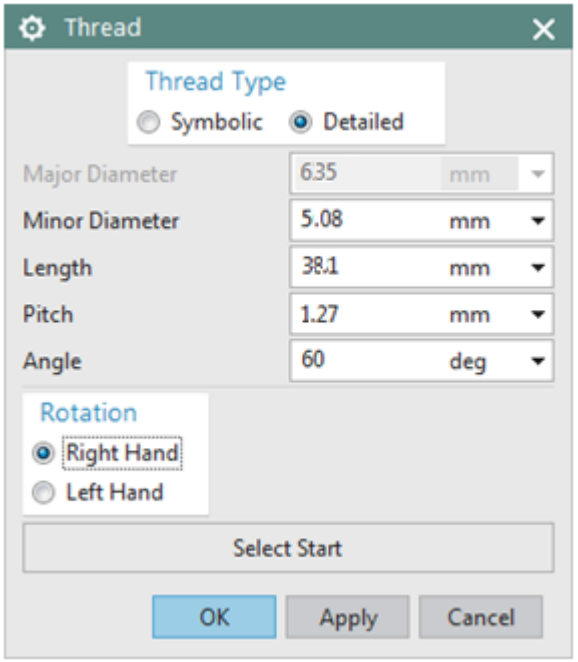


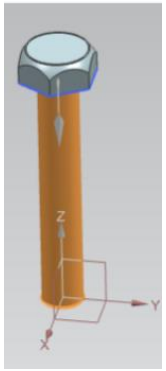
Note: This blend feature on the bolt hat can be created also by revolving cut with a section about its axis, you can try it out.

5.10 Hexagonal Screw

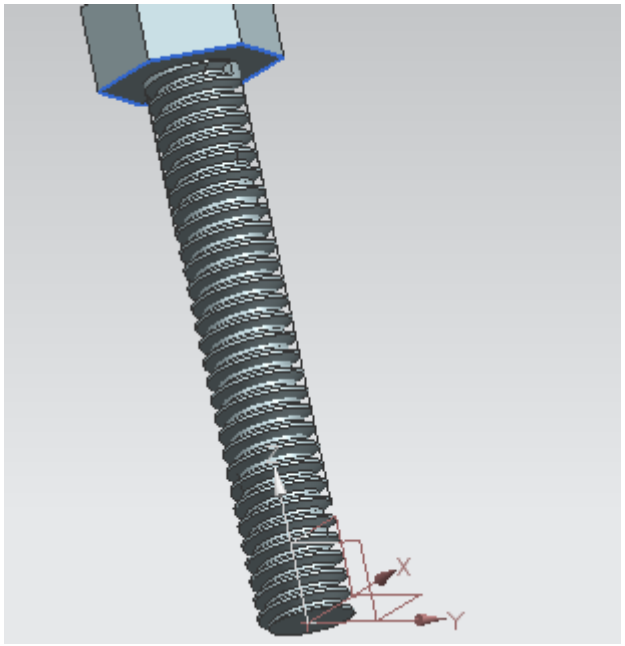
Now we will add *Threading* to the hexagonal bolt.

52. Choose **Insert →Design Feature →Thread**
53. Click on the **Detailed** radio button
54. Keep the **Rotation** to be **Right Hand**
55. Click on the bolt shaft (the long cylinder below the hexagon head) Once the shaft is selected, all the values will be displayed in the *Thread* window. Keep all these default values.
56. Click **OK**





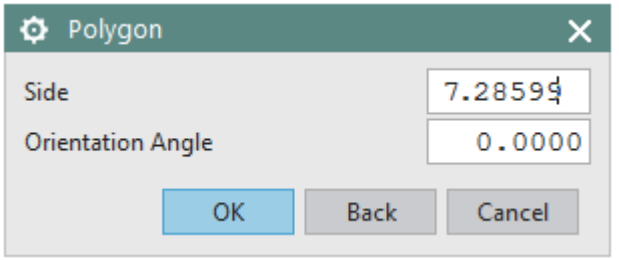
5.10 Hexagonal Screw

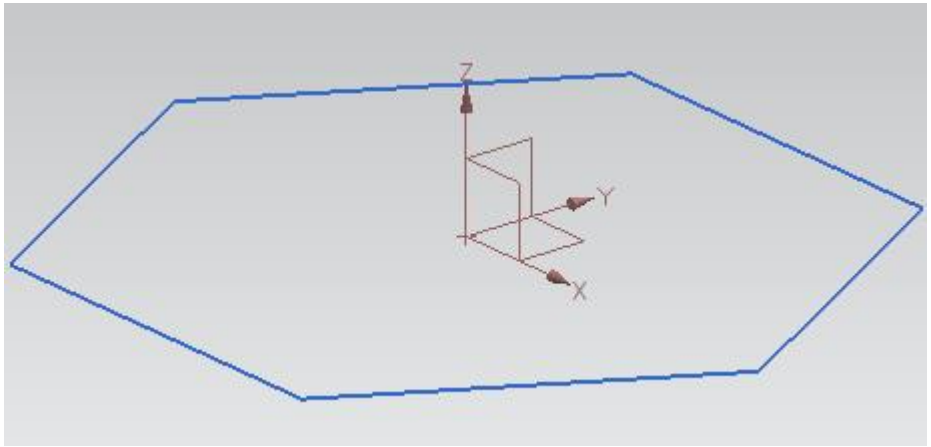


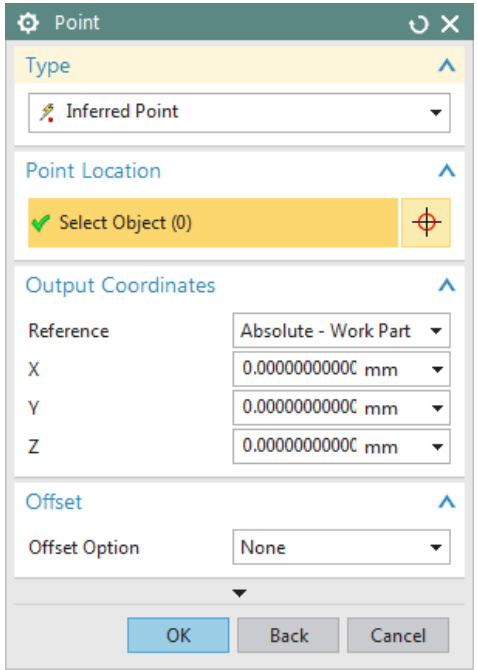
5.11 Hexagonal Nut

Create a new file and save it as **Impeller_hexa-nut.prt**

57. Choose **Insert →Curve →Polygon**
58. Input **Number of Dides** to be **6**
59. Create a hexagon with each side measuring **7.28599 mm** and constructed at the **Origin**
60. Choose **Insert →Design Feature →Extrude**



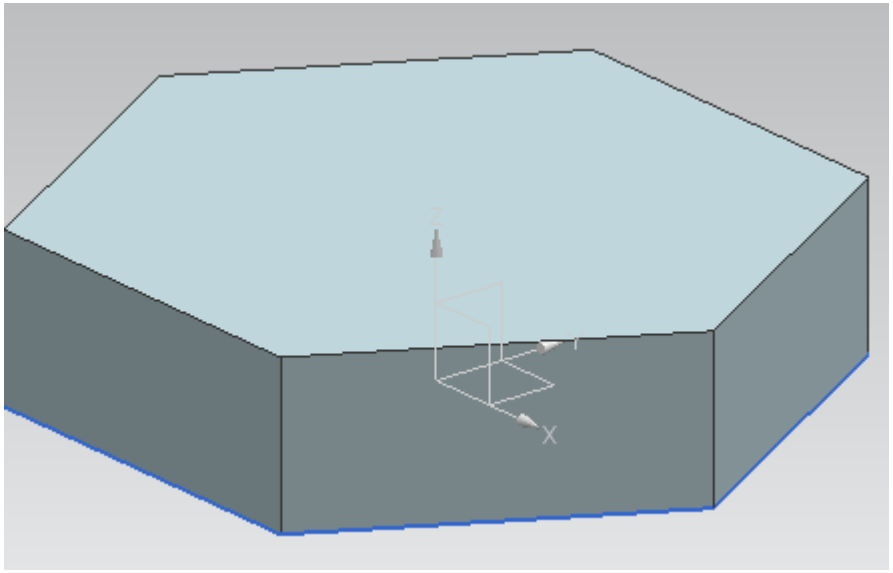




5.11 Hexagonal Nut

61. Choose **Insert →Design Feature →Extrude**

Select the **Hexagon** to be extruded and enter the **End Distance** as **3.175 mm.**



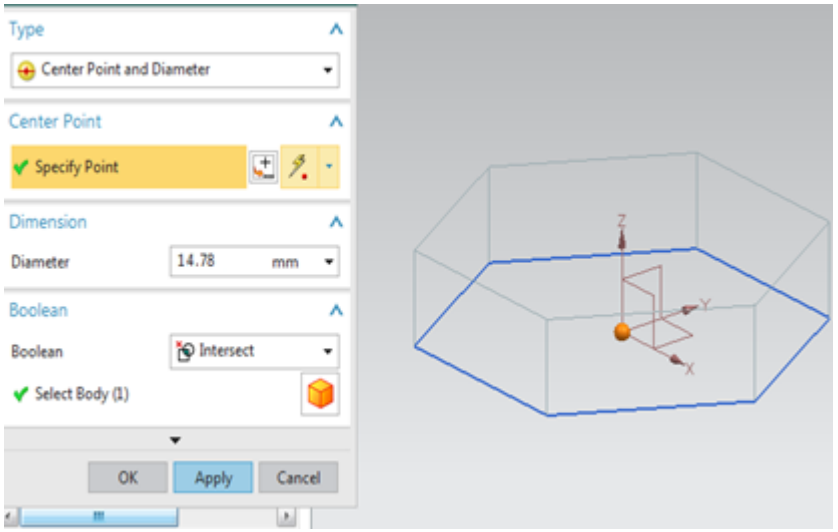
5.11 Hexagonal Nut

62. Choose **Insert →Design Feature →Sphere**

63. Enter the **Center Point** location in the **Point Dialog** window as follows XC = 0; YC = 0; ZC = 3.175

64. Enter the **Diameter** value **14.78 mm**

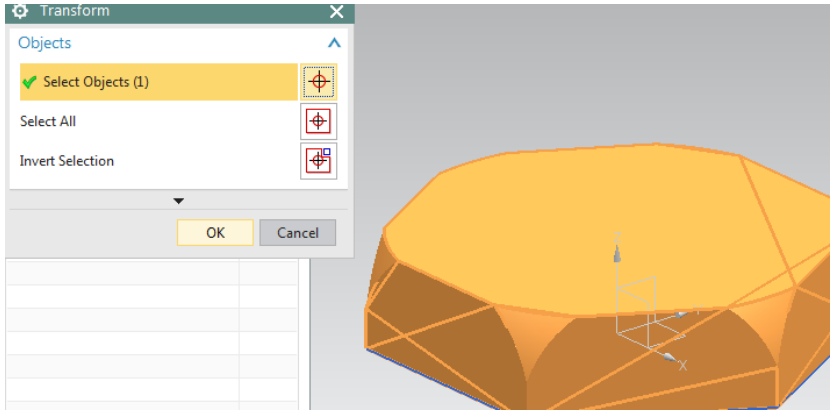
65. In the **Boolean** operations dialog box select **Intersect** and click **OK**



5.11 Hexagonal Nut

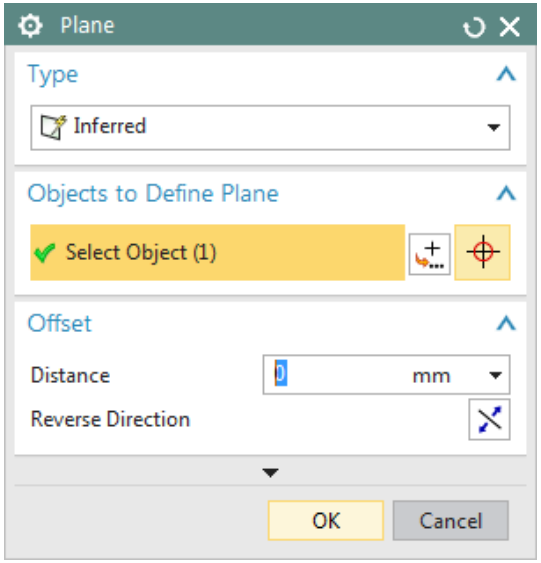
The model will look like the following. We will now use a *Mirror* command to create the other side of the *Nut.*

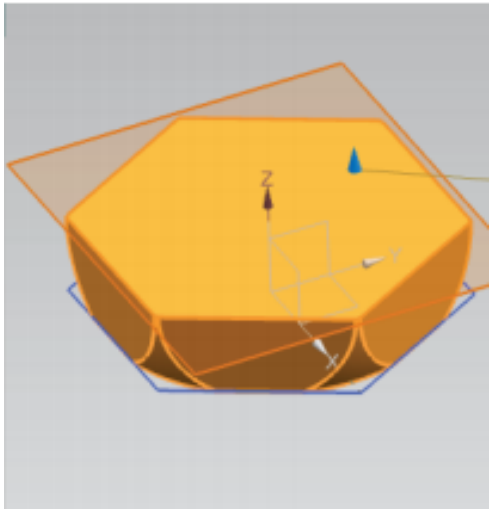
66. Choose **Edit →Transform**
67. Select the model and click **OK**



5.11 Hexagonal Nut

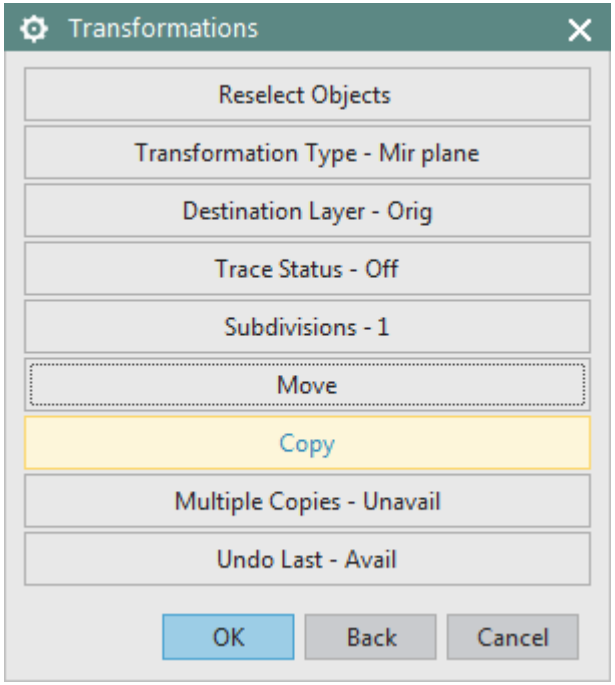
68. Click **Mirror Through a Plane**
69. Click on the flat side of the model as shown. Be careful not to select any edge.
70. Click in **OK.**





5.11 Hexagonal Nut

71. Choose **Copy**
72. Click **OK**
73. Click **Cancel**

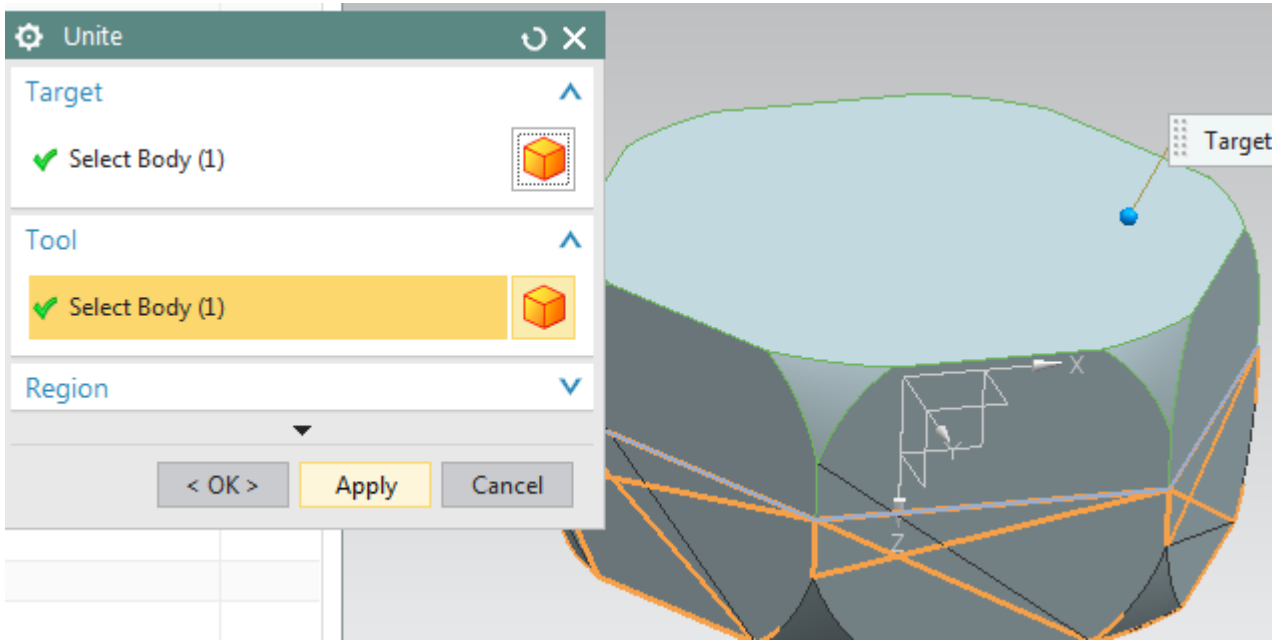


5.11 Hexagonal Nut

You will get the following model.

74. Choose **Insert** →**Combine Bodies** →**Unite**

75. Select the two halves and **Unite** them



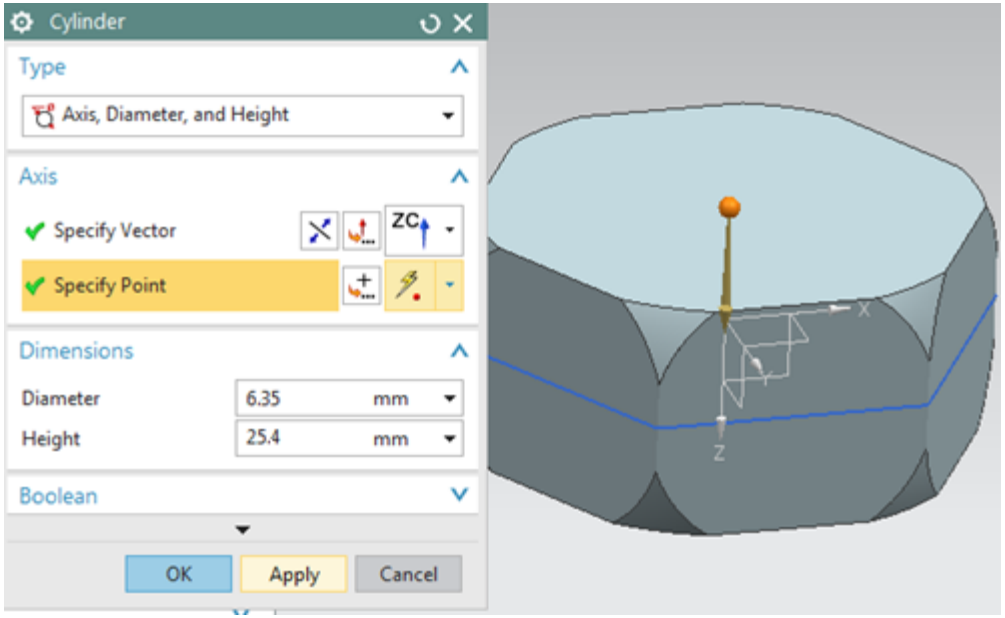
5.11 Hexagonal Nut

76. Insert a **Cylinder** with the vector pointing in the **ZC-Direction** and with the following dimensions:

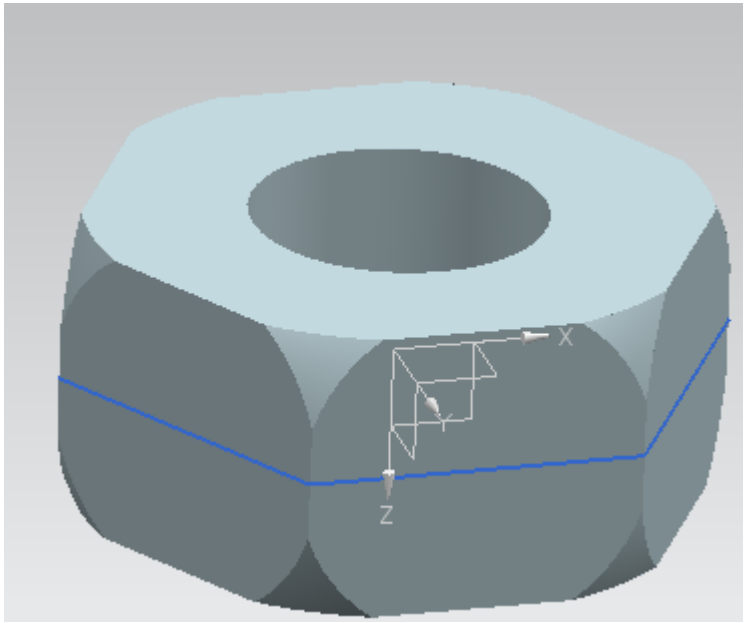
Diameter = **6.35** mm

Height = **25.4** mm

77. Put the cylinder on the **Origin** and **Subtract** this cylinder from the hexagonal Nut.



5.11 Hexagonal Nut

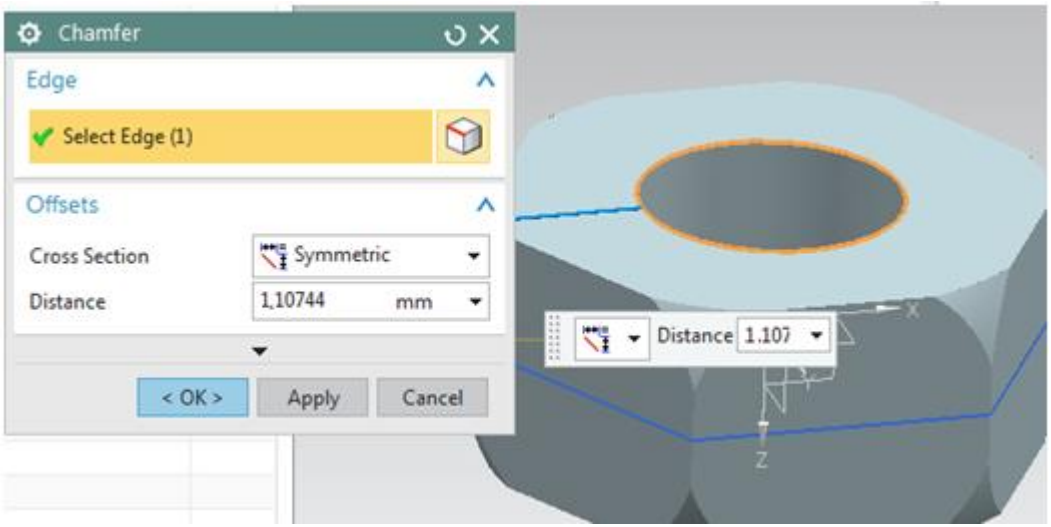


5.11 Hexagonal Nut

Now, we will chamfer the inside edges of the nut.

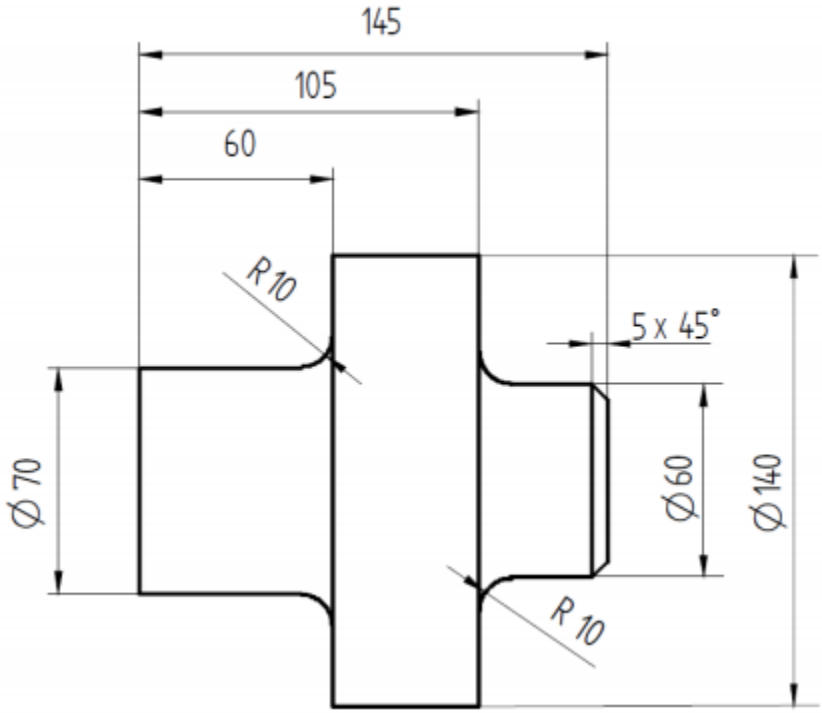
78. Choose **Insert →Detail Feature →Chamfer**
79. Select the two inner edges as shown and click **OK**
80. Enter the Distance as **1.10744** mm and click **OK**

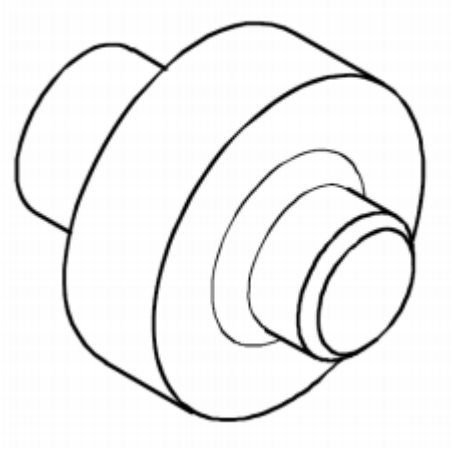
You will see the chamfer on the nut. Save the model.

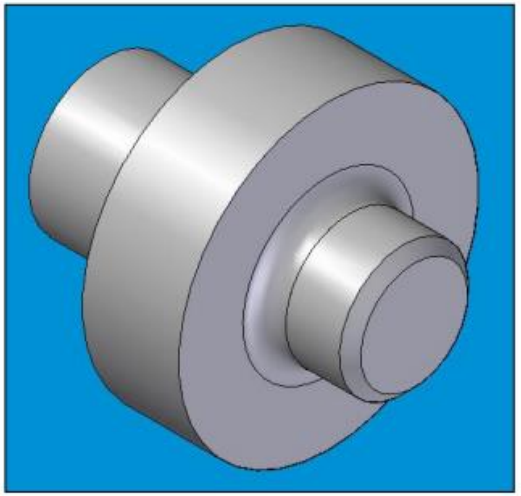


Aula 04

Exercise 1







Aula 04

Exercise 2