MAE 656 - Advanced Computer Aided Design

04. 2D and 3D Solids – Doc 03

3D Solid Simulation

Introduction

In this class we will simulate the cover of a pressure cylinder.

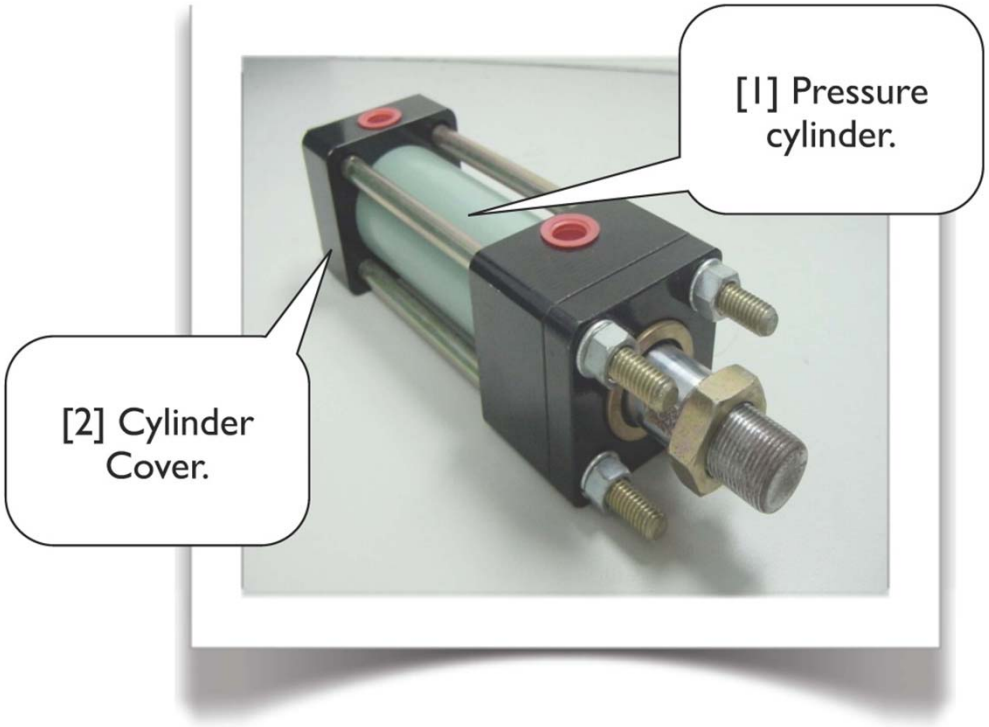
This simulation will show the steps that have to be followed to construct a 3D fem model and the results that can be obtained from it.

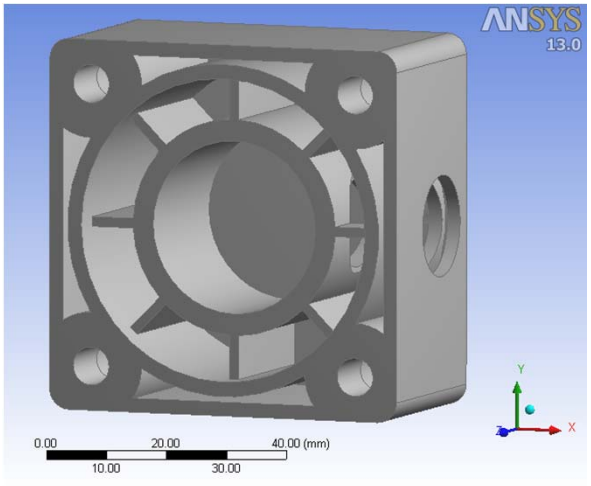
We will see that most of the effort is required in the definition of the model.

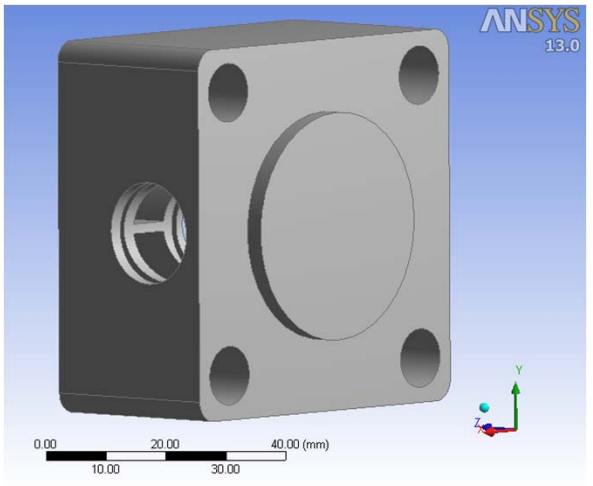
This example has been taken from the book:

Huei-Huang Lee. *Finite Element Simulations with ANSYS Workbench 13. Theory – Applications – Case Studies*. SDC Publications (2011), ISBN 978-1585036530

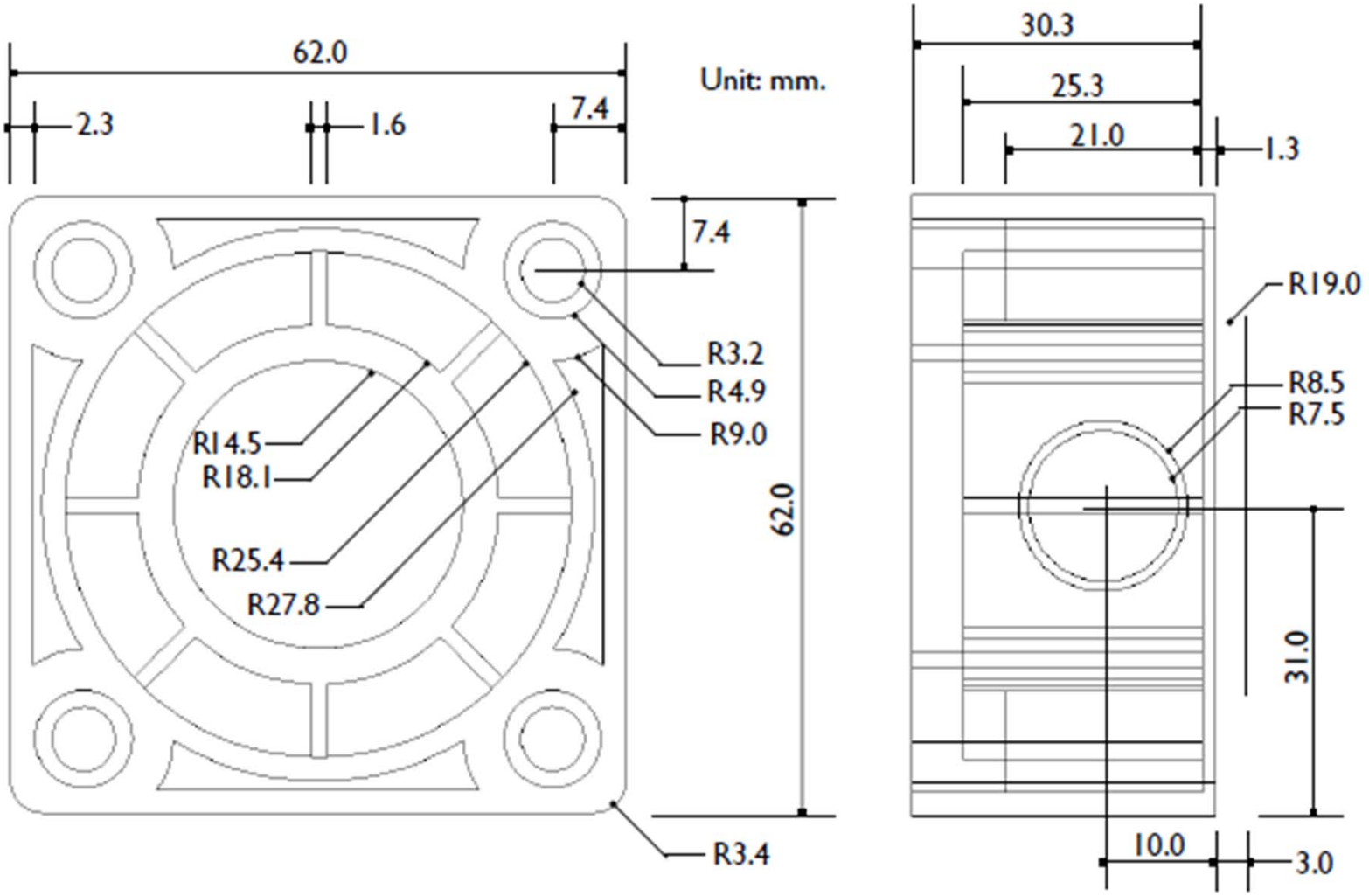
Cover of a Pressure cylinder



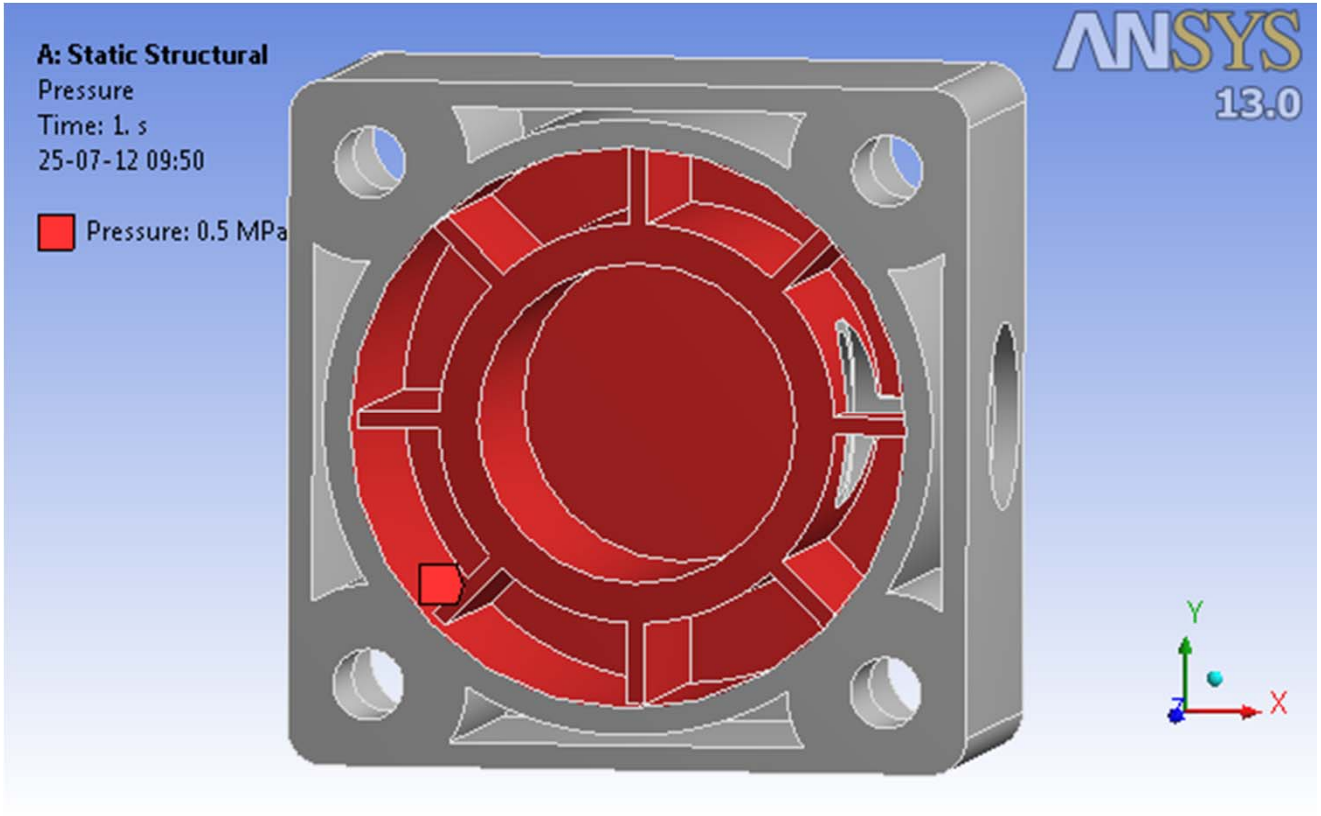




Most relevant dimensions

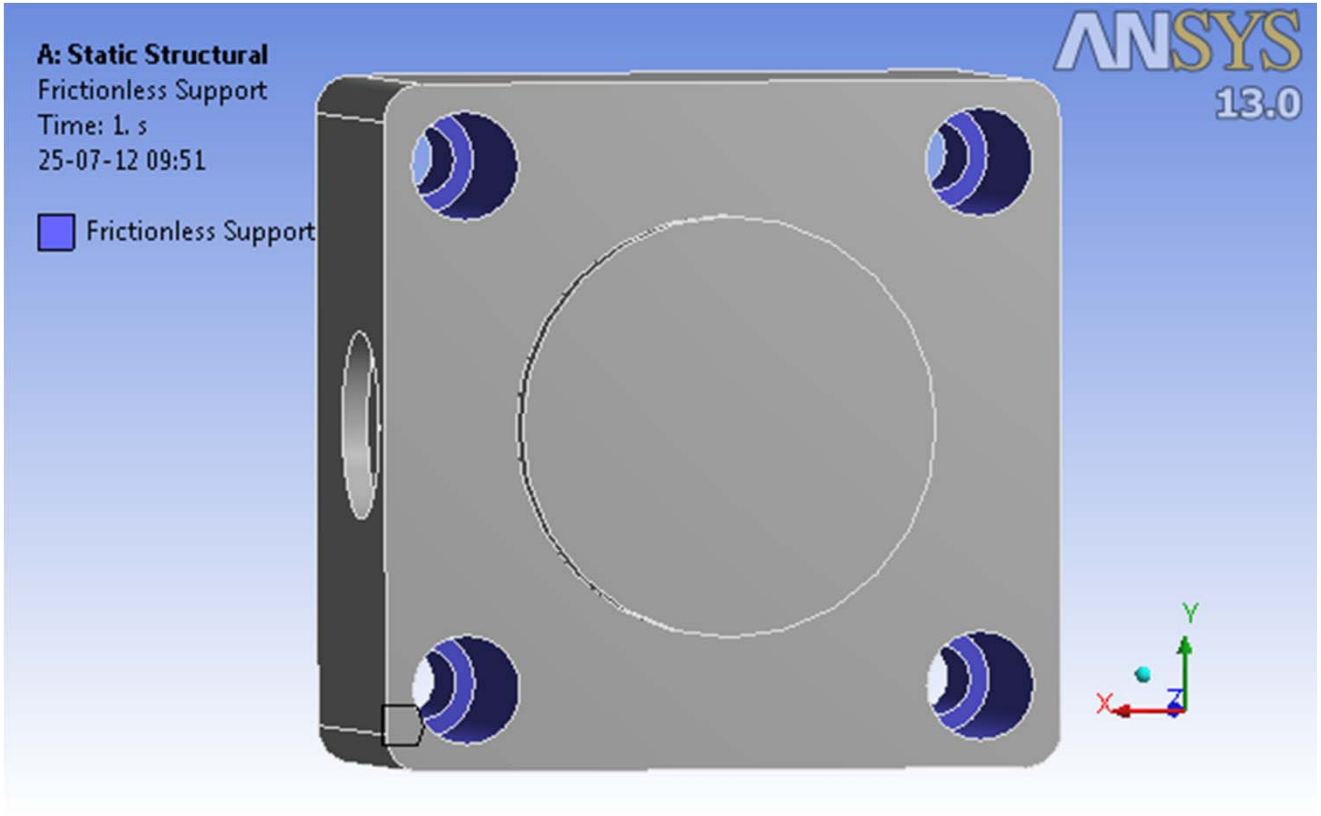


Boundary Conditions



The maximum pressure applied by the cylinder is 0.5 MPa. All surfaces in contact with the cylinder are affected by this pressure

Boundary Conditions



A frictionless support does not allow the displacement of a surface in its normal direction

Pressure Cover Validation

The specifications of the cover require a radial deformation smaller than 10 micrometers (0.01mm).

The fracture stress of the plastic used is 50 MPa. The Von-Mises stresses in the material should be lower than this maximum value.

Do we fulfill these two requirements?

Engineering Data

The pressure cover is manufactured with a plastic which mechanical properties are:

Young Modulus (E) = $2.2 \cdot 10^{10}$ Pa

Poisson Modulus (ν) = 0.3 Pa

As the material is expected to remain in elastic range, no other mechanical property is required.

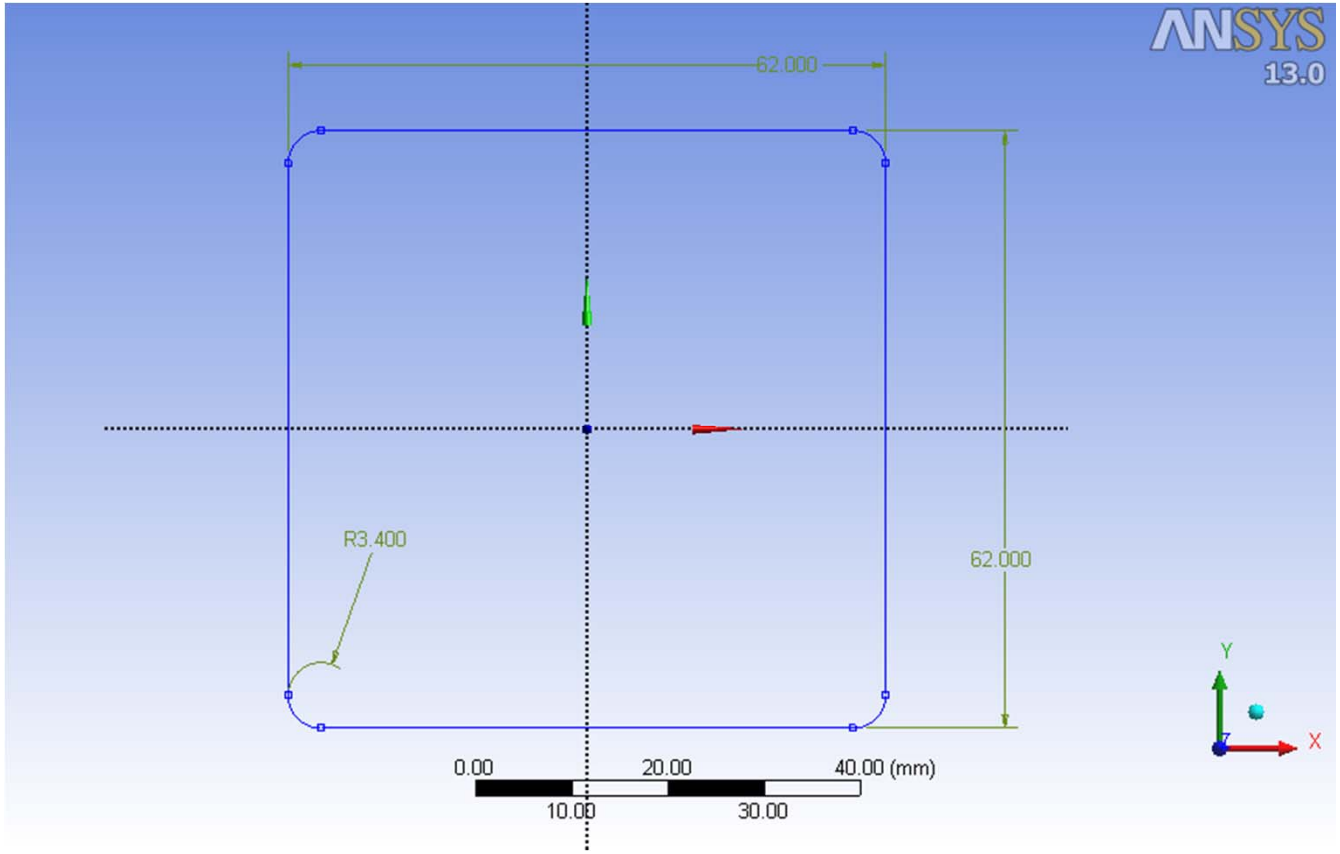
Self-weight will not be considered, therefore density is not required

Geometry Definition

In the following are described the steps that may be followed to generate the cover pressure geometry.

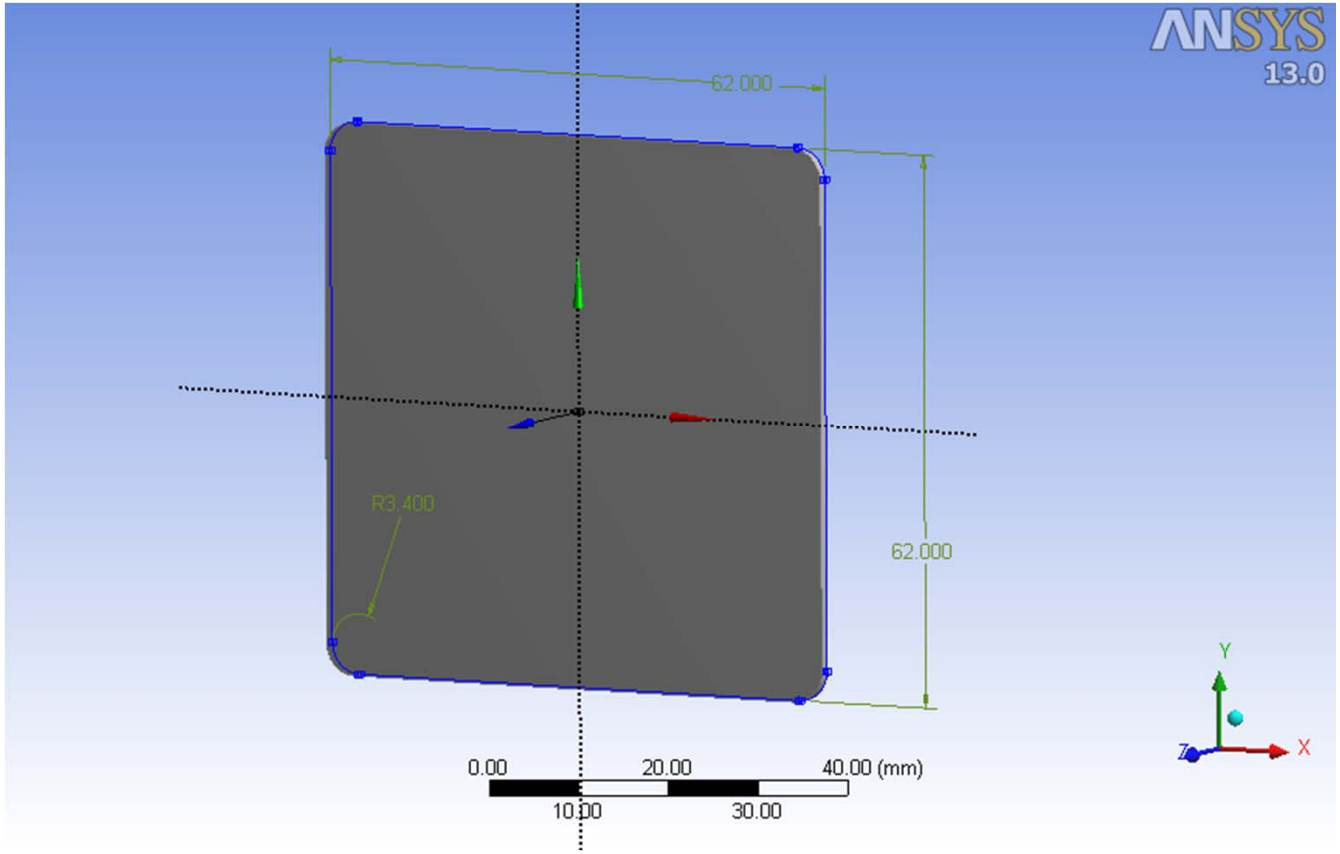
The units used to create the geometry are millimeters.

Geometry Definition



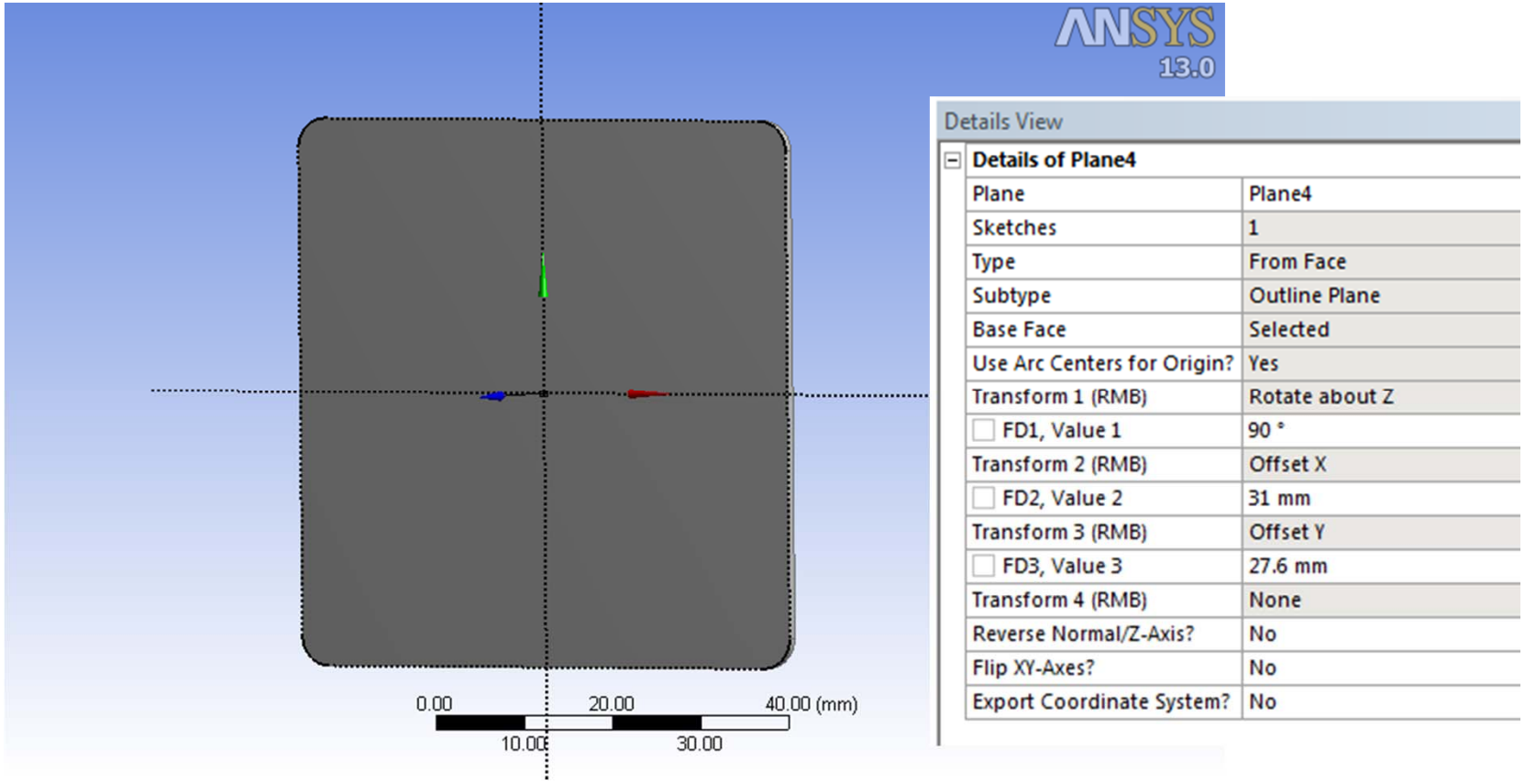
Draw the following sketch (sketch 1) in plane xy

Geometry Definition



Extrude sketch 1 1.3mm

Geometry Definition

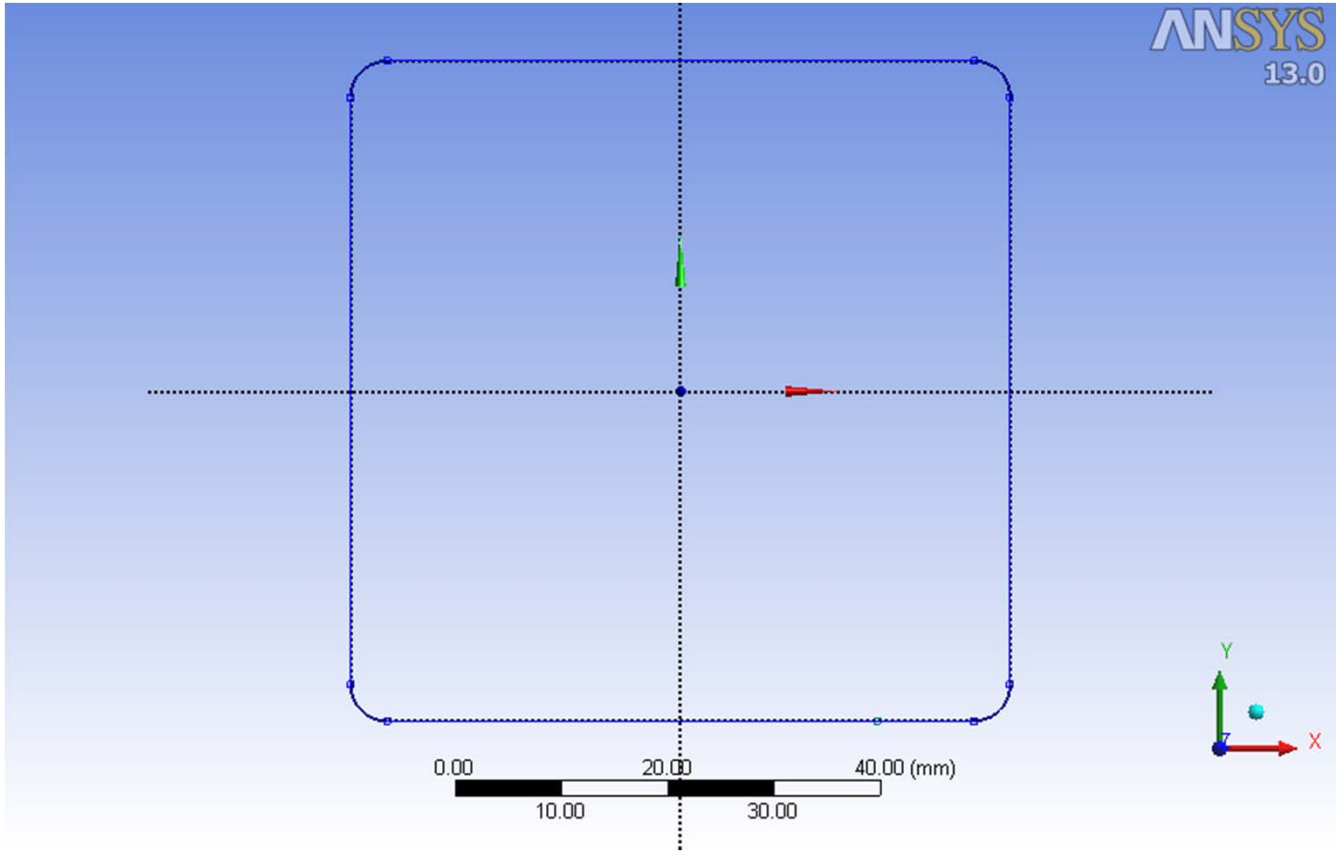


Details View

Details of Plane4	
Plane	Plane4
Sketches	1
Type	From Face
Subtype	Outline Plane
Base Face	Selected
Use Arc Centers for Origin?	Yes
Transform 1 (RMB)	Rotate about Z
FD1, Value 1	90 °
Transform 2 (RMB)	Offset X
FD2, Value 2	31 mm
Transform 3 (RMB)	Offset Y
FD3, Value 3	27.6 mm
Transform 4 (RMB)	None
Reverse Normal/Z-Axis?	No
Flip XY-Axes?	No
Export Coordinate System?	No

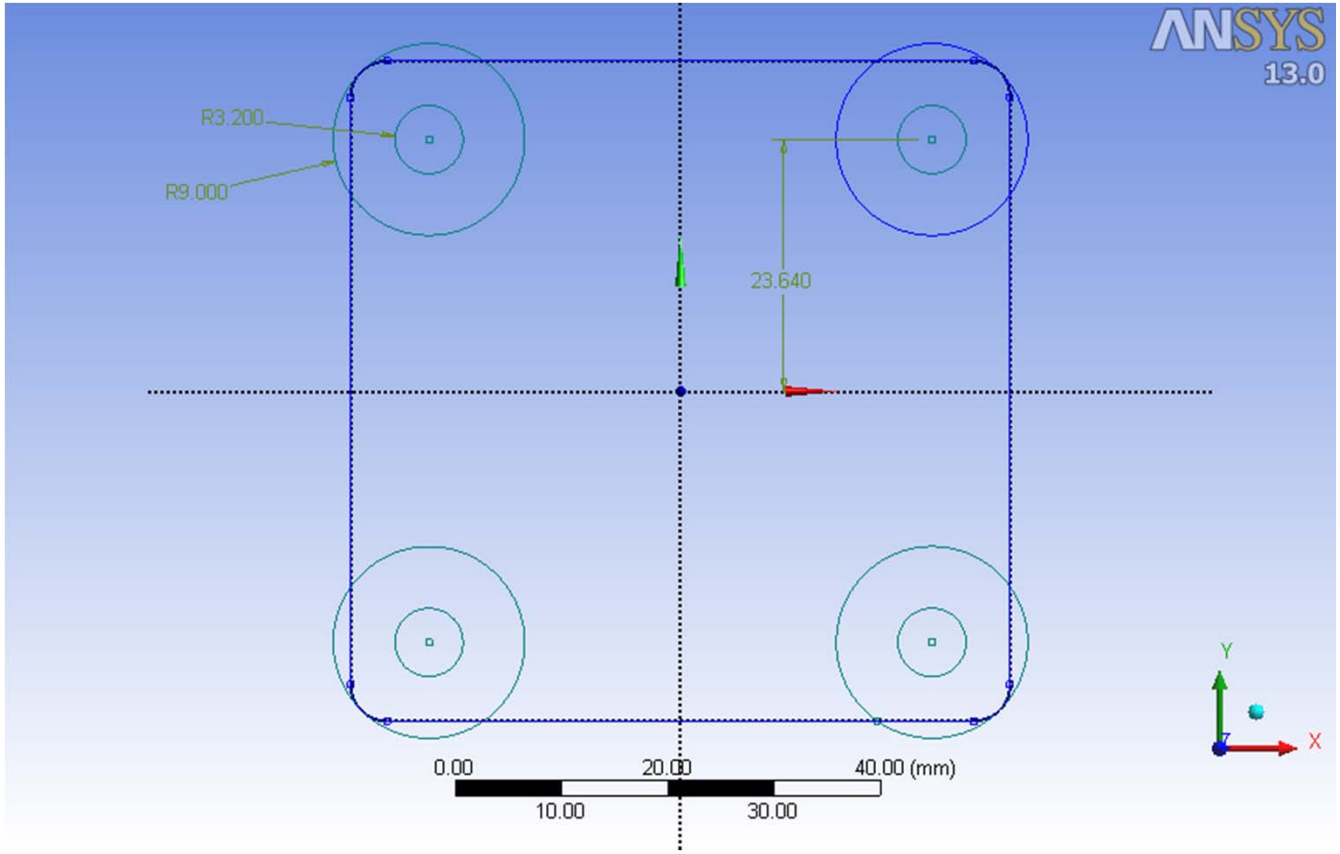
Create a new plane from the extruded face of sketch 1 and centered in that face. In this plane we will draw all circular regions and stiffeners.

Geometry Definition



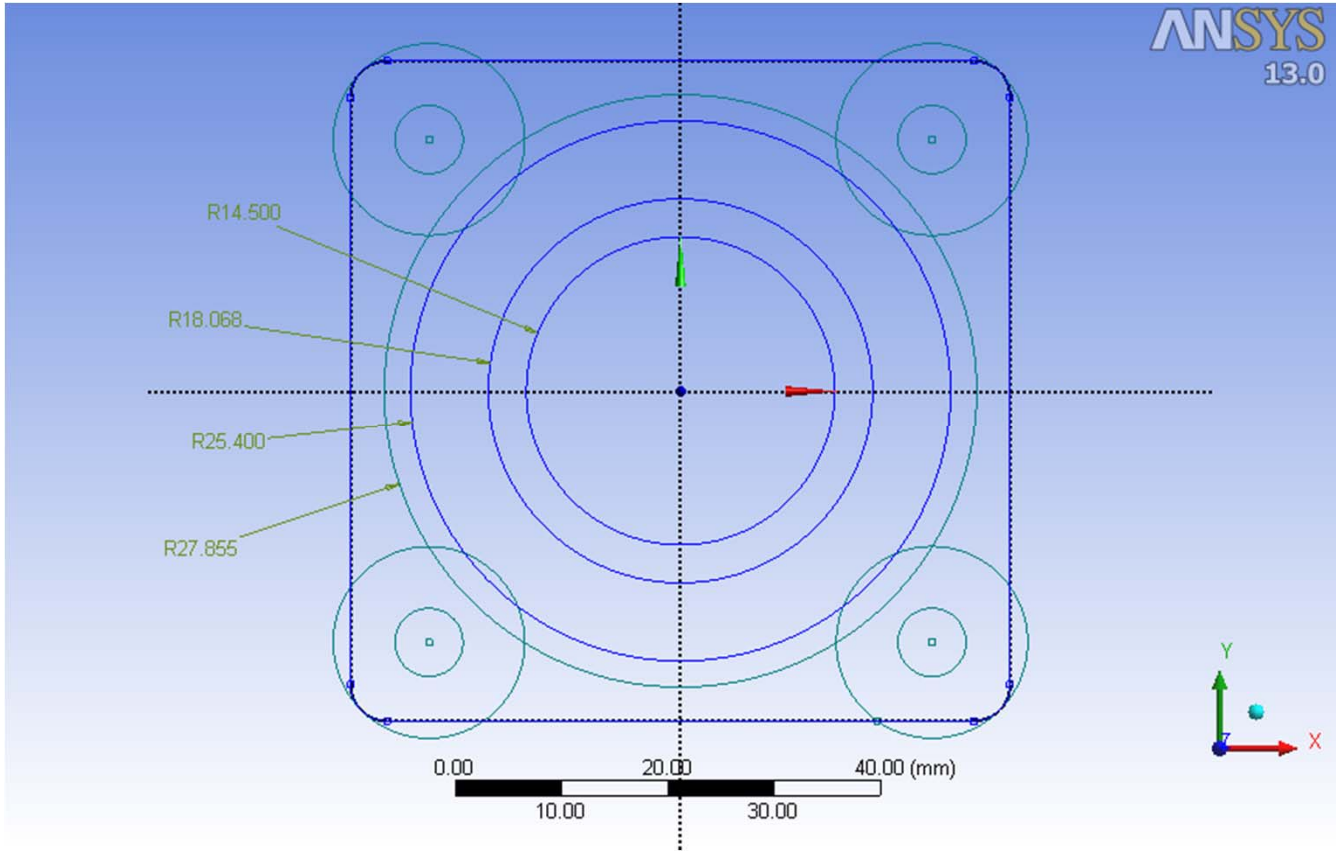
As the plane has been created as an *Outline Plane* it has a boundary. We will duplicate the outline boundary of the plane.

Geometry Definition



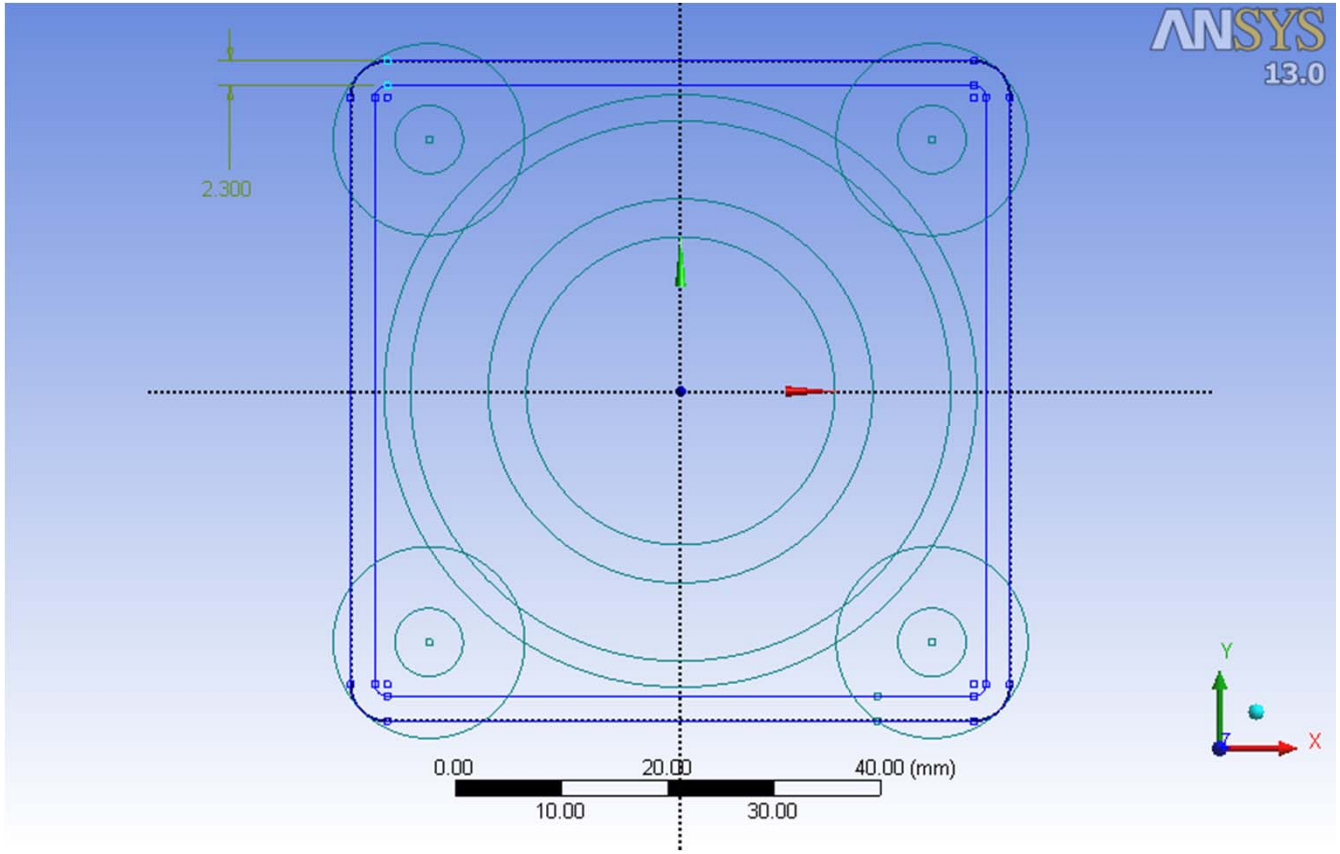
Draw the following circles. Draw one pair and replicate them

Geometry Definition



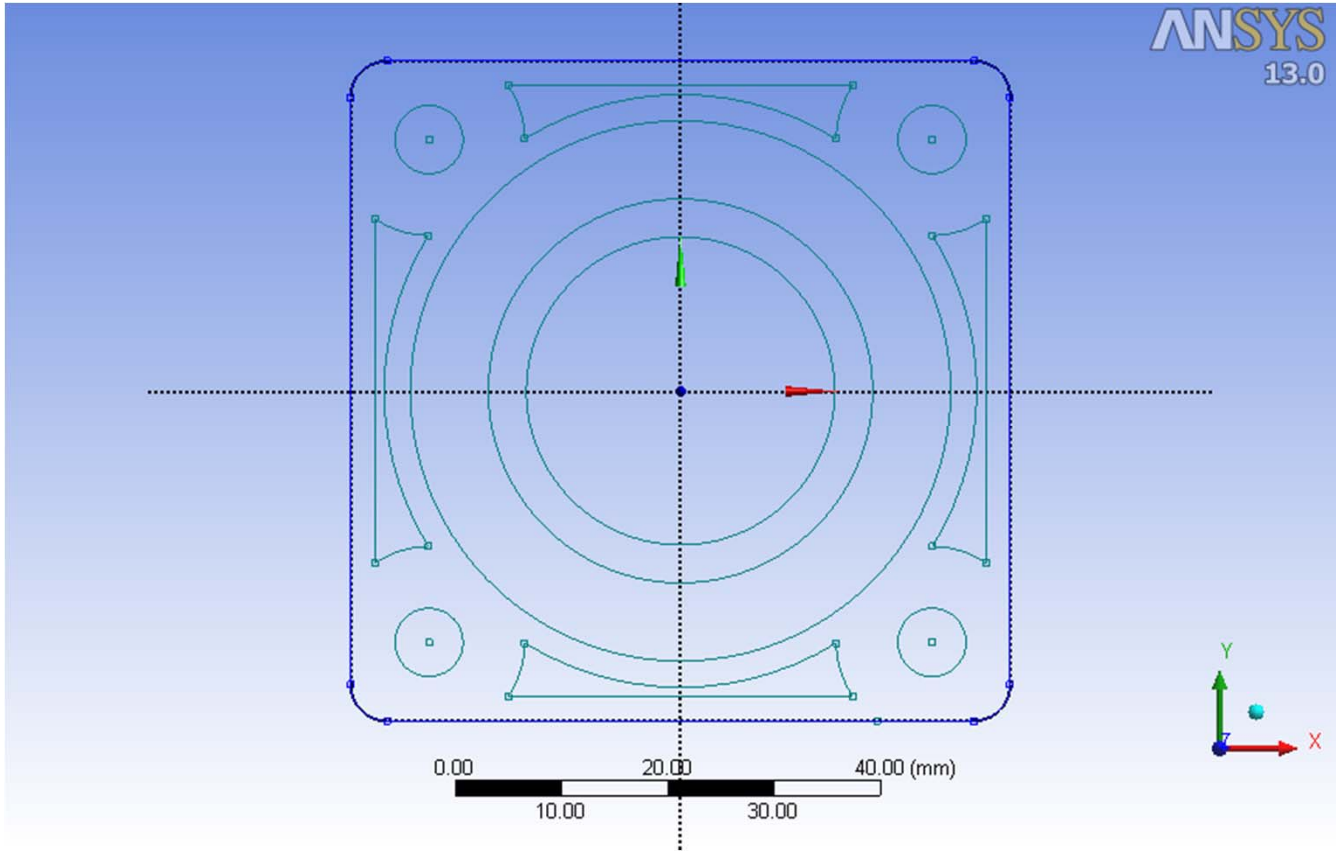
Draw the interior circles, centered in the xy axis origin

Geometry Definition



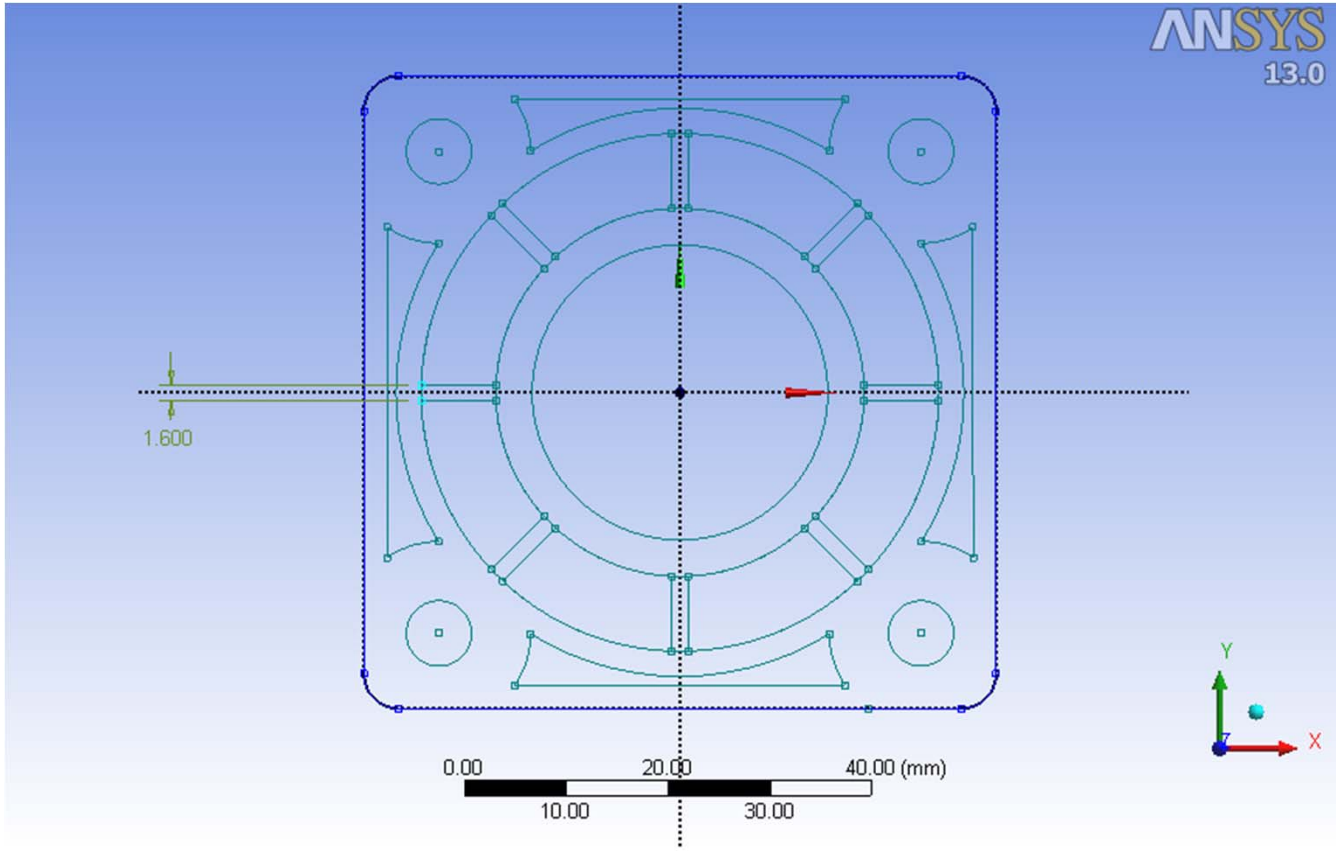
Offset the exterior line 2.3mm inwards

Geometry Definition



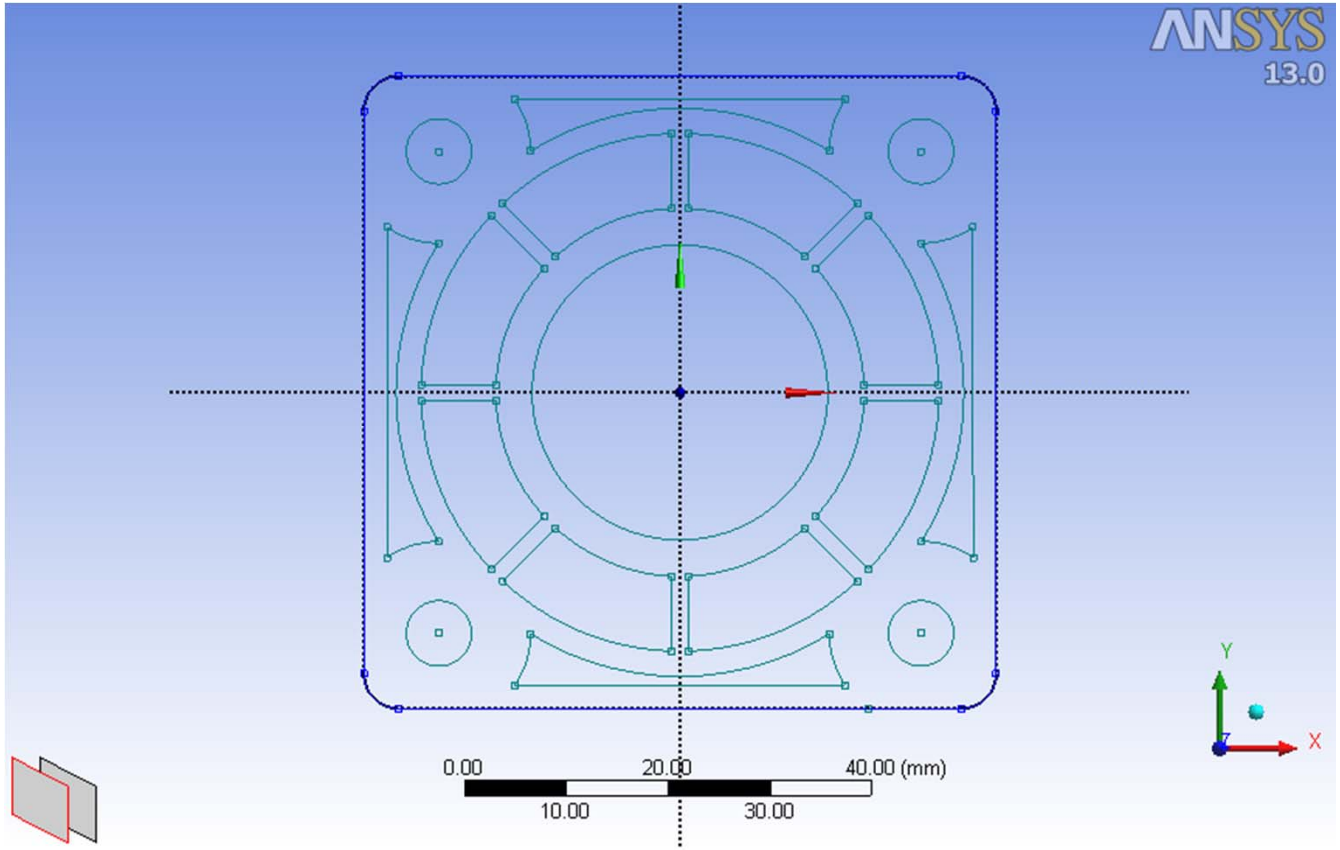
Trim some of the shapes drawn to get the following figure

Geometry Definition



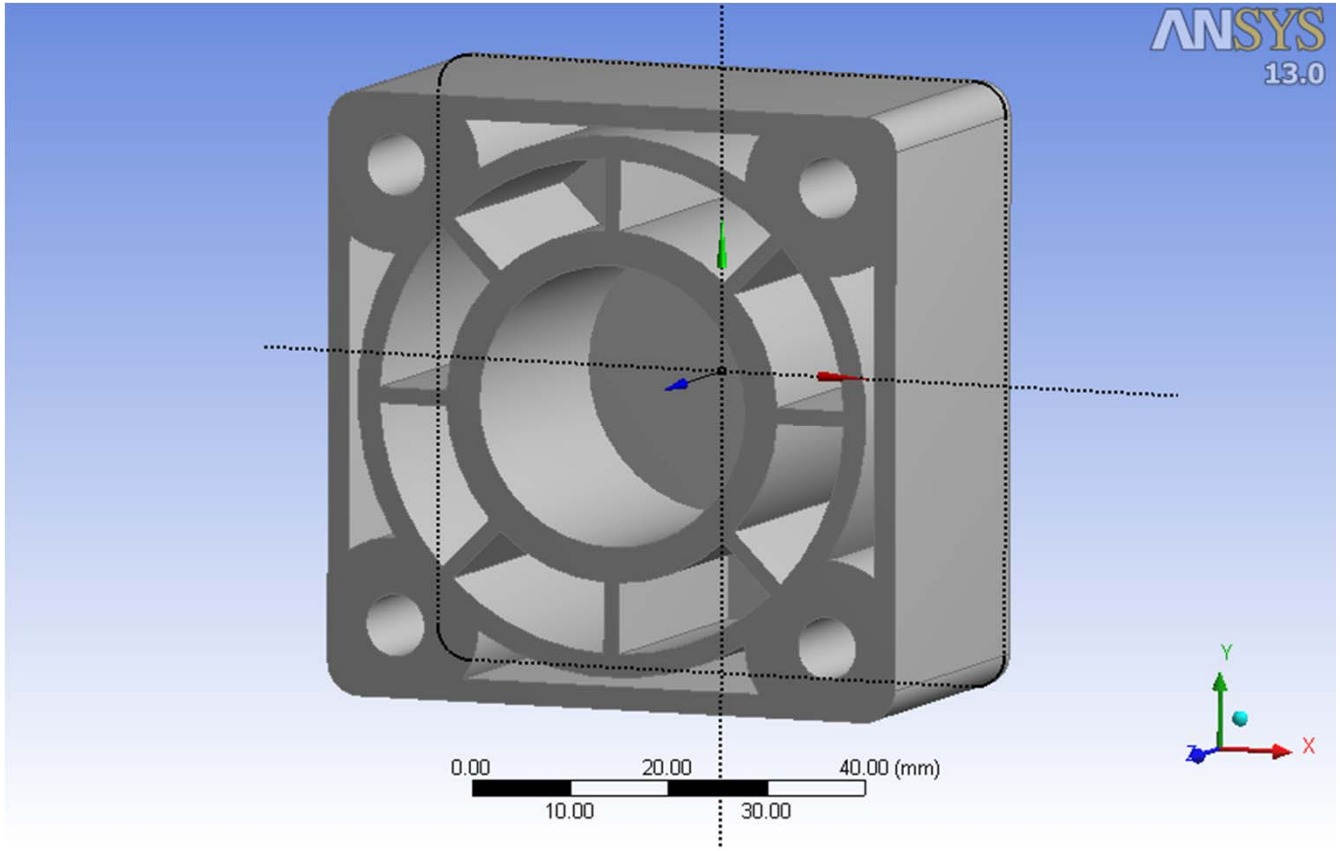
Create the following lines, separated 1.6mm. To do so you may create the first two vertical or horizontal lines and replicate them afterwards

Geometry Definition



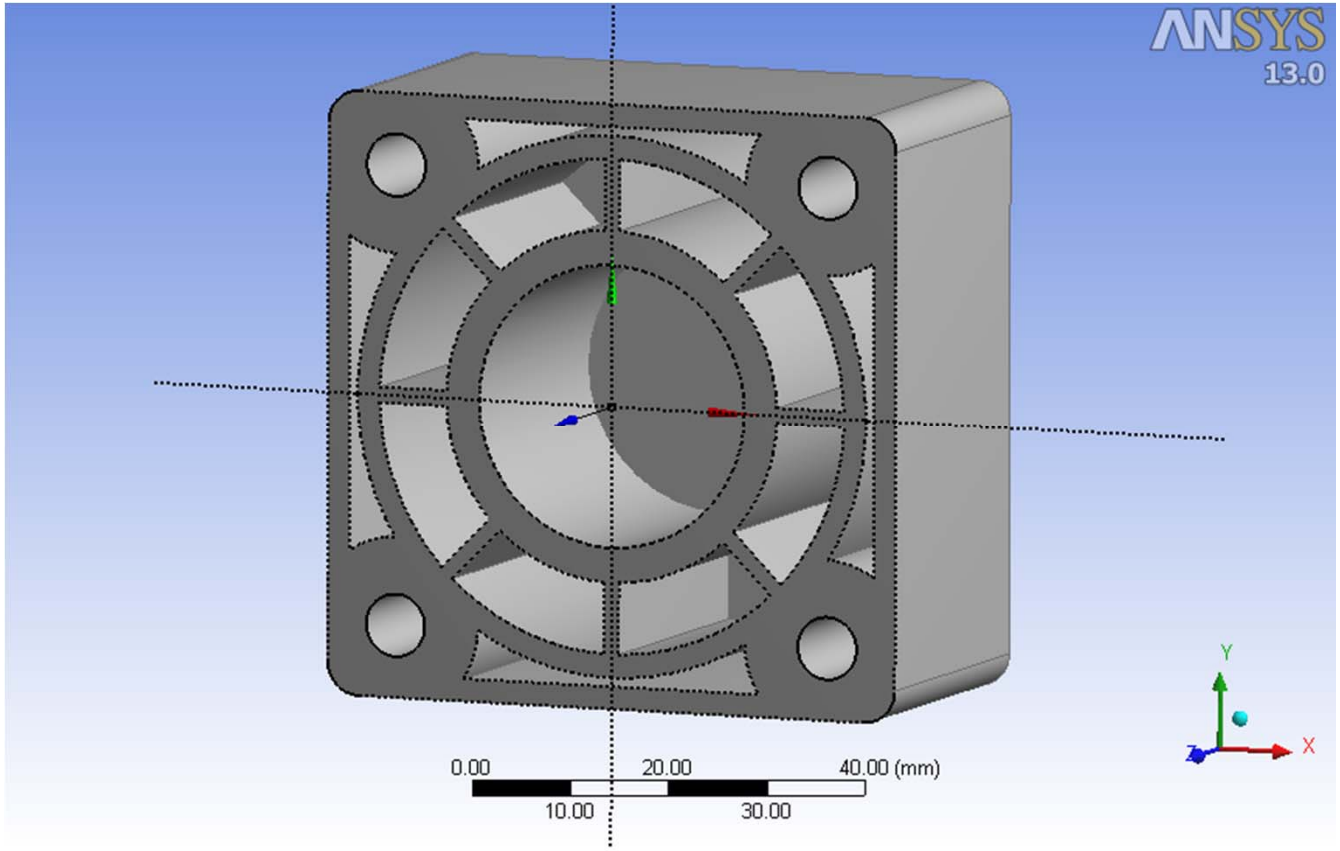
Trim the different objects to obtain the final sketch 2

Geometry Definition



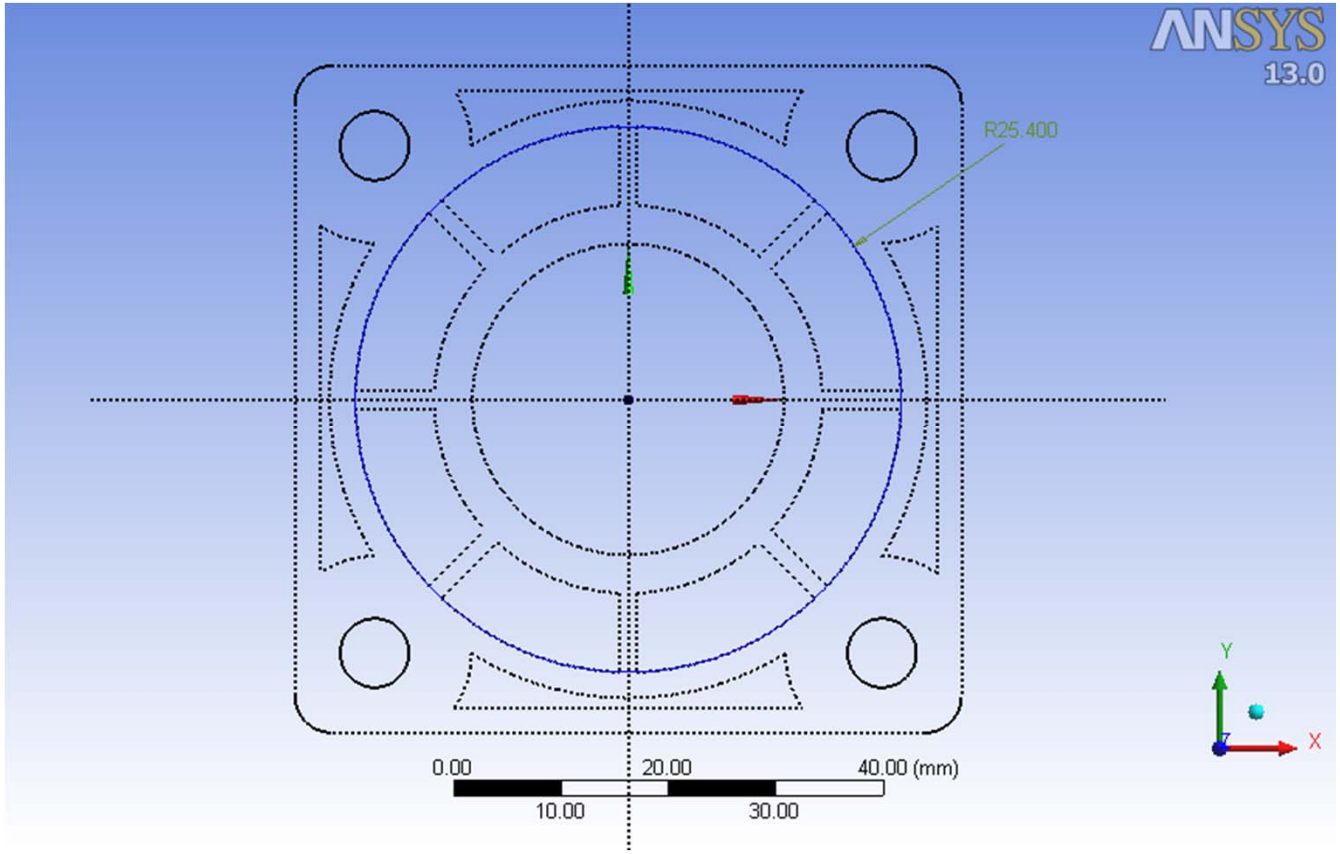
Extrude 30.3mm sketch 2

Geometry Definition



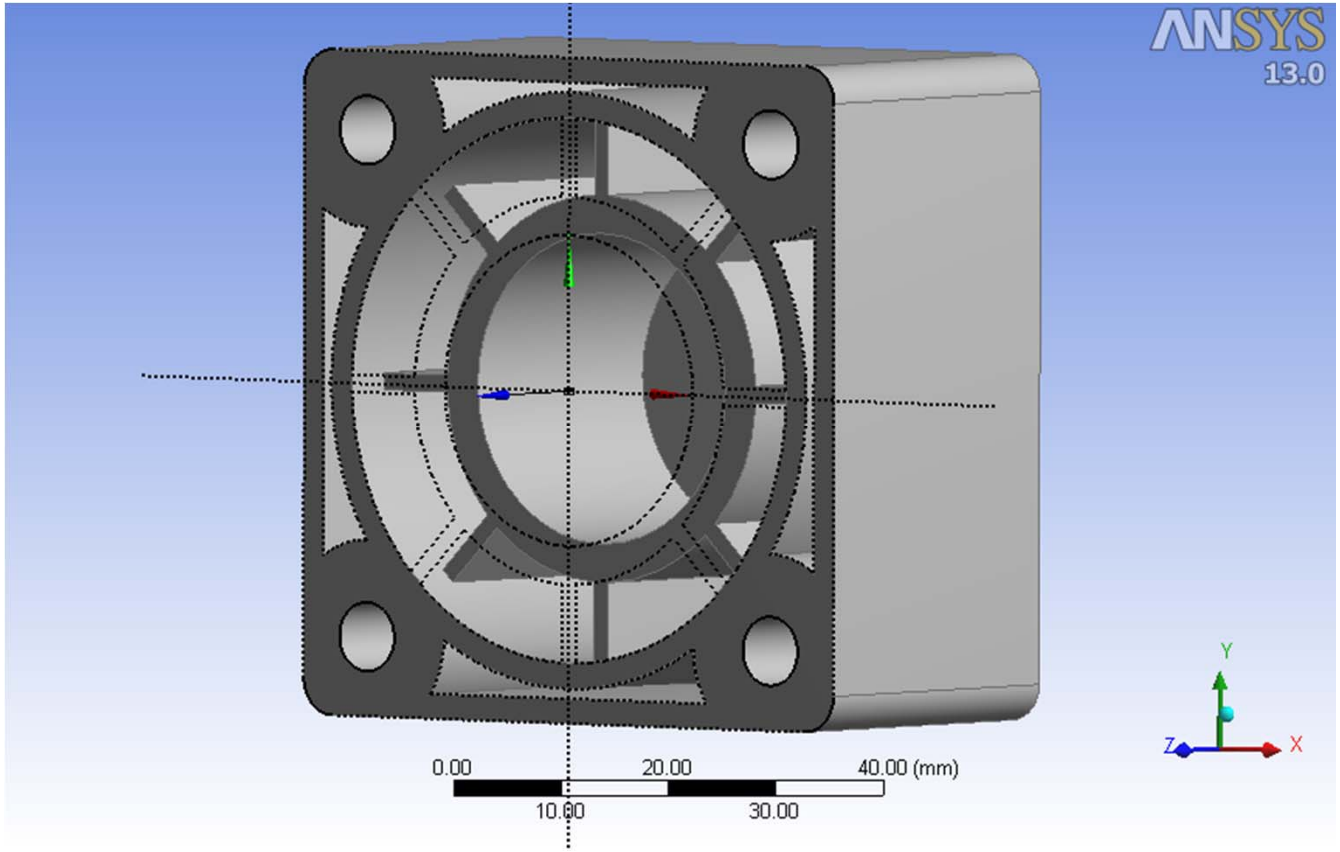
Create a new plane from the outer face of the extruded sketch 2

Geometry Definition



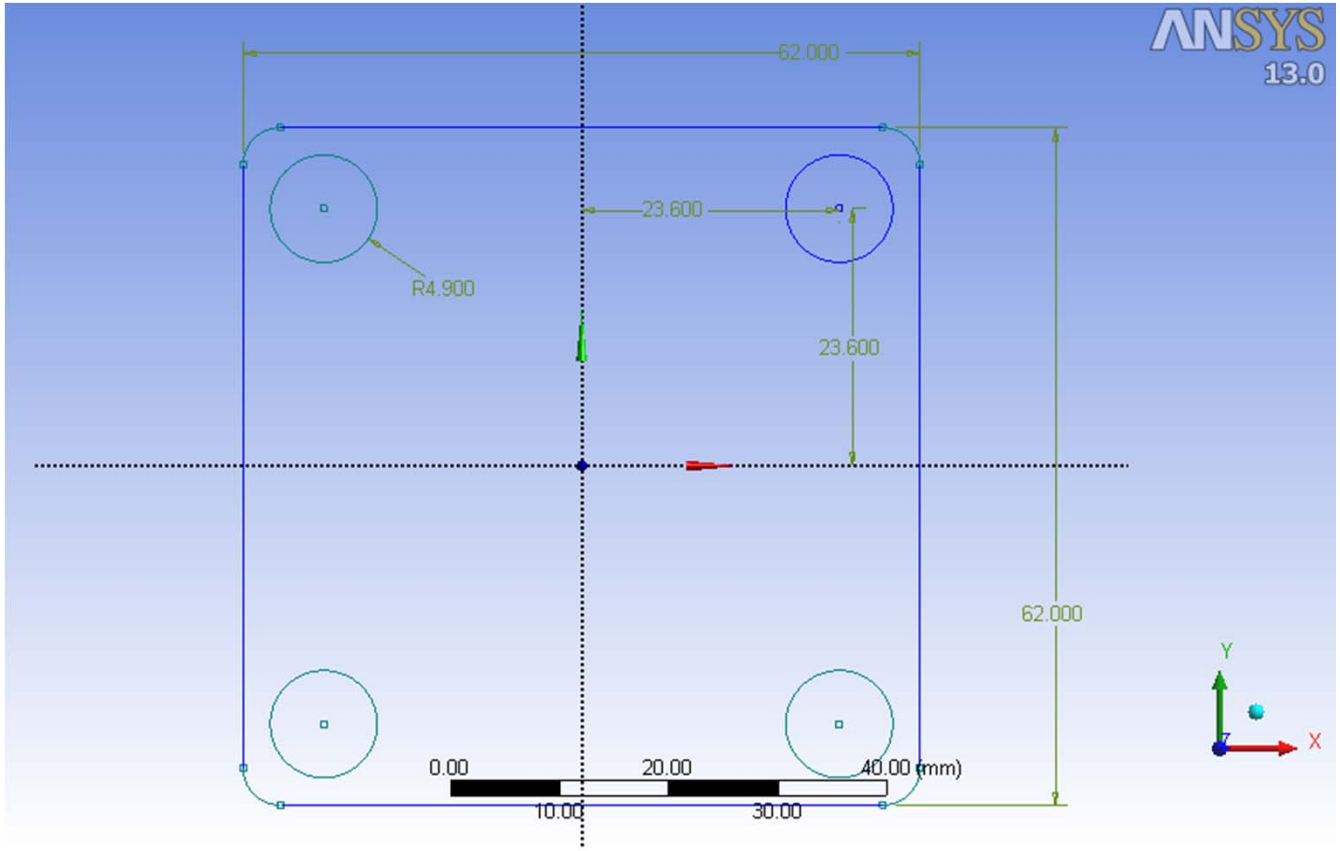
Draw a circle of radii 25.4mm in the new plane (sketch 3)

Geometry Definition



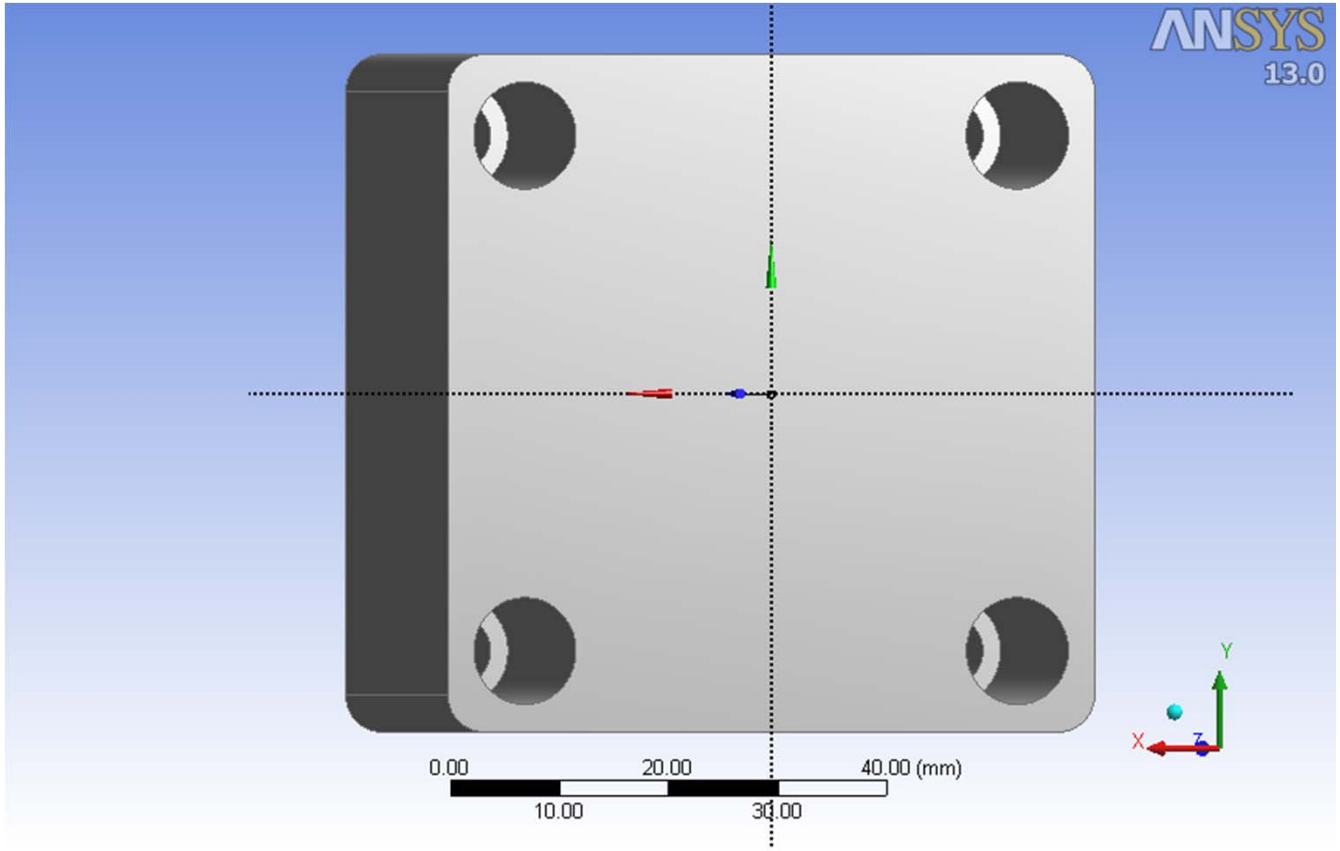
Extrude 5mm the circle in *Reversed* direction and with the *Cut Material* option to remove part of the extruded sketch 2

Geometry Definition



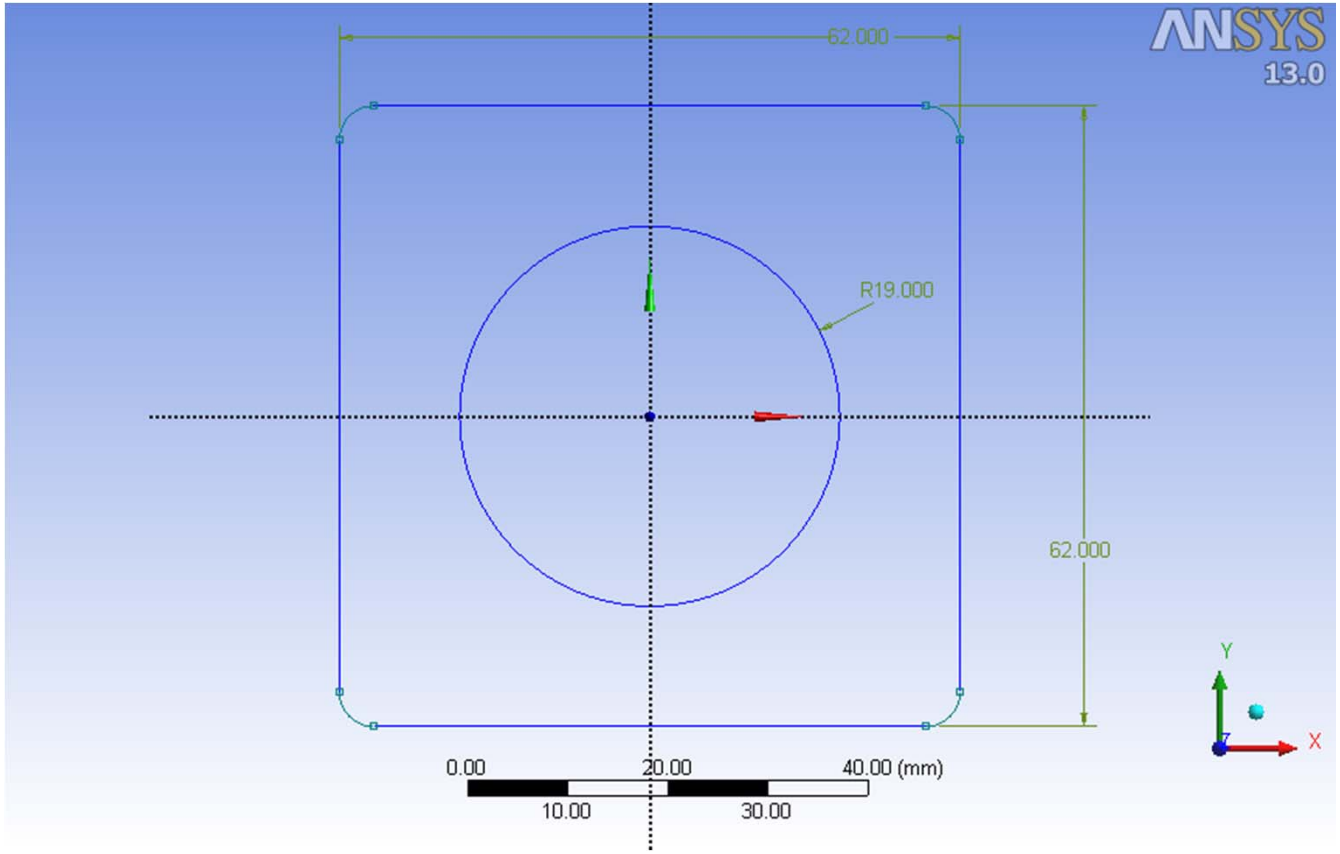
Add a *New Sketch* in XY plane and draw in it the following 4 circles (sketch 4). It is necessary to select the new sketch option. If not, the new geometry will be added to sketch 1.

Geometry Definition



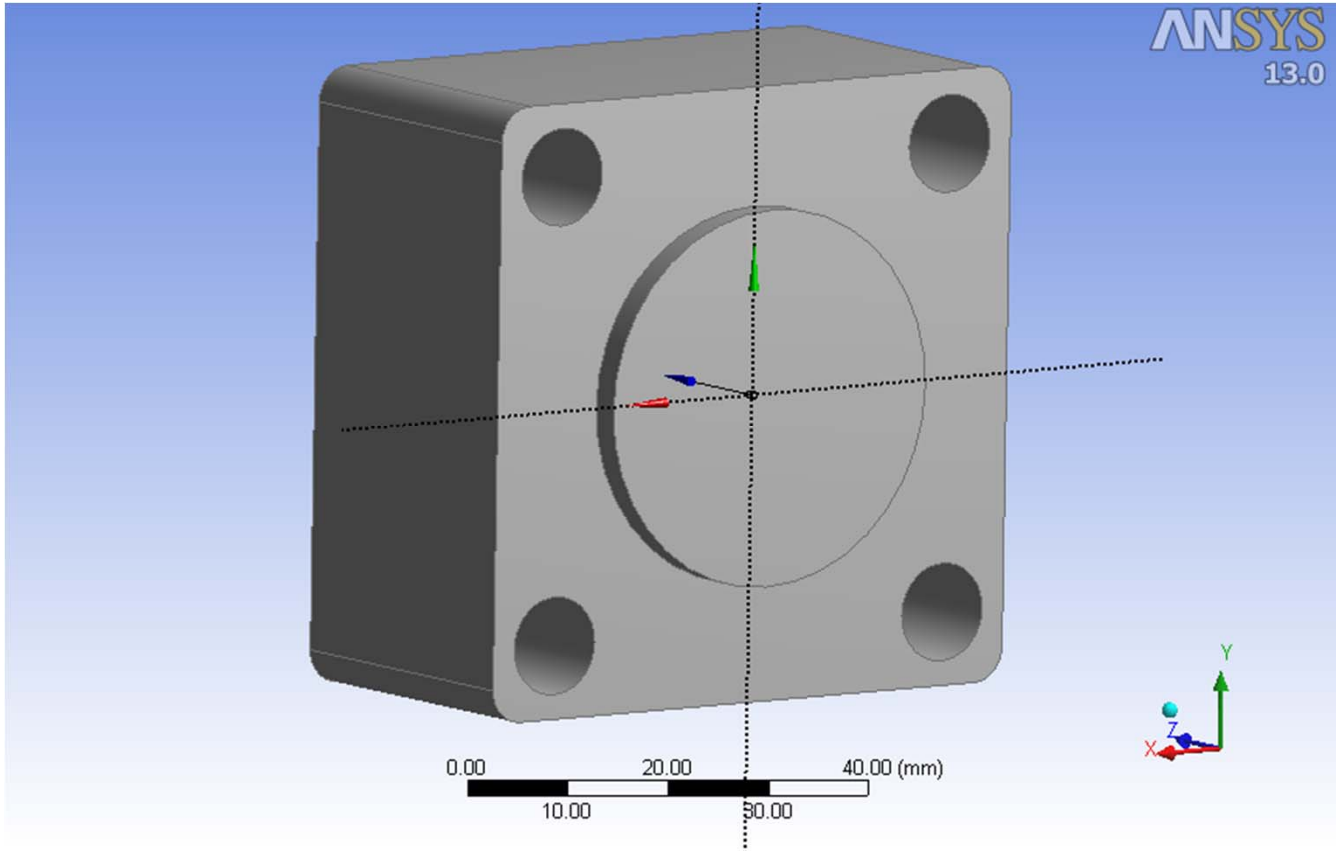
Extrude 21mm with the Remove material option sketch 4

Geometry Definition



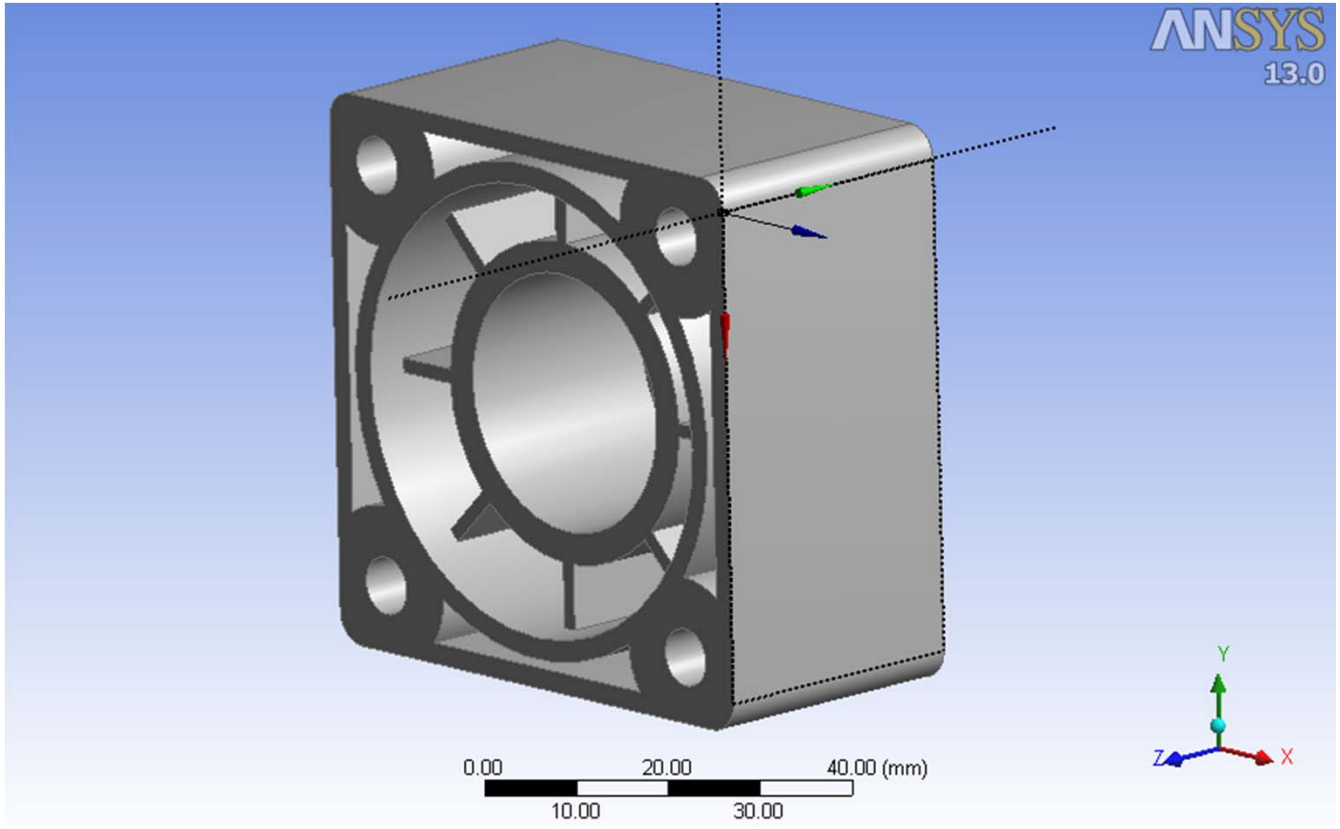
Add a new sketch in plane xy and draw the following circle (sketch 5)

Geometry Definition



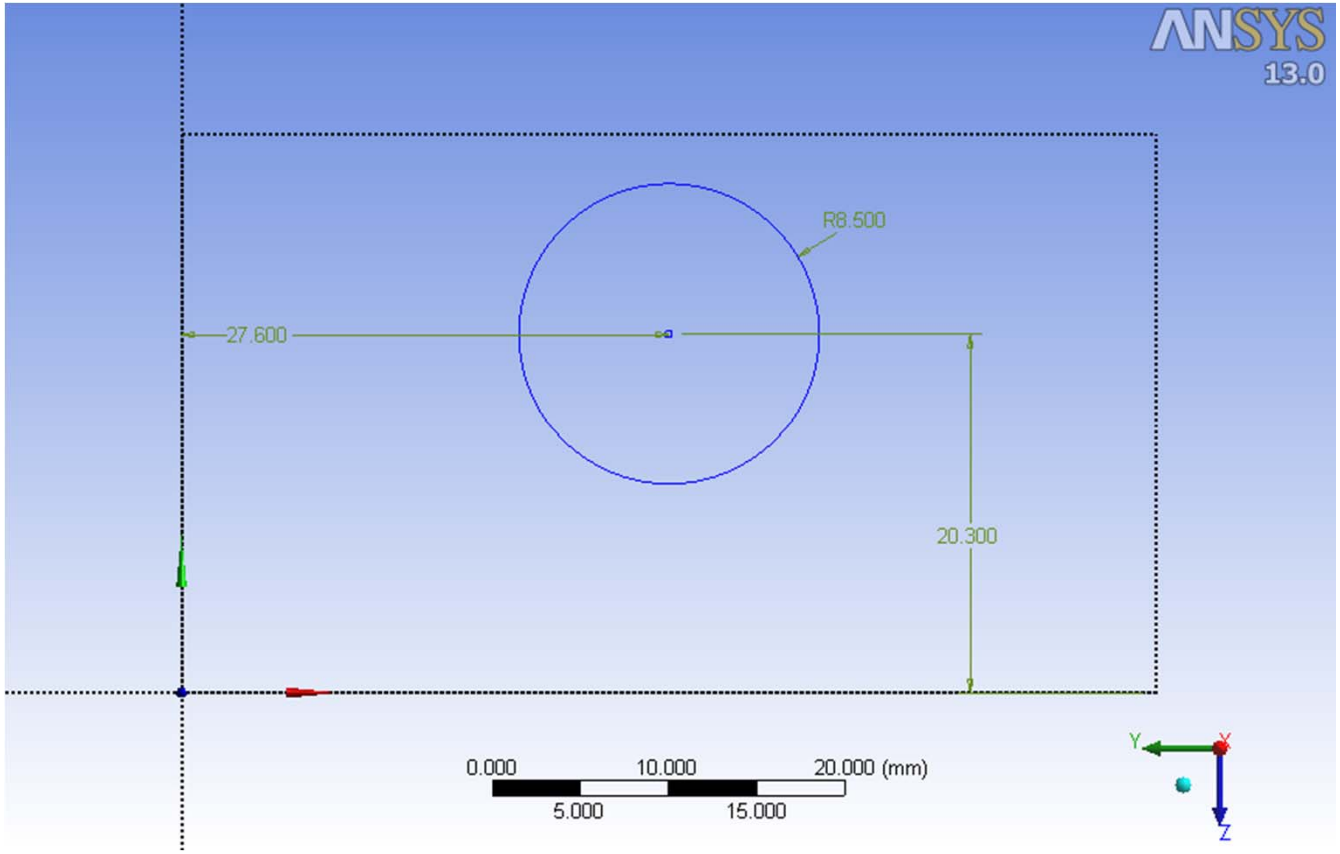
Extrude in reversed direction sketch 5 a depth of 3mm

Geometry Definition



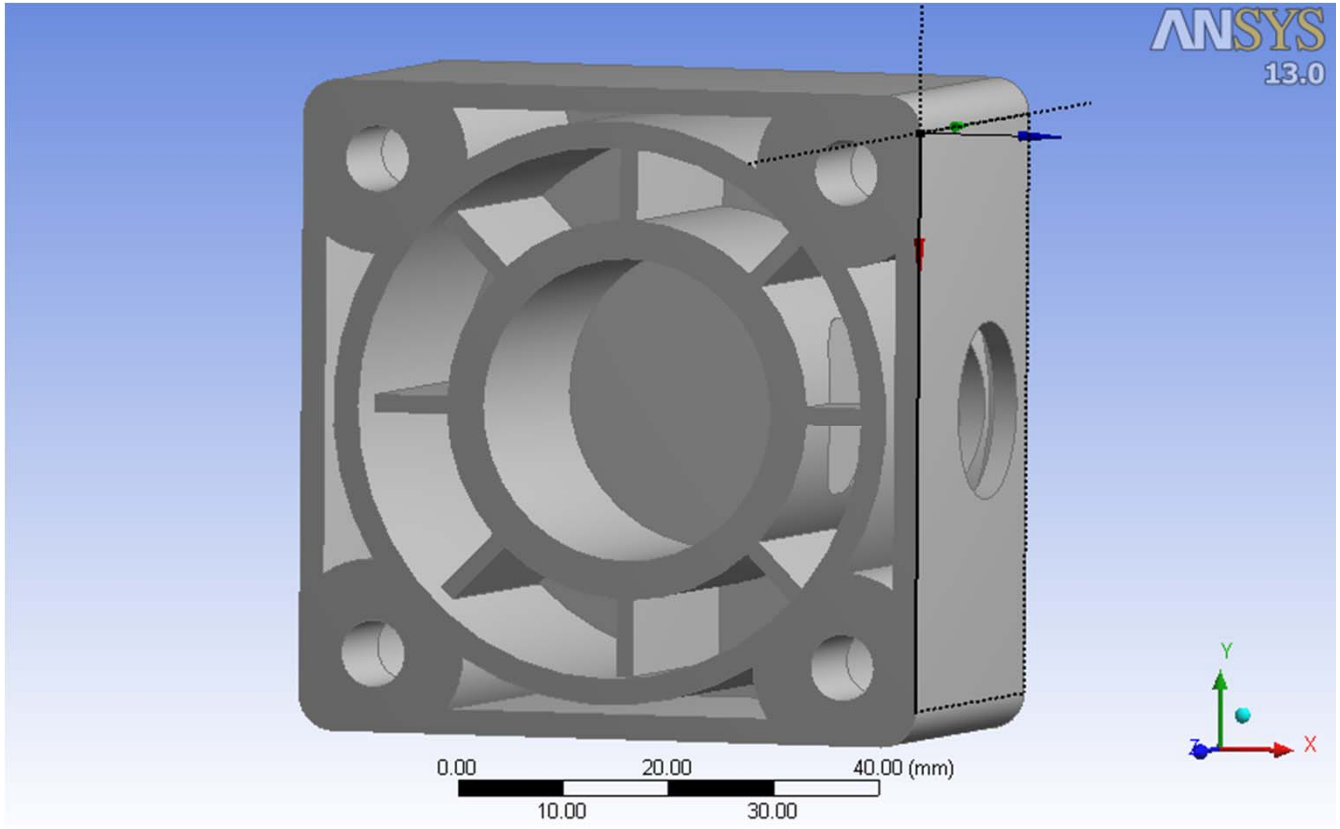
Create a new plane in one of the faces of the cover. Be sure that the axis look like the ones drawn in the picture.

Geometry Definition



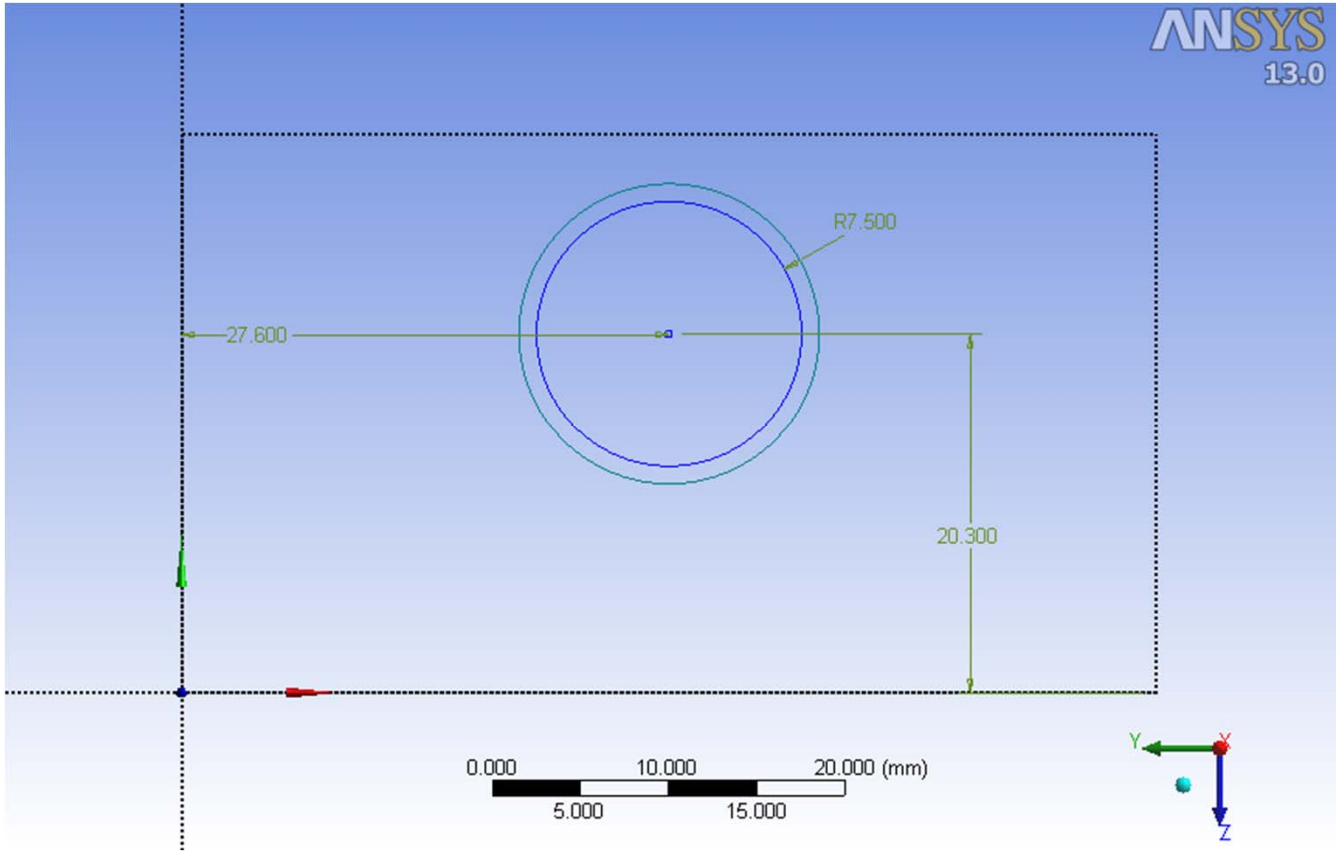
In this new plane create the new sketch 6 shown in this figure

Geometry Definition



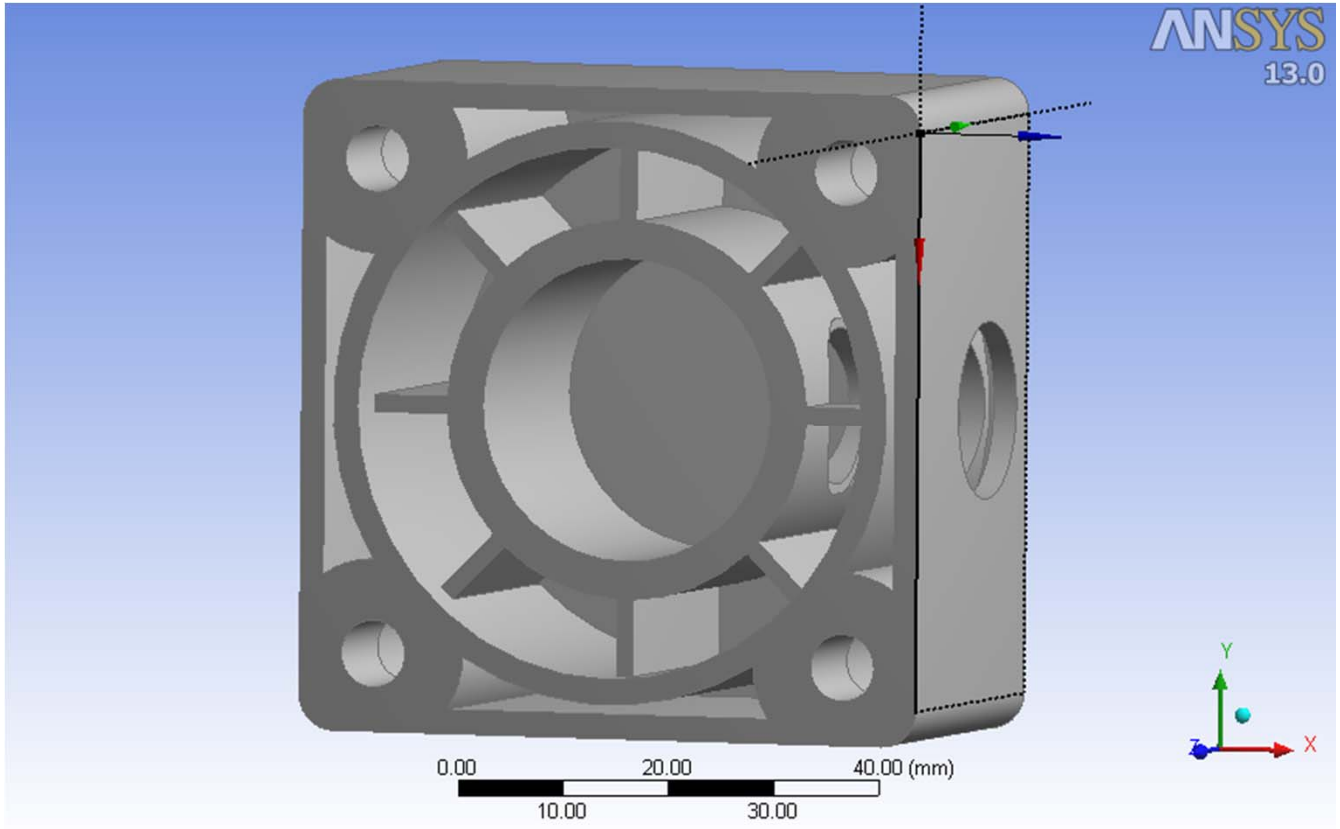
Extrude sketch 6, with the remove material option, 15mm

Geometry Definition



Create a new sketch (sketch 7) in the last created plane

Geometry Definition



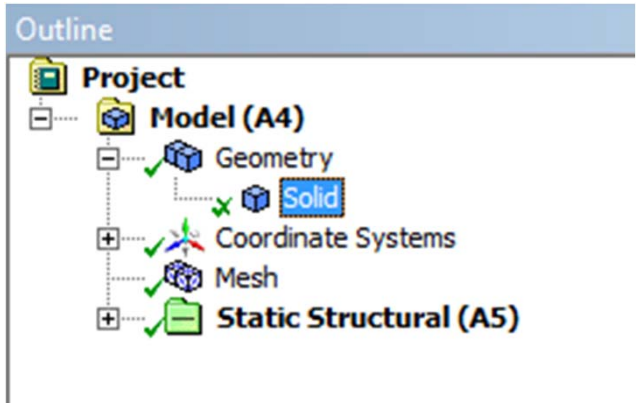
Extrude skectch 5 31mm with the *remove material* option

Ansys Mechanical

At this stage we have the geometry completely defined.

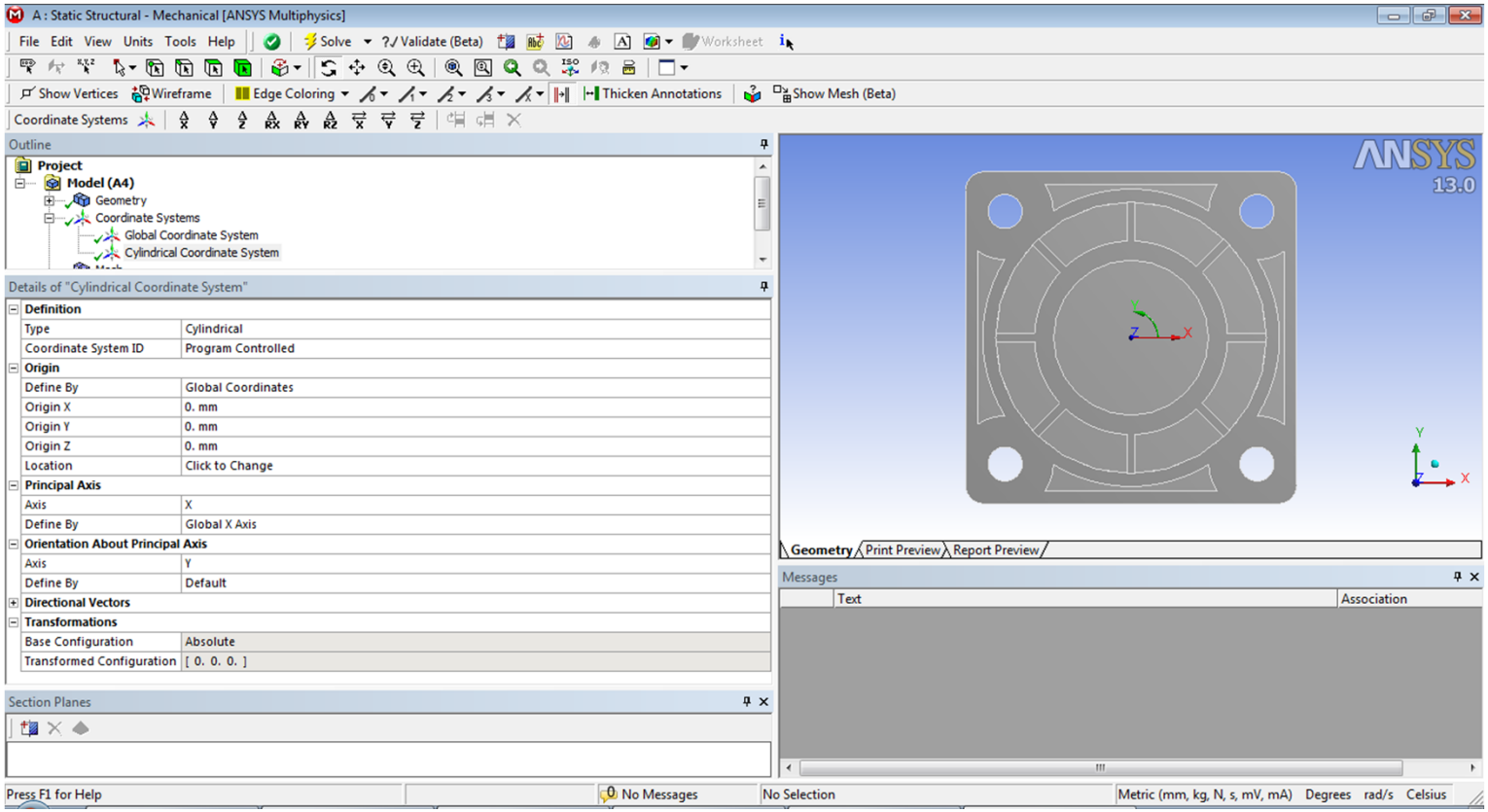
We can open Ansys mechanical and start defining the material, mesh and boundary conditions.

In Geometry/Solid, we will change the material of our structure to the previously defined plastic



Cylindrical Coordinate System

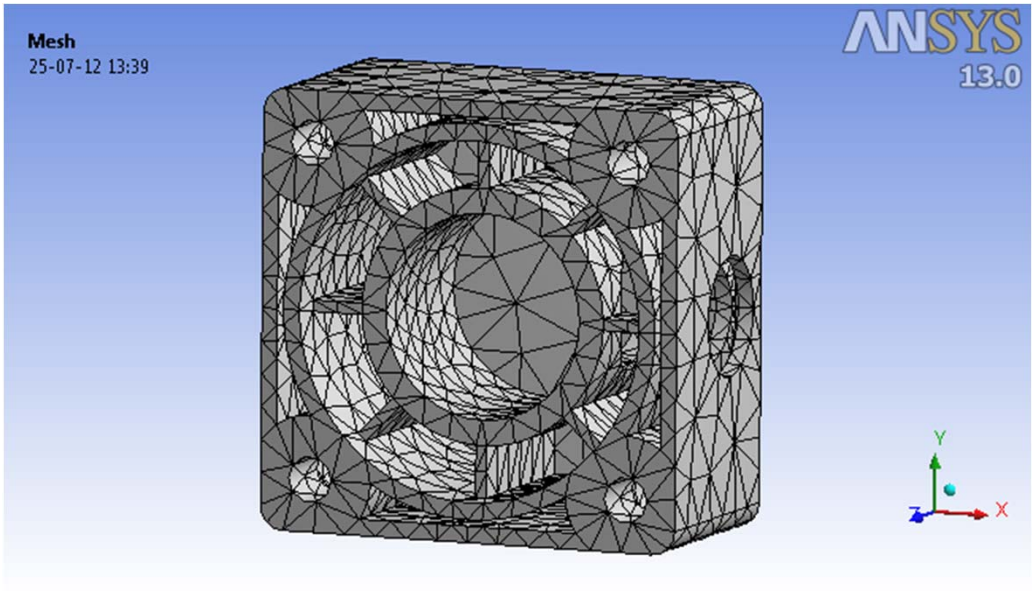
We will define the following cylindrical coordinate system, that will be used for result visualization.



Mesh

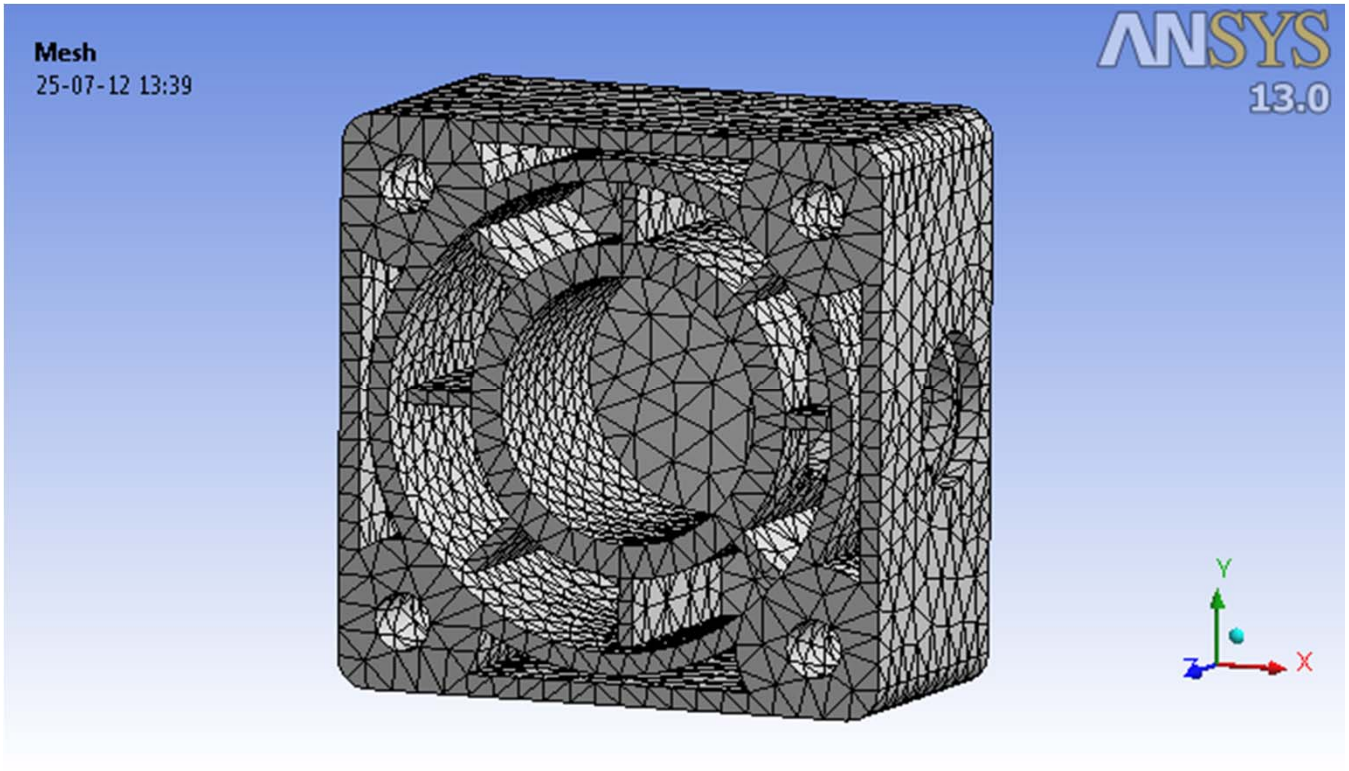
To solve this example we will use the mesh suggested by ansys. Modifying only the Relevance parameter.

In *Preview Surface Mesh* we can see the mesh suggested for a relevance value of zero:



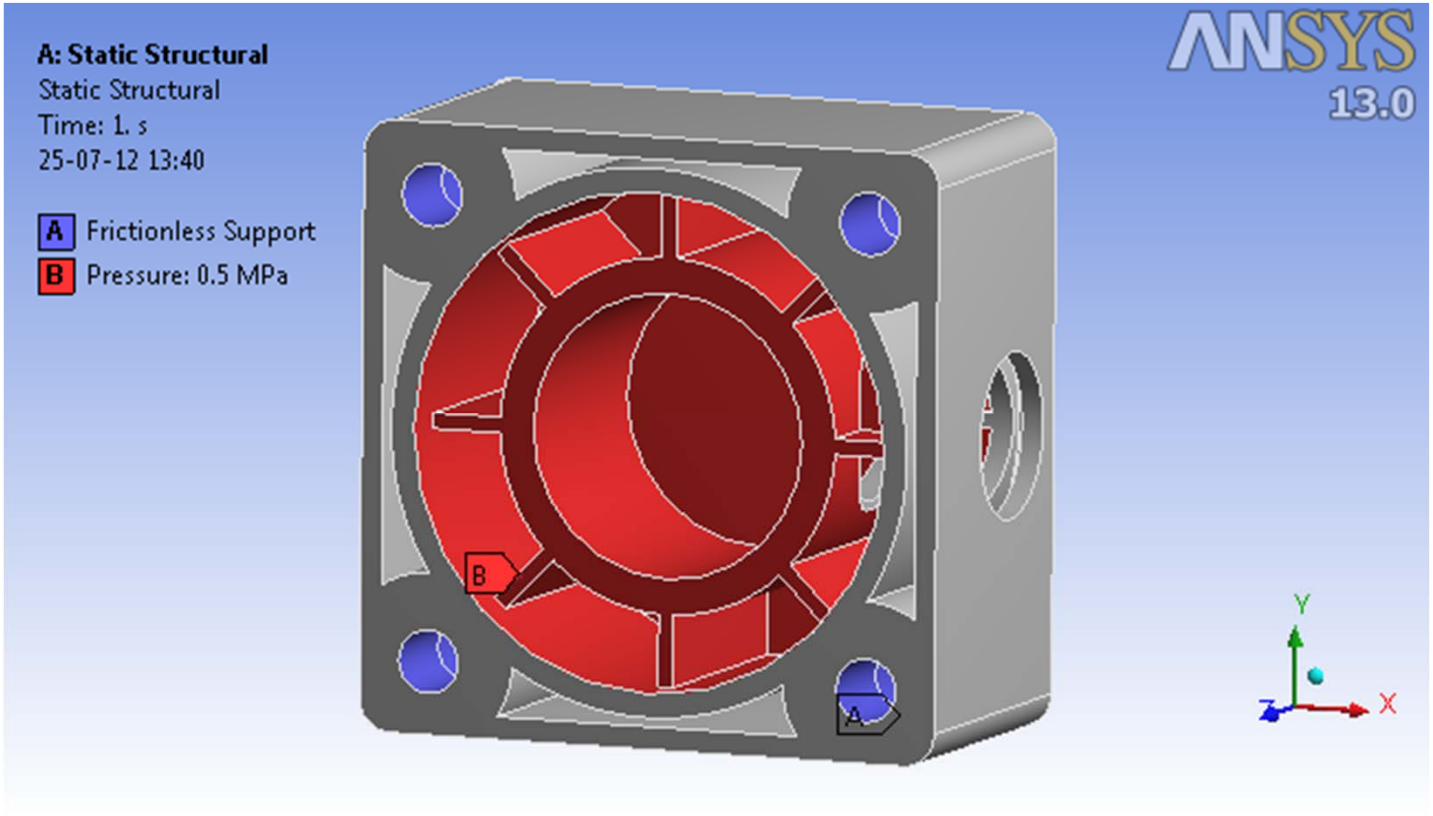
Mesh

We will increase the relevance to 100 to obtain the following mesh:

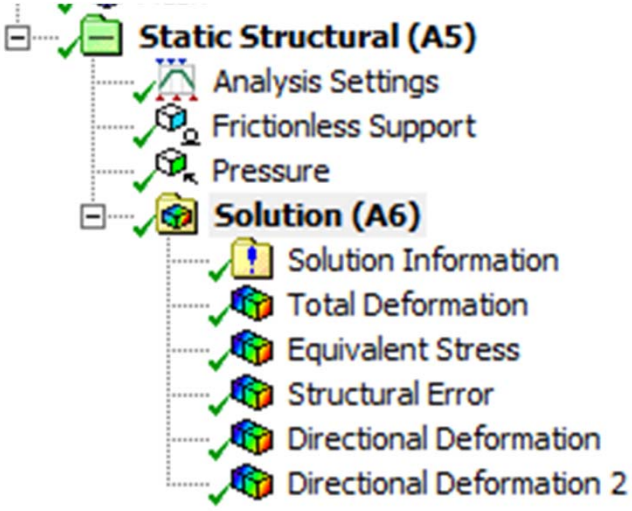


Boundary conditions

We will apply the frictionless support to a total of 12 faces and the 0.5MPa pressure to a total of 39 surfaces



Results



Details of "Directional Deformation 2"	
Scope	
Scoping Method	Geometry Selection
Geometry	1 Face
Definition	
Type	Directional Deformation
Orientation	X Axis
By	Time
Display Time	Last
Coordinate System	Cylindrical Coordinate System
Calculate Time History	Yes
Identifier