



Place Features: Inventor 2008

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In this article we shall focus on how to place simple 3D Primitive objects in Autodesk Inventor 2008. Primitive features for 2D include – point, line, arc, polygon etc, however 3D Primitive objects include Cube (Block), Cylinder, Sphere, Pyramid, Torus etc,

This method of placing features leads to quick, easy and accurate part modeling. Let's have a look at the method of creating a cube by place feature technique. Further you will notice how to create a Cylinder, Sphere, Pyramid, Tours and other shapes using place feature option available in Autodesk Inventor 2008.

Let's go ahead with the tutorial,

Objective: To create a CUBE (BLOCK) of sides 50mm.

There are two methods available to create the block. Let's go ahead with the first method,

Method – I

Step 01: Start Autodesk Inventor 2008, and select the default template as part modeling. I have selected (Metric) Standard (mm) – millimeters

Step 02: Once you have opened Sketch Panel, scroll down the 2D Sketch panel to find the Place Feature as shown in figure 01

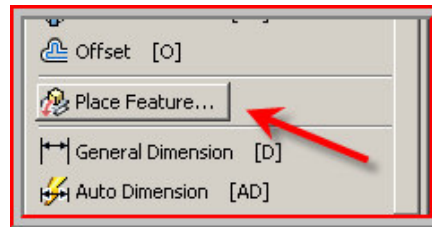


Figure 01 Selecting Place Feature

Step 03: The place feature window opens for the user to select the type of 3D primitive he wants to create as shown in figure 02

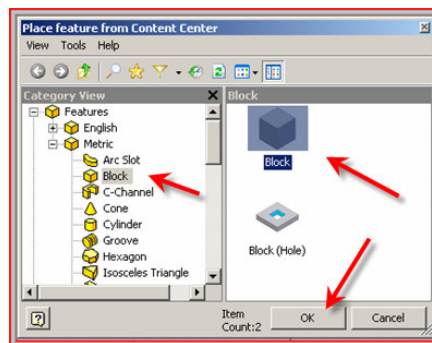


Figure 02 – Place Feature Interface

Step 04: Now select block and solid block and click on OK as shown in figure 02. However you create Boolean operations on the part file by selecting the appropriate tools.

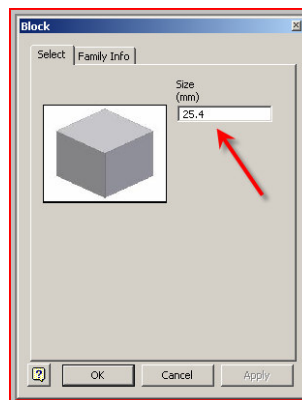


Figure 03 – User Input Dialogue Box

Step 05: Now you will be prompted with user input dialogue box to enter the size of the block. (Length X Width X Height) as a single value as shown in figure 03, also enter the value as 50mm

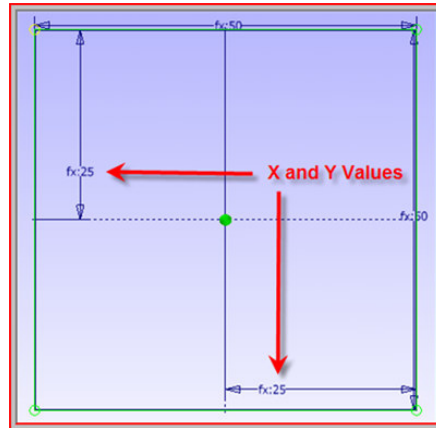


Figure 04 – 2D Sketch

Step 06: Now right click and select done if the values of X and Y are correct as shown in figure 05. In case you need to alter you can click on the dimensions and change them accordingly.

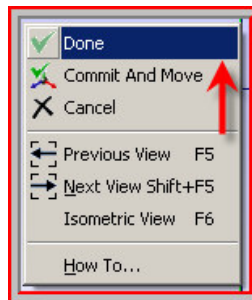


Figure 05 – Selection option

Step 07: Now press Isometric View to see the Block as shown in figure 06

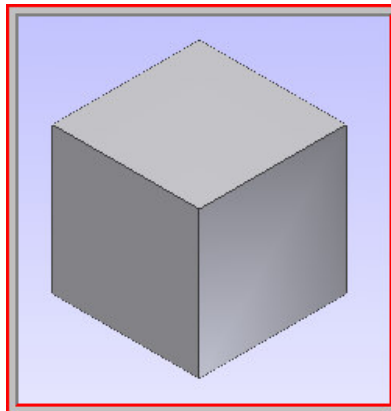


Figure 06 – BLOCK

Step 08: However, in case you create the same block using the regular method, the time required is more. This includes creating the sketch, dimensioning, finish sketch and later use the extrude command to create a block height of 50mm.

Now let's have a look at how sphere can be directly created without creating a sketch and performing revolve option.

Creating Sphere:

Step 01: Click on Place Feature and select Sphere as shown in Figure 07

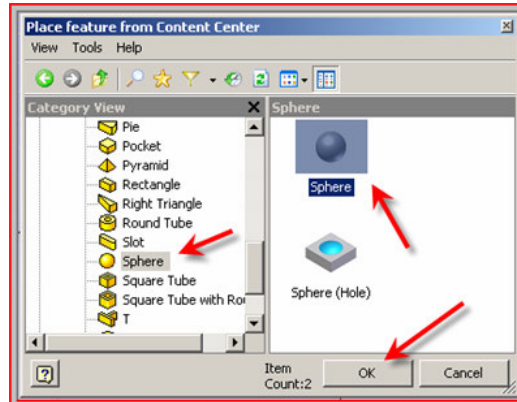


Figure 07 – Select Sphere option

Step 02: Once you click on OK, enter the diameter as 50mm as shown in figure 08 and click on OK, further click on done to get a sphere of diameter of 50mm as shown in figure 09

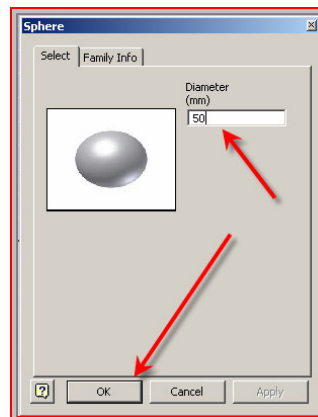


Figure 08 – Input Dialogue Box



Figure 09 – Sphere



Further the following feature can be created without creating a sketch

- Arc Slot
- Block
- C – Channel
- Cone
- Cylinder – can be used for creating shaft
- Groove
- Hexagon – can be used for creating Head of Bolt
- Isosceles Triangle
- Jog
- Key 1 – Flat
- Key 2 – Flats
- Key Round
- Key Square
- L
- Parallelogram
- Partial Cone
- Partial Round Tube
- Partial Torus
- Pie
- Pocket
- Pyramid
- Right Rectangle
- Round Tube
- Slot
- Sphere
- Square Tube
- Square Tube with Round
- T
- Torus
- Trapezoid

Note: These tools can be used as building blocks to create a final part model.