Creating a Simple Object “BLOCK”

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The object that will be created in this exercise.

STARTING TO CREATE A PART

Click Create new object button shown above, and New window appears.

Note:

Choose Open icon if you wish to continue working on the unfinished part.
The Three Datum Planes (FRONT, TOP & RIGHT) and Coordinate System PRT_CSYS_DEF appears as shown below.

Type in block as the part name.

Then click OK button.
CREATING A PROTRUSION

From **PART** menu, select

**Feature -> Create -> Solid -> Protrusion**

**Extrude -> Solid -> Done**

**One Side -> Done**

You will be prompted to select or create a **SKETCHING PLANE**.

Pick **FRONT** datum plane as sketching plane, then choose **Okay** to accept the direction of feature creation.

Then pick **TOP** datum plane on the graphic window.

The Sketcher – drawing window appears as shown below.
Select icon

Create lines icon.

Create rectangle icon.

Create circle icon.

Create an arc icon.

Create fillet icon.
Click **Create rectangle** icon, and sketch the rectangle as shown below.

Choose **Create a fillet** icon.

Pick on the top and right lines of the rectangle, then pick on the top and left lines.

Your sketch should look like below.
The geometry has been automatically dimensioned, but if you choose to re-dimension the geometry, choose **Dimension** icon shown on the left.

To dimension an arc, click the left mouse button on the arc, then position the cursor to desired location on the sketch and click the middle mouse button. Similarly applied to dimension a line.

To dimension distance between two lines, click the left mouse button on the first line, then the second line, and position the cursor to desired location on the sketch and click the middle mouse button.
MODIFYING DIMENSIONAL VALUES

Choose Modify icon shown on the left.

Click on the numerical values of ALL dimensions, and Modify Dimensions window appears.

Make sure to check off the Regenerate box.

![Original values and Modified values comparison]

Click the Check button.

CREATING EXTRUSION

Click the Check icon to continue with the current section.

Blind -> Done

You will be prompted in the message window to enter the depth.
Type in 10, then hit Enter.

Click OK button.

The resulted solid object is shown below.

SAVING THE PART

To save the object, click Save button shown above.

ADDING A HOLE

From PART menu, select

Feature -> Create -> Solid -> Hole

The Hole window appears, and set the following:
Click the above icon to complete hole creation.

The resulted object is shown on the left.

**CREATING A SIDE CUT**

From **FEAT** menu, choose,

*Create -> Solid -> Cut*
*Extrude -> Solid -> Done*
One Side -> Done

When prompted to select or create a sketching plane, pick the right surface of the block.

Okay – to accept the direction of feature creation. The arrow must point into the object.

Select **TOP** surface of the block as **TOP** reference plane for orientation.

Choose **Create Lines** icon to draw two vertical lines – see figure below.

Choose **Create an arc** icon to draw the arc that connect the bottom two end points of the line – see figure below.

Choose **Dimension** icon to dimension the section you just created.
Choose **Modify** icon to modify the dimensions as to be shown in the following:
Click **Check** icon to continue with the current section.

Click **Okay**

**Thru All -> Done**

Click **OK** button.

Click **Saved view list** icon, then choose **Default** view.

Shade the object.

**ADDING THE TEXT**

From **PART** menu, select

**Feature -> Create -> Solid -> Protrusion**

**Extrude -> Solid -> Done**

**One Side -> Done**

You will be prompted to select or create a SKETCHING PLANE.

Pick FRONT surface of the block, then choose **Okay** to accept the outward direction of feature creation – see Figure below.
Choose TOP as horizontal reference for

Then pick TOP surface of the object – see Figure below.

Pick this TOP surface as TOP reference plane.

Pick this FRONT surface as sketching plane.

The Sketcher – drawing window appears as shown below.
From pull-down menu **Sketch**, select **Text**

You will be prompted with,

Select start point of line to determine text height and orientation.
   Pick a point located at lower left corner of the object - see Figure.

Select second point of line to determine text height and orientation.
   Pick another point located vertically above the first point - see Figure.

On the Text window, type in "Your Name" in the box below Text Line.

Click **Check** button.
Click **Check** icon to continue with the current section.

Click **Okay**

**Blind** -> **Done**

Enter Depth – **0.5**, and hit **Enter** or click **Check** button.

Click **OK** button.
COLORING THE PART

From pull-down menu **View**, select

**Model Setup -> Color & Appearances**

The **Appearances** window appears.

Click **Add** button.

The **Appearances Editor** appears.

Click this button to set colors.
The **Color Editor** window appears.

Click **Color Wheel** bar.

The **Color Wheel** will be expanded.

Choose any color you wish on the color wheel e.g. light green.

Click **OK** button.

Click **Add** button.

Repeat the same procedure to add another color e.g. red.

After your finish to add colors, click **Close** button.

If you want the color of the object to be for example, light green, click light green color button, then click **Set** button. The whole object will be in light green color.

To change the front surface of "Your Name" to be red color, click the **red** button, and then under **Set Object Appearance**, change **Part** to **Surfaces** as shown in the Figure below.
Carefully select the front surface of your name as shown below.

If necessary use **Query Sel** to pick the correct surface.

Click **Done**

Click **Set** button.

Click **Close** button.

The resulted object is as shown below.

Save the part.