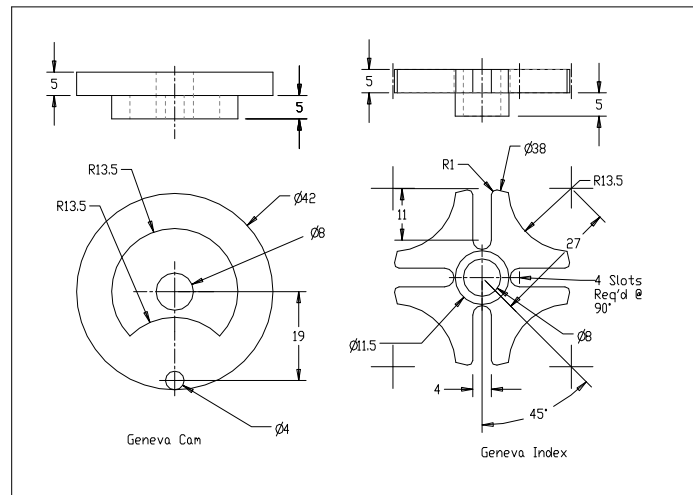


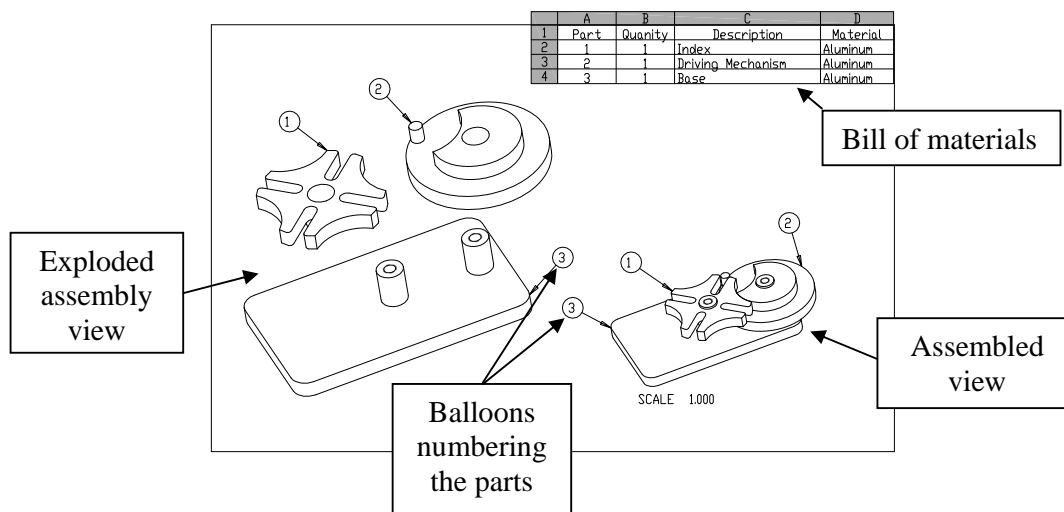
## 8 – Working Drawings in AutoCAD

Most engineering designs consist of more than a single part. Usually there are a several or many parts that must fit and work together. When we are creating the drawings of a design, we must create dimensioned drawings of each part so that they can be manufactured correctly and we must also show how the parts fit together so they can be assembled correctly.

The drawing of a single part is called a detailed drawing. It contains one or more orthographic views of the object, dimensions and possibly sectional and isometric views. In essence, a detailed drawing may contain all of the elements we have been studying this semester. An example of a detailed drawing is shown below.



The drawing above shows two parts. Each part is shown in two different views, a front and a top view using a third angle projection. This is a detailed drawing of the two parts. It contains sufficient information to build the parts. It does not tell us how to assemble the parts. That is shown in the drawing below.



**Note:** The assembly views in the drawing above are shown as isometric or pictorial views. This is usually not done in AutoCAD drawings. AutoCAD assembly views are usually orthographic because of the difficulty in drawing pictorial views.

The Bill of Materials (BOM) lists all of the parts needed to make the machine. Each part is numbered in the BOM and the same number is used in the balloons that point to each part. The bill of materials lists the part number, the name of the part, and the material from which the part is made.

Both exploded and unexploded assembly views are shown in this particular example but if the assembly can be fully explained with a single view, then only one view need be shown. The view may be similar to one of the ones in the drawing above or it may be a sectional view. Sectional assembly views are especially useful if some parts are inside of housings or are concealed from view.

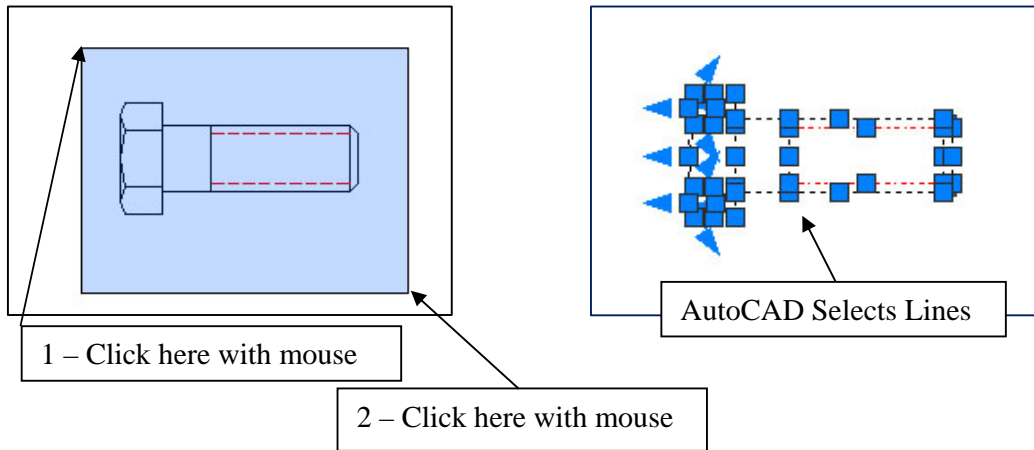
A set of working drawings consists of a detailed drawing for each part that must be manufactured and sufficient assembly drawings to explain how the parts are assembled.

### **Detailed Drawings**

The first step in creating a set of working drawings is to create the detailed drawings showing each part. Here you usually show a front view and possibly other views (top view, right side view, left side view, etc.) if they are required to fully describe the geometry. Sectional views and isometric views may also be included if the geometry is not easily understood from the orthographic projections. Dimensions are included on these drawings so the part can be manufactured correctly. The title block on the drawing should contain the name of the part and this should be the same name used in the Bill of Materials.

After the detailed views are drawn, you can create the assembly drawings. These drawings are created by combining views from the detailed drawings to show how the parts are assembled. The process involves several simple steps.

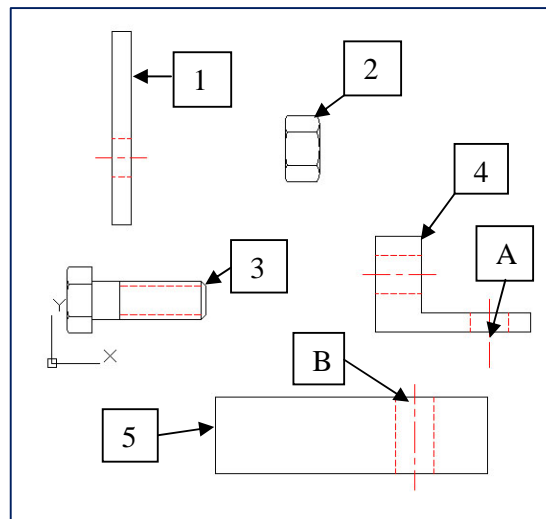
1. First you need to create the assembly drawing. You can do this by opening the Standard\_A4 drawing sheet (You may have named it differently.) Once it is open, remember to use “Save as” to save the drawing under a different name. I usually use the abbreviation “\_ASM” as the last characters in the file name so that I will know that this is an assembly drawing.
2. Open the detailed drawing to show the view of the part you want to insert into the assembly drawing.
3. Turn off all layers that are not needed in the view so that what is left can be easily selected. Dimensions and projection lines are not usually shown in assembly drawings so these layers should be turned off.
4. Select the view you want to copy to the assembly drawing by dragging a box around it with the mouse. Move the mouse to a location to the left and above the view and click. Next, move the mouse to a location to the right and below the view and click again. AutoCAD will select all of the objects in the view.



5. Hold down the CTRL key and press the C key. This will copy everything you have selected to the program clipboard.
6. Next, you switch from the detailed drawing to the assembly drawing you are creating. You can do this by clicking on the big red “A” in the upper right corner and select the assembly drawing in the list of open files.
7. Paste the view from the detailed drawing into the assembly drawing by holding down the CTRL key and pressing the V key. Use the mouse to move the view to a convenient location then click to complete the copy.
8. Repeat this process for all of the parts in the assembly. When you have finished, you should have a drawing containing copies of all of the parts in the assembly. The figure below shows a bracket that is used to bolt two plates together. We will use this as an example.

### Building the Assembly

After the parts have been copied to the assembly drawing, we can arrange them so that they are oriented correctly. We start by connecting Parts 4 and 5. Use the MOVE command to move Part 4 so that points “A” and “B” are at the same location. Point “A” is located at the intersection of the centerline and the bottom of Part 4 and point “B” is located at the intersection of the centerline and the top of Part 5.

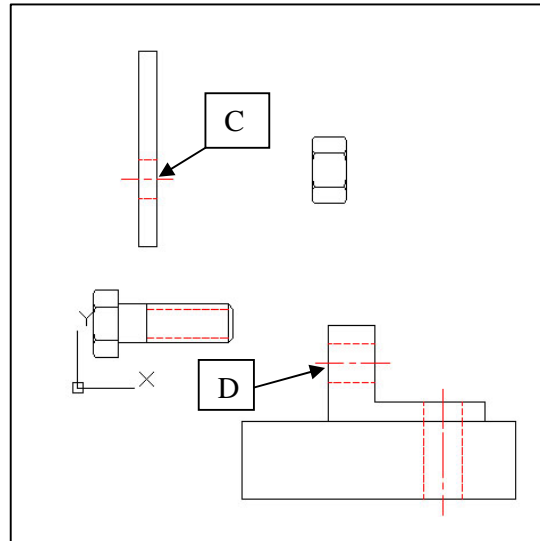


**MOVE**

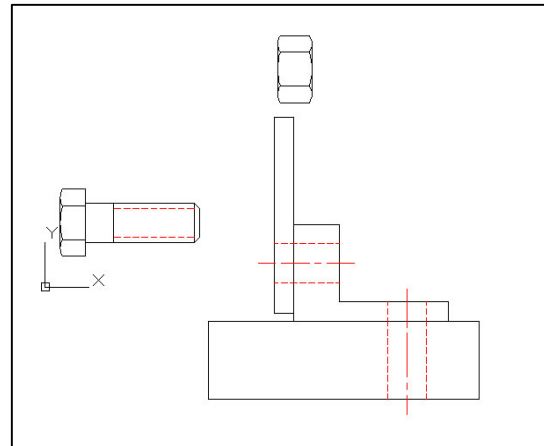
Select objects: **w**  
Specify first corner:  
Specify opposite corner:  
Select objects:  
Specify base point  
  
Specify second point

*{Part 4 will be selected with a window}*  
*{Click above and to the left of Part 4}*  
*{Click below and to the right of Part 4}*  
*{Press enter – the selection is complete}*  
*{Click on Point “A” at the intersection of the hole centerline and the bottom of Part 4}*  
*{Click on Point “B” at the intersection of the hole centerline and the top of Part 5}*

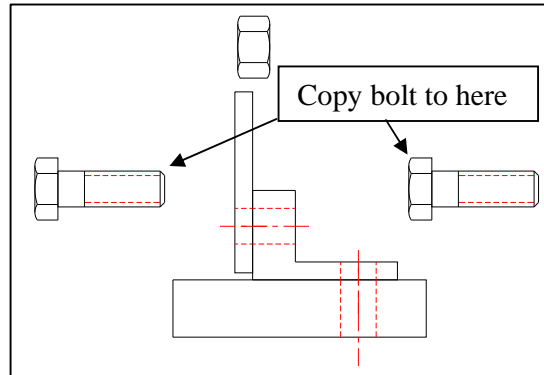
The resulting placement is shown in the figure on the right.



Next we will align point “C” with point “D”. Point “C” is located at the intersection of the centerline with the right side of Part 1. Point D is located at the intersection of the centerline and the left side of Part 4 as shown in the figure on the right. Here again, use the **MOVE** command to move Part 1 into the correct location.



Next, create a copy of the bolt and insert the copy into the vertical hole. The copy is created with the **COPY** command as shown below.



#### **COPY**

Select objects: **W** *{select the bolt with a window}*  
 Specify first corner: *{click above and to the left of the bolt}*  
 Specify opposite corner: *{click below and to the right of the bolt}*  
 Specify base point: *{select a point on the bolt}*  
 Specify second point *{select a point away from all other objects to place the bolt}*  
 Specify second point or [Exit/Undo] <Exit>: *{press enter}*

The results of the copy are shown in the figure above. The bolt on the right will be rotated 90 degrees clockwise and inserted into the hole. The first step is to rotate the bolt which is done with the **ROTATE** command. This command also requires that you select all of the objects to be rotated and you can again use the **W** option to create a selection window. When you rotate the bolt, be careful not to rotate it over another part. You can control the rotation by picking a base point (center of rotation) that is near the end of the bolt

#### **ROTATE**

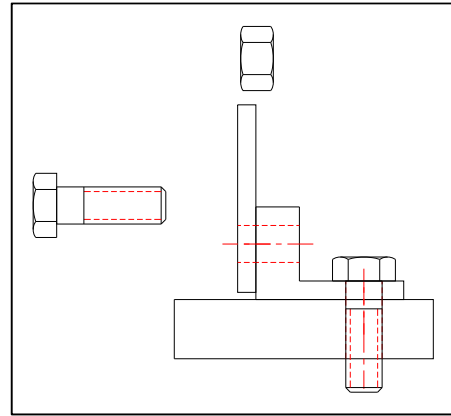
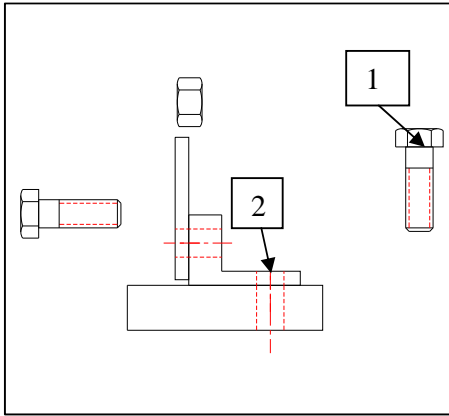
Select objects: **W**  
 Specify first corner: *{click above and to the left of the bolt}*  
 Specify opposite corner: *{click below and to the right of the bolt}*  
 Select objects: *{press enter}*  
 Specify base point: *{click near the bottom to keep the bolt from rotating over another part. In this case pick point 1}*  
 Specify rotation angle: **-90** *{the angle is negative because the rotation is clockwise}*

Now we insert the bolt into the vertical hole. Use the **MOVE** command and select a mid point on the bolt head (point 1) and the intersection of the centerline and the top of the hole (point 2) as the base point and the second point in the **MOVE** command.

#### **MOVE**

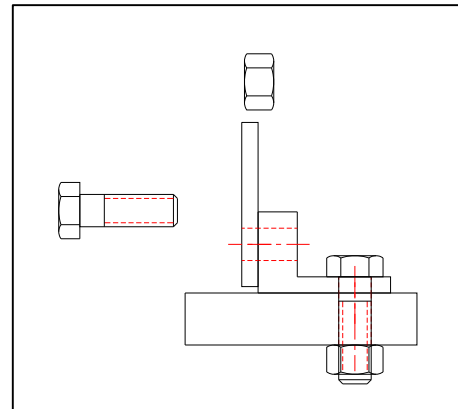
Select objects: **W**  
 Specify first corner: *{upper left}*  
 Specify opposite corner: *{lower right}*  
 Select objects: *{press enter}*

Specify base point: **mid** *{enter mid and select point 1}*  
 Specify second point: *{let OSNAP find the intersection of the top of the hole and the centerline – point 2}*



When you have finished, your drawing should resemble the one shown above on the right.

Next copy the nut, rotate it -90 degrees, and place it on the bolt as shown in the figure at the right. You can use the **COPY**, **ROTATE**, and **MOVE** commands just as you did for the bolt. The resulting drawing is shown on the right.

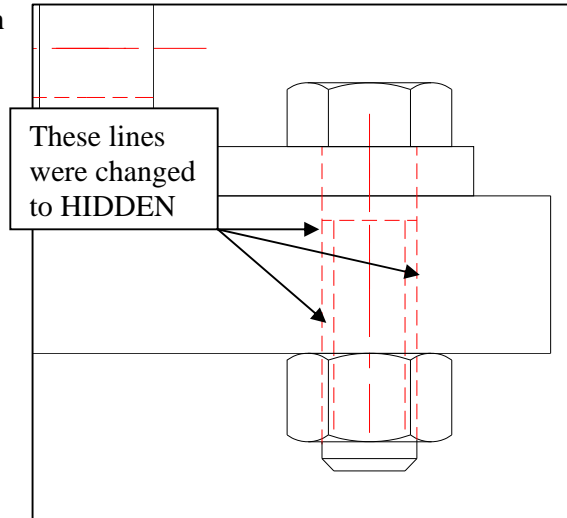


Some of the interior lines of the bolt, should be changed to hidden lines. You can make these changes with the **CHANGE** command.

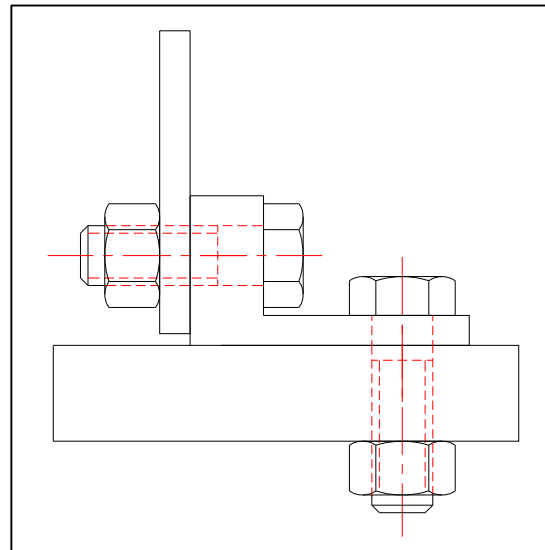
**CHANGE**  
*{Select the lines that should be changed to the hidden layer}*

```
Select objects: 1 found
Select objects: 1 found, 2 total
Select objects: 1 found, 3 total
Select objects: {press enter}
Specify change point or [Properties] ]: P {we are changing the properties}
Enter property to change [Color/Elev/LAyer/LType/... : LA {we want to change the layer}
Enter new layer name <Object>: hidden
Enter property to change [Color/Elev/LAyer/LType/... : {press enter}
```

The bolt with hidden lines with correct hidden lines is shown on the right.



Continuing moving and rotating parts until the parts are fully assembled and your drawing looks like the one on the right.



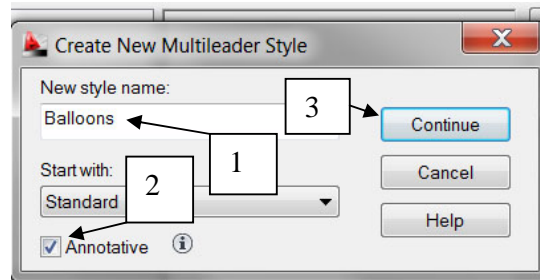
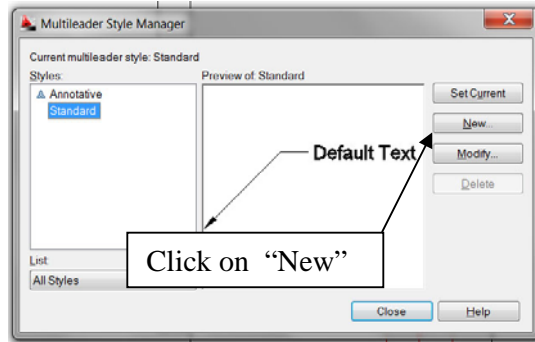
## Creating Balloons

Next, we need to create the balloons that identify parts. Balloons are called leaders in AutoCAD and we must first define balloon style leaders before we can draw them. Start by typing:

`mleaderstyle`

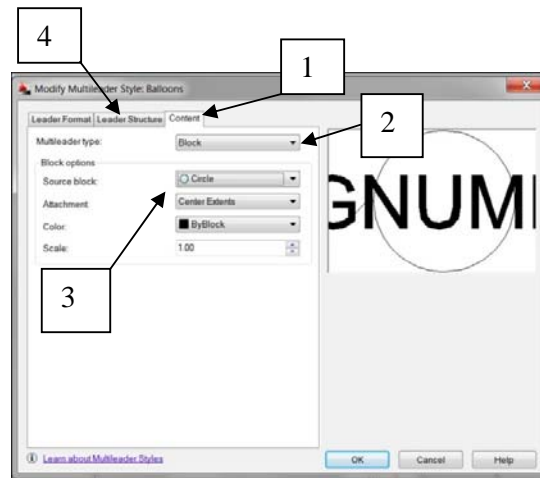
This produces the dialog box on the right. Click on the “New” button to create a new leader style.

1. Enter “Balloons” for the name of the new leader style we are creating.
2. Mark the “Annotative” check box.
3. Click on “Continue”.



Next we define how the new multileader will look in the dialog box shown on the right.

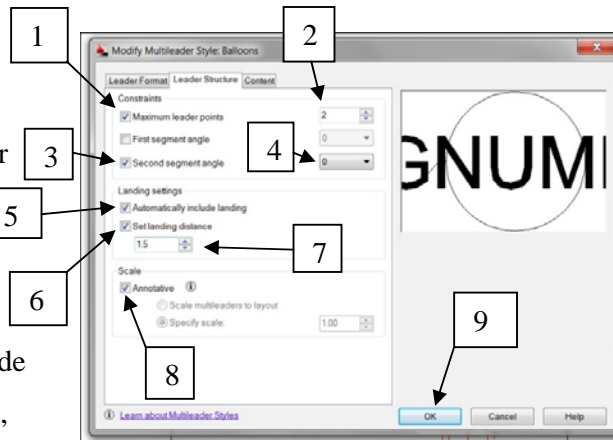
1. Click on the “Content” tab
2. Select “Block” for the multileader type
3. Select “Circle” for the source block
4. Select the “Leader Structure” tab





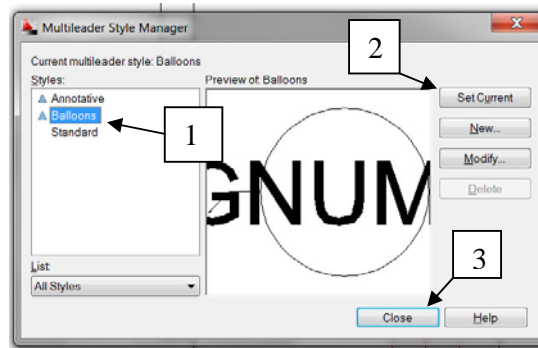
The settings in the “Leader Structure” tab are:

1. Mark the “Maximum leader point” check box.
2. Set the number to 2
3. Mark the “Second segment angle” check box
4. Set the angle to 0
5. Mark “Automatically include landing”
6. Mark “Set landing distance”
7. Set the landing distance to 1.5 or what seems reasonable. This is a personal preference.
8. Mark “Annotative”
9. Click “OK”



In the “Multileader Style Manager” dialog box:

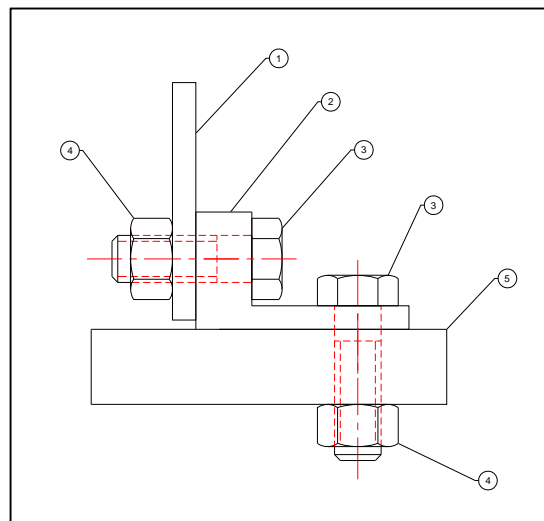
1. Make sure “Balloons” is highlighted.
2. Click on “Set Current”
3. Click on “Close”



Now you are ready to draw the leaders. Change the layer to “Dimension” layer and create the balloons by clicking on:

1. Click on the “Annotate” tab at the top of the window
2. Click on the “Multileader” icon in the annotate ribbon at the top of the window.
3. Click at a point on a part where you want the leader to start
4. Click where you want the balloon to be placed
5. Enter the number of the part.

Repeat the process above until all of the parts have been numbered. When you have finished, your drawing should look like the one on the right. The numbers may appear a little small but they are annotative and will have an appropriate size on the drawing sheet.

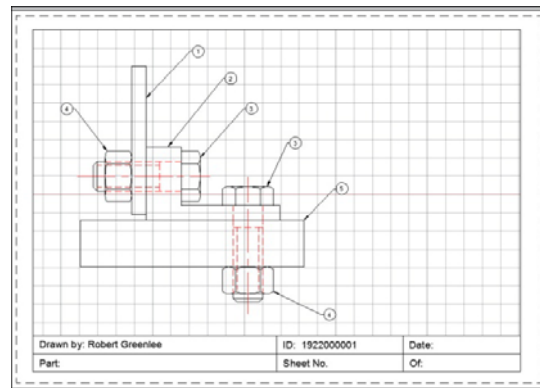


## Placing on the Sheet

Next we will place the drawing on the sheet. We do this using the same technique we have used before. Click on:

1. Click on the “Layout1” tab at the bottom of the window.
2. Double click in the drawing area to activate the viewport.
3. Enter the command **zoom all**
4. Use the mouse to drag the drawing to the correct location. Size the drawing by scrolling the wheel on the mouse.
5. If the arrowheads are too small, correct their size by modifying the Multileader style. Use the “Leader Format” tab to change the size of the arrowheads.

The drawing should be located as shown in the figure on the right. The assembly is placed on the left side of the drawing sheet to leave room for the bill of materials.



## Bill of Materials

Next we will create the bill of materials (BOM). The bill of materials is a table that lists:

1. The part number
2. The part name
3. The number of parts required for the complete assembly
4. The material from which the part is made. The material is not listed for purchased parts such as bolts, nuts, etc.

You can create the table with the TABLE command or you can just draw it. It is frequently simpler to just draw it and that is what we are going to do. We will use the OFFSET command to create this table.

**OFFSET**

Specify offset distance: 10  
Select object to offset:  
Specify point on side to offset  
or [Exit/Multiple/Undo] <Exit>: **M**

*{Distance between lines}  
{Click on the top border line}*

Specify point on side to offset:

*{Enter M to create multiple offsets using  
the 5mm offset distance}*

Specify point on side to offset:

*{Click below the top border line 6 times  
until all of the lines have been created}*

Specify point on side to offset:

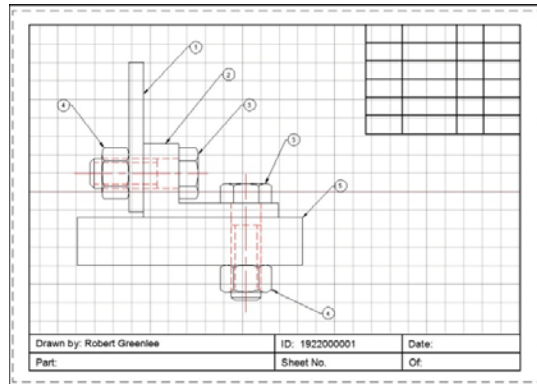
Specify point on side to offset:

Specify point on side to offset:

Specify point on side to offset:

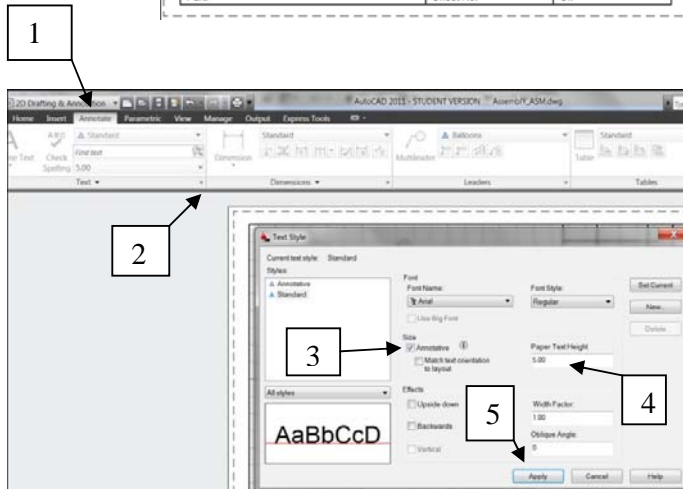
Specify point on side to offset or [Exit/Undo]: *{Press enter twice to  
exit}*

Next we create vertical lines by offsetting the right border line. Create lines at 20, 35, 65, and 85 from the right border. After you have created the vertical lines, use the **TRIM** command to trim the lines to form a table. The results are shown on the right.



Next, use the **MTEXT** command to fill in the table. You may need to set the size of text used in the table. Do this by clicking on:

1. Click on the “Annotate” tab.
2. Click on the “Text” arrow
3. Check the “Annotative” box
4. Set the text height to 5
5. Click on “Apply” then “Close”



Finally, fill in the table as shown below.

Part	Name	Req	Mat
1	Plate	1	Steel
2	Bracket	1	Steel
3	Bolt	2	
4	Nut	2	
5	Base	1	Steel

Drawn by: Robert Greenlee	ID: 1922000001	Date: 5 October 2010
Part: Plate Bracket Assembly	Sheet No. 1	Of: 1