

---

# 2D Drawing - Basic

In this task, you'll create a new 2D drawing of this object in ThinkDesign's drawing environment independently without referring to the model environment. We will use the **New Drawing** to introduce the basic ThinkDesign commands to create and edit 2D geometry. Command concepts in this task are: Groups, Insert from file, Insert dimensions, Edit dimensions, Edit properties, Style, Insert a title block, Property manipulation, Transform and Printing.

## Table of Contents

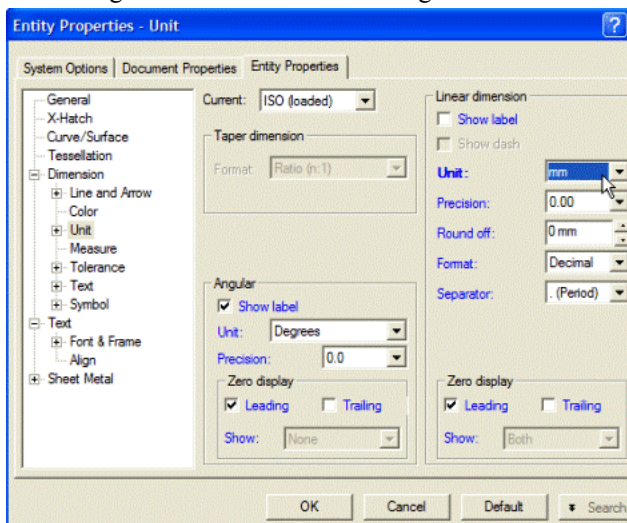
1. Step 1: Setting up the Drawing .....	1
2. Step 2: Inserting the section view .....	3
3. Step 3 Outlining the Top View .....	7
4. Step 4: Adding the Holes .....	9
5. Step 5: Adding and Modifying Groups .....	14
6. Step 6: Dimensioning (Smart and Ordinate ) .....	19
7. Step 7: Title Block and Printing .....	24

## 1. Step 1: Setting up the Drawing

Start by creating a **New Drawing**.

Before we get started let's make some changes to our drawing environment.

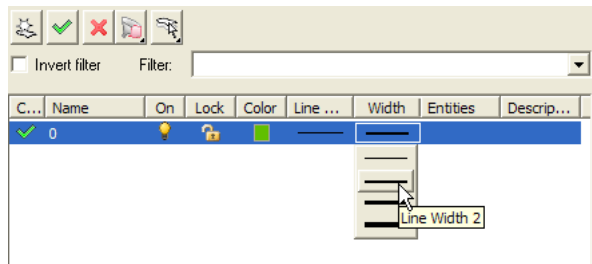
- Go to **Tools** → **Options/Properties**
- Select tab **Entity Properties** and click on **Unit**.
- Select **mm** as **Unit**
- Click **OK**.
- If the grid is on let's turn it off using **Grid Off**.



Now let's set the current environment defaults for basic properties, for entity creation, and then define a new set-

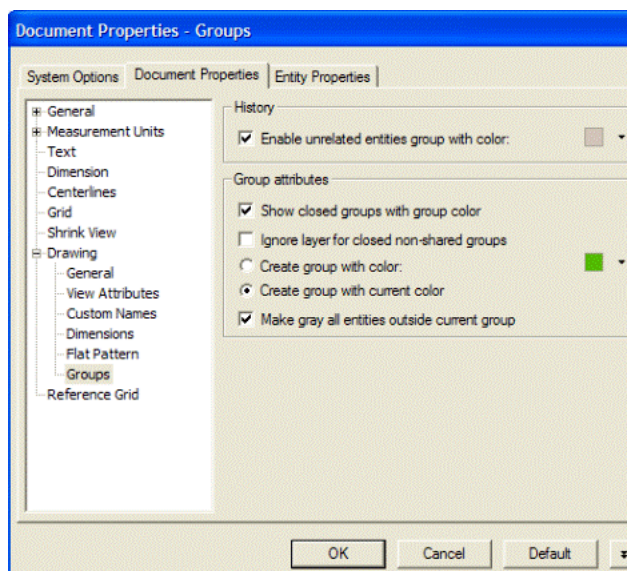
tings for dimensioning.

- Open the History Tree and Click on the Layer tab set the following attributes under Layer 0 as shown below.
- Set **Line Type** to 1 .
- Set **Line Width** to 2.
- Set **Color** to 3.



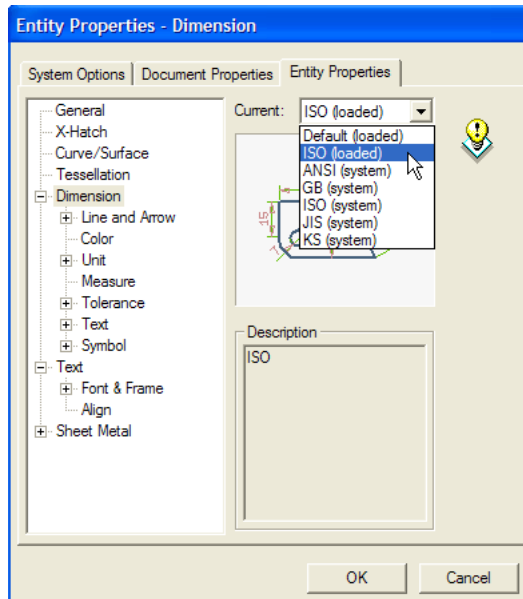
Later in this task we will be creating Groups. We do not want to show the Groups with Group color, we rather show them with the color that is current. Let's make this change.

- Right click on the workspace and select **Options/Properties...**
- Select Document properties.
- Go to Drawing -> Group Colors.
- Select the options as shown in the image below.
- Click OK.



See also: Properties (basic concepts)

In the beginning we set current the style as ISO using the tab Entity Properties -> Dimension.



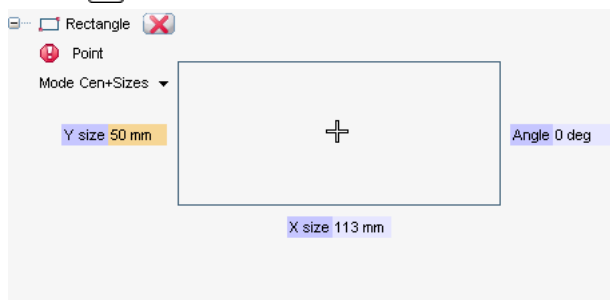
Dimensions are special entities. Using “Entity Properties” on Dimensions, it is possible to set the “Style”, like appearance, dimension colors, dimension orientation and position, arrowheads, units of measure, and text height.

## 2. Step 2: Inserting the section view

In this step, we will create a rectangle that we'll use for the top view of our part. Then we'll insert a section view that is already created and align it with the rectangle.

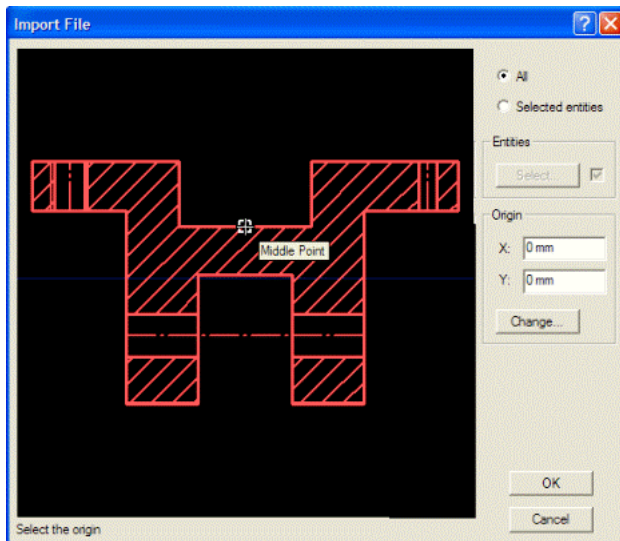
Draw a rectangle.

- Start the **Rectangle** command.
- Set the Mode to Cen+Sizes.
- Set the X Size 113mm and the Y Size 50mm.
- Click anywhere in the workspace to place the rectangle.
- Hit **Esc** to end the command.

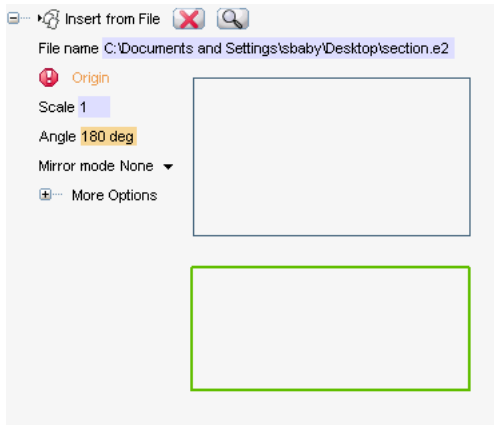


- Use **Insert From File** and browse to the downloaded folder for the file named Section.e2. This file will reside in the files folder inside the corresponding task folder wherever it is downloaded.
- In the import file window select **All**.
- Set the origin by pressing the Change button and then selecting the mid point of the horizontal line as shown.

- Click OK.



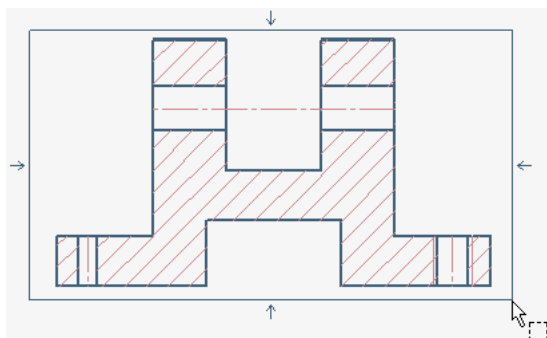
- Set the angle as 180 deg under the Insert from File selection list.




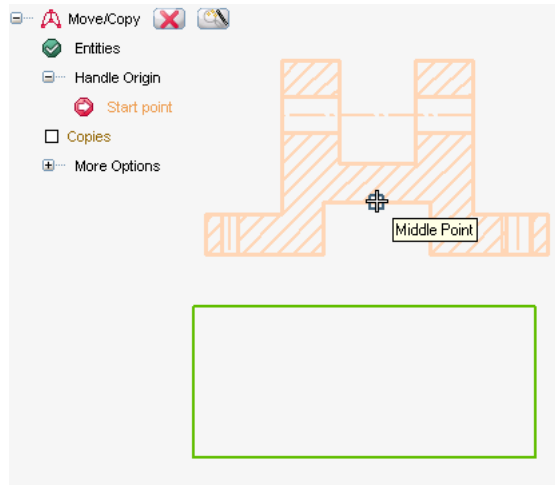
- Click on the workspace to enter the positioning point as shown (doesn't have to be exact, we'll adjust the position of it in the next step).
- Hit **[Esc]** to end the command.


Now we need to align the section view in the 'X' axis with our rectangle.

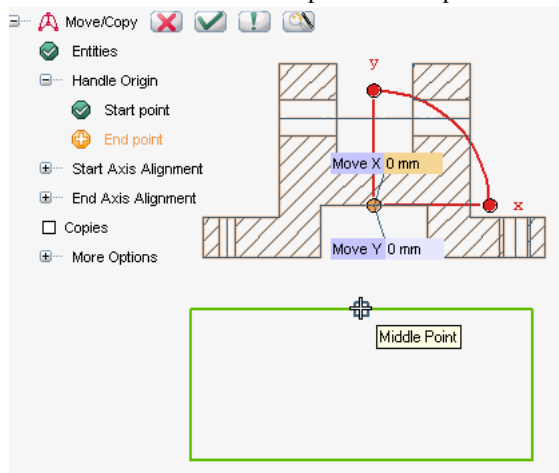
- Select, **Edit** → **Move/Copy**.
- **Select Window** around the entire section view to select it.






- After selecting the view, ThinkDesign will place the start point of the handle in an arbitrary place. To move the handle origin; in the selection list, right click on the  Start Point and select Reset, then snap to the mid point of the horizontal line of the section view as shown.

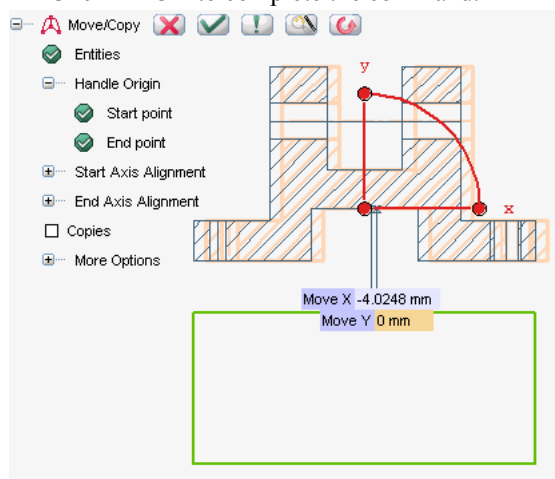


- For the  End Point snap to the mid point of the upper horizontal line of the rectangle, as shown below.



- Since we only want to move the view in the 'X' axis, hit the  key to highlight the Move Y mini dialog box and enter 0. Hitting the  key again, enters the Move Y mini dialog box value and displays the preview.

- Click  OK to complete the command.

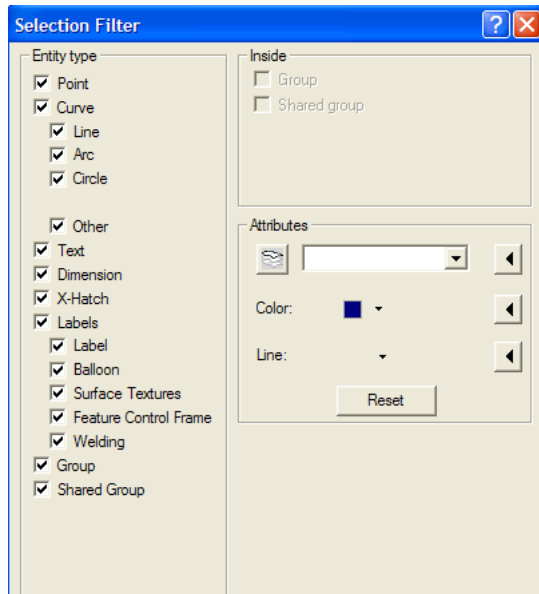


**NOTE:**

After coming out of the Move/Copy command the section view will remain highlighted. To de-highlight it, click on the background or hit the **[Esc]** key.

The lines that make up the section view are in a different color from the rectangle. Let's change them to green.

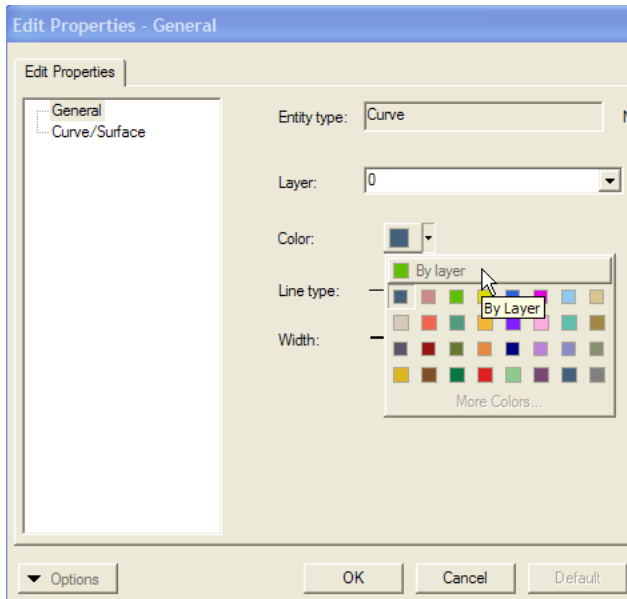
- Right click on the workspace and select **Select Filter**.
- In the color filter choose Color 1 (which is black in this task). The color on your display may be different depending on your **Colors and Materials** settings. Click **[X]** Close Window to close the Selection Filter window. The selection filter is now set to Color 1.
- Right click anywhere in the workspace and use the **Select All** command.



Now change the Color property.

- Right click on the workspace and select **Properties** from the context menu.
- Go to Entity Properties -> General and select Color 3.

Concept of Entity property: Every entity has its own set of properties and those properties will depend on the kind of entities used in drawing. ThinkDesign enables you to modify properties of existing objects or set the current values of properties that will be applied to all new entities. If you pre-select entities and then call the Properties dialog from the context menu, you can edit the properties of the pre-selected entities. If however, you select no entity but call the Properties dialog, then you can set the values of properties for all new entities. It is important to remember that the displayed Entity Properties dialog is different and depends on whether an entity is pre-selected or not.

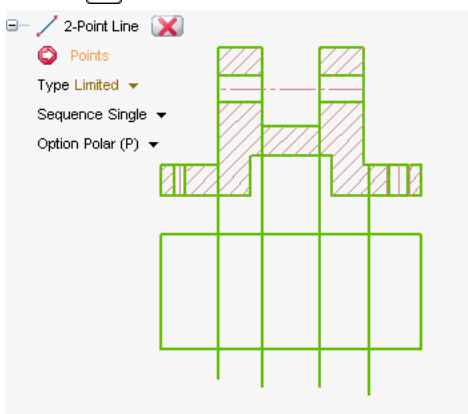


- Click OK.

### 3. Step 3 Outlining the Top View

Next, we'll draw some vertical lines to project our view. But first, turn on the **Hidden Dynamic Grid** to snap to the Vertical-90 direction.

- Start the **Two-point Line** command.
- In the selection list, set the Type to Limited.
- Set the Sequence to Single
- Draw four vertical lines as shown below.
- Hit **[Esc]** to end the command.



Turn the **Grid Off** as we don't need it anymore.

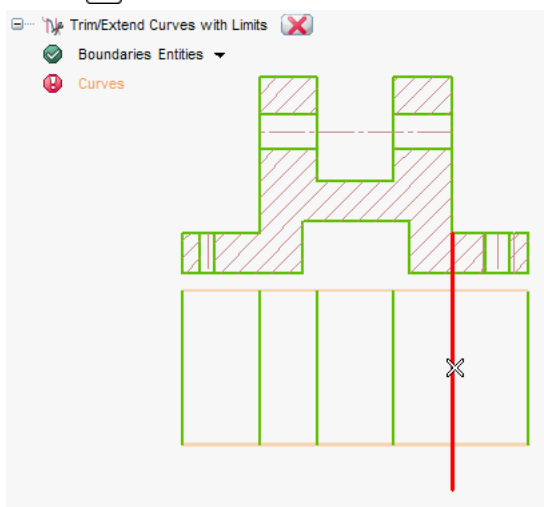
#### NOTE:

After exiting some commands the last entity drawn remain highlighted. To de-highlight it, click on the back-

ground or hit the **Esc** key.

Next we need to trim the new lines.

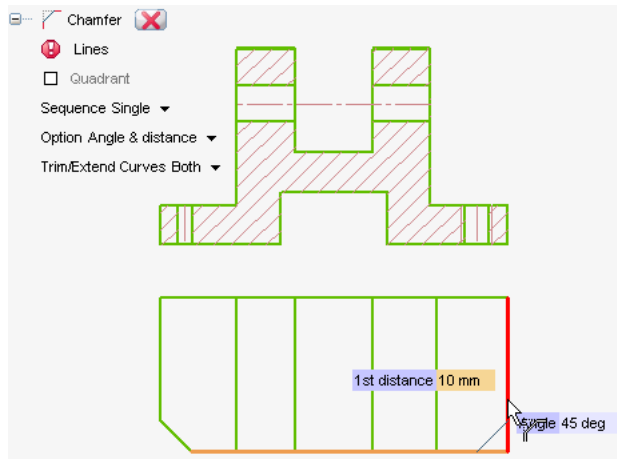
- Start the **Trim/Extend Curves with Limits** command.
- For the two boundaries, pick the two horizontal lines of the rectangle.
- Right click in the workspace, and select Continue.
- For the curves to be trimmed, pick the vertical lines, inside the rectangle.
- Hit **Esc** to end the command.



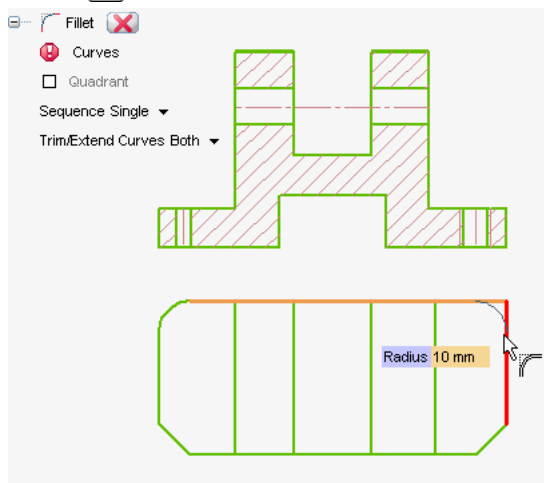
Lets add chamfered corners and fillets. How? With the 2D Chamfer and 2D Fillet commands of course!

- Start the **Insert Chamfer** command.
- Set the Sequence to Single.
- Set the Option to Angle and Distance.
- Set the 1st Distance 10mm and the Angle 45.
- First select the far left vertical line and then the bottom horizontal line.
- Repeat the operation on the right side.
- Hit **Esc** to end the command.





- Start the **Insert Fillet** command.
- Set the Sequence to Single.
- Set the Radius 10mm.
- Select the far left vertical line and the top horizontal line.
- Repeat the operation on the right side.
- Hit **[Esc]** to end the command.

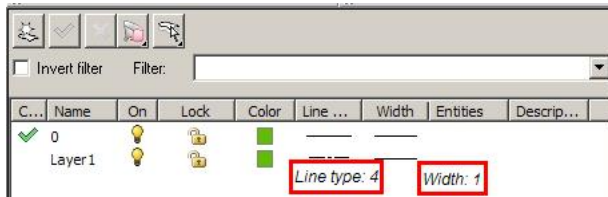


## 4. Step 4: Adding the Holes

In this step we will add mounting holes. But, before we can draw the holes, we need to create an axis line to help align the holes.

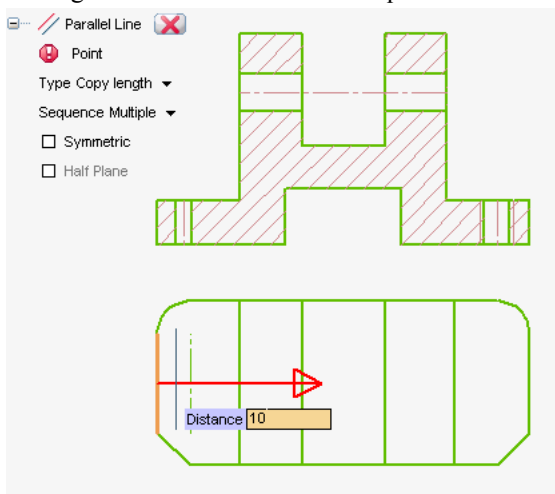
Often you would want to differentiate some specific lines from others. In this case, we want to make the axis line look different from the other lines of the part.

- Open the History Tree and Click on the Layers tab.
- Right Click in the Layers tree and create a New Layer with the attributes as shown.

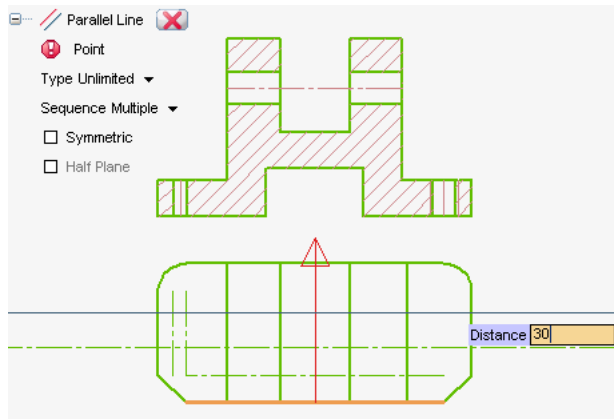


Now we'll add some vertical construction lines using the **Parallel Line** command to aid in the location of the holes.

- Start the **Parallel Line** command.
- Set the Type to Copy length.
- Set the Sequence to Multiple.
- Select the left vertical line and look at the arrow. If it does not point as shown in the image, double click on it to reverse the direction.
- Set the Distance 5mm, then hit the **Tab** key to enter another value for a second line.
- Set the Distance 10mm, then hit the **Tab** to accept the value.
- Right click and select Break Sequence.



- Select the lower line making sure that the arrow is pointing upwards.
- Set the Distance 10mm, then hit the **Tab** to accept the value.
- Then in the selection list, set Type to Unlimited.
- Set the Distance 20mm, then hit the **Tab** key to enter another value for a second line.
- Set the Distance 30mm, then hit the **Tab** to accept the value.
- Press **Esc** to end the command.



As before, use **Edit** → **Trim/Extend Curves with Limits** to trim the unlimited lines within the rectangle boundaries.

**NOTE:**

As you have probably noticed by now; when entering values into the mini dialog boxes, hitting the **Tab** key not only accepts the value, but toggles between the mini dialog boxes as well.

By using Layers, we can manage our drawing and keep track of the entities that we draw. Let's enter names for our Layers.

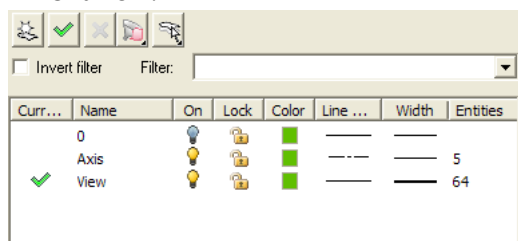
Open the Layers tab from the History tree.

We have 64 entities on layer 0 and the 5 axes on layer 1. Fill the description field with corresponding names.

**Default Layer**

It is not possible to Delete or Rename the Default layer (i.e., Layer 0). So create a dummy layer and number it as Layer 0 and then rename the other 2 layers.

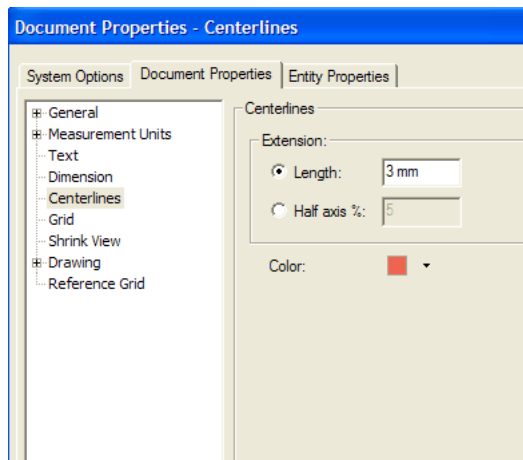
- Enter "view " for layer 0.
- Enter "axis " for layer 1.
- Click OK.



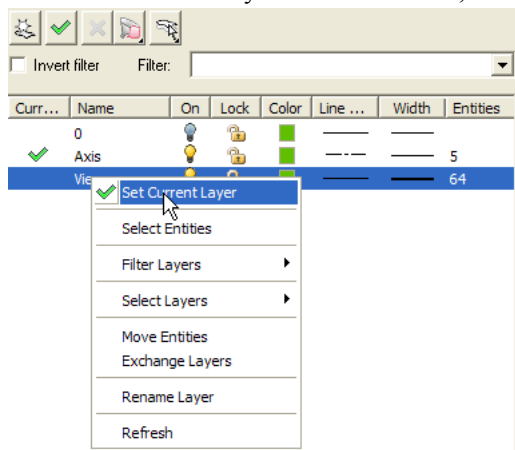
One last step before drawing the hole. Let's make sure the centerlines aren't too long.

- Select, **Tools** → **Options/Properties**
- Go to Document properties tab and click on Centerlines

- In the Centerlines box, change the Extension: Length to 3 and the Color: to Red.
- Click OK.



- Set Current the Layer "View" if it isn't, as shown in the image below.

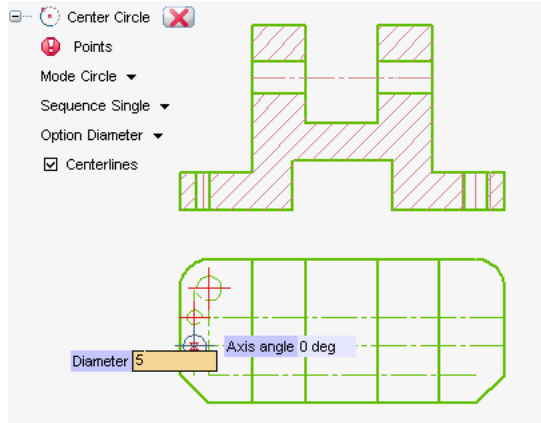


We're ready to add the first hole.

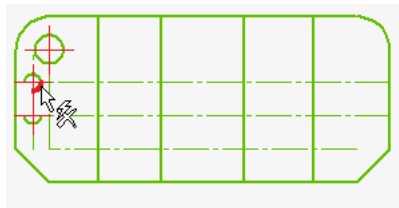
- Start the **Center Circle** command.
- Set the Mode to Circle.
- Set the Sequence to Single.
- Set the Option to Diameter.
- Check the  Centerlines option.
- Place the 1st circle on the axis at the top end point of the right vertical axis line. Set the Diameter 8.5mm.
- Place the other 2 circles at the intersection points, between the 2 unlimited horizontal axes lines and the left vertical axes line. Set the Diameter 5mm. You may have to use the **Intersection Point Snap** to place the circles.
- Hit  to end the command.

**NOTE:**

While creating the circles, after picking the center point for the circle drag your mouse away from the center point to expose the mini dialog box.

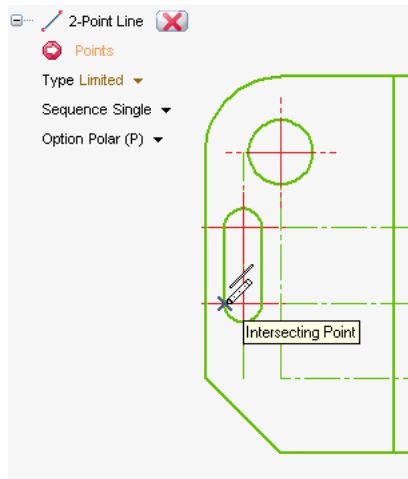


- Now with the **Smart Delete**, delete the 4 inner quarters of the two diameter 5 circles as shown.
- Hit **[Esc]** to end the command.



Now we need to draw a couple of lines between the end points of the two arcs to create the slotted hole.

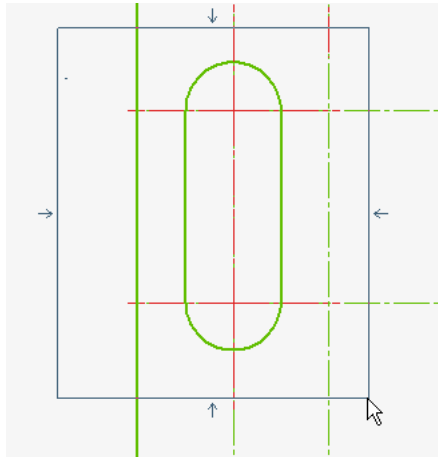
- Start the **Two-point Line** command.
- Set the Type to Limited.
- Set the Sequence to Single.
- Draw a line between the end points of the arcs.
- Hit **[Esc]** to end the command.



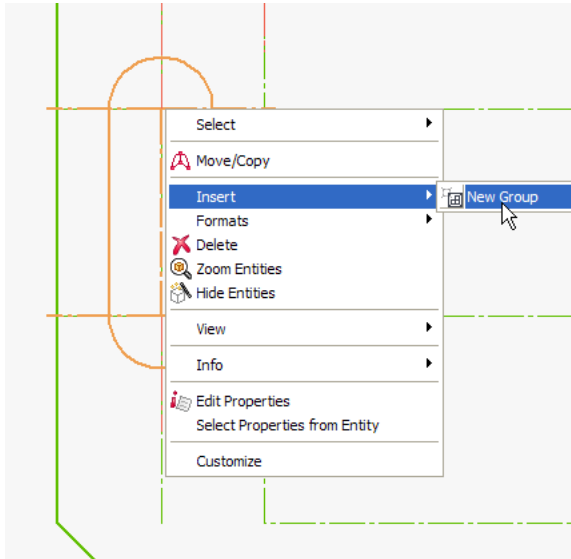
## 5. Step 5: Adding and Modifying Groups



In this step, we'll turn the holes and centerlines into 2 single entities by making them a 'Group'. We will then use that Group to create the second set of holes.

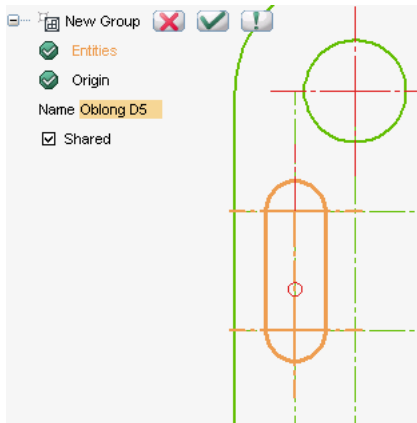
- Using a **Select Window** (drag your mouse left to right), select the oblong hole and centerlines. Make sure you encompass the arcs, lines, and center lines only.




- Select **Insert** >> **Group** >> **New**. Or, you can use the context menu by right clicking when the entities are highlighted, and then selecting **Insert >> Group** as shown.

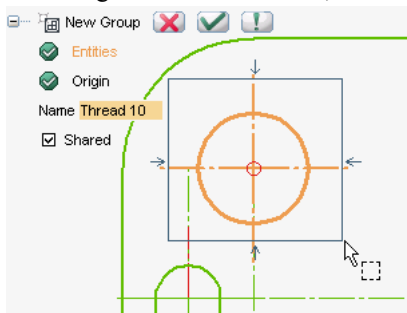


- Reset the origin by right clicking on the  Origin in the selection list, then click the center of the circle for the origin point.
- Enter OblongD5 for the Group name.
- Check the  Shared option.
- Hit  Apply to create the Group.

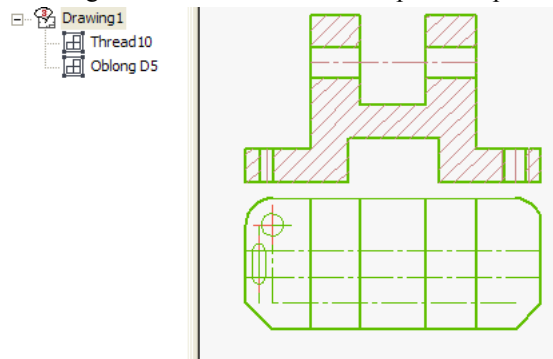


Now we need to create another Group for the circle.

- Since we only hit the  Apply icon the Group command is still active.
- Using the **Select Window**, select the circle and the center lines.



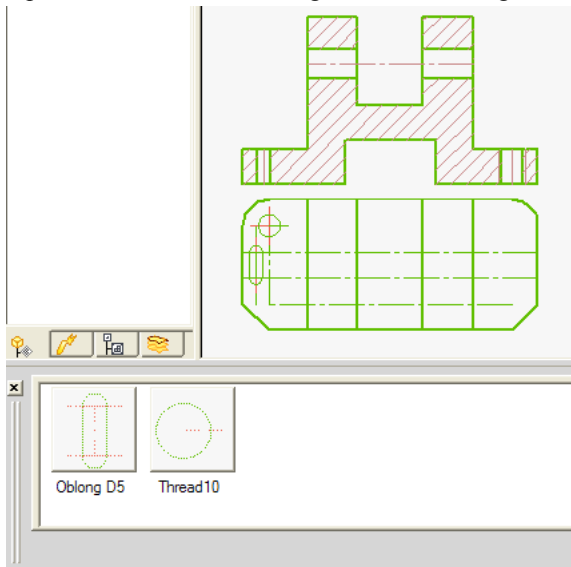
- The origin will still be located at the center of the circle.
- Enter Thread10 for the Group name.
- Leave the  Shared option checked.
- Hit  OK.
- Drag the left border of the workspace to open the History Tree, you'll see the 2 new Groups we created.



Now insert these Groups into the Group Library. A Group Library is a collection of many groups used frequently. It is simply a folder where each group is defined as a drawing file (.e2). The file name and group name remain the same.

- Click on the icon **Group Library**.
- To add the two Groups to the library, simply drag and drop them from the drawing to the library.

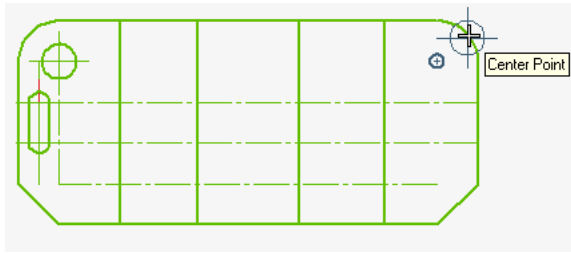
See Introduction to Groups within thinkdesign's Help section to learn a few more advanced concepts on this topic. See also; Web Training task 2D Drawing - Advanced.



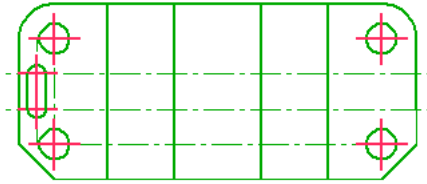
Now we'll insert the Group from the library to create 3 more holes.

- Double click the icon of the Group Thread10 to insert or right click on the icon and select Insert.
- Snap to the center point of the right fillet to place the Thread10 Group.




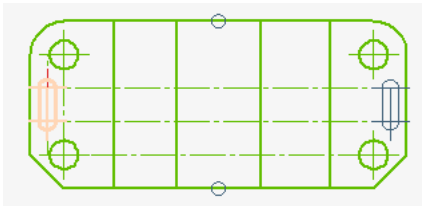


- Then place the Thread10 Group at each end point of the lower horizontal axis line.
- Hit **[Esc]** to end the command.



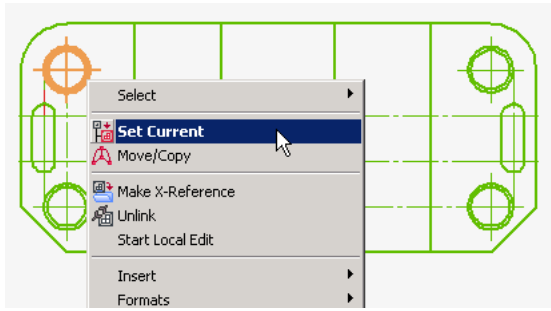
It's possible to copy a Local Group without using the library. We can create a mirror object of the OblongD5 Group on the right portion of the drawing. Mirroring those entities as a group is easier than doing them individually.

- Start the **Mirror Entities** command.
- Select the OblongD5 Group.  
Set the  Symmetry Axes: to 2 Points.
- In the selection list check the  Copy option.
- Pick the mid points of the upper and lower horizontal lines.
- Click  OK.

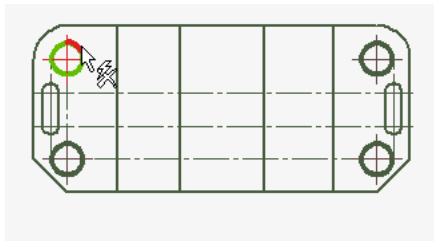


Now we'll edit the Thread10 shared Group to give the correct representation. One advantage of a shared Group is that you edit only one and the change is propagated to all the copies of the group.

- Right click on any one of the Thread10 Groups and choose **Set Current Group**
- Start the **Center Circle** command.
- Select the center of the circle in the Group.
- Uncheck the Centerlines option if it's not already done.
- Set the Diameter10.
- Hit **[Esc]** to end the command.



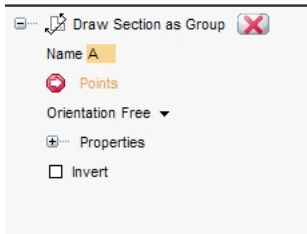
- With **Smart Delete** command, delete a quarter of external circle.
- Hit **[Esc]** to end the command.
- Double click on the workspace to close the group. Or, right click and select Reset Current Group.



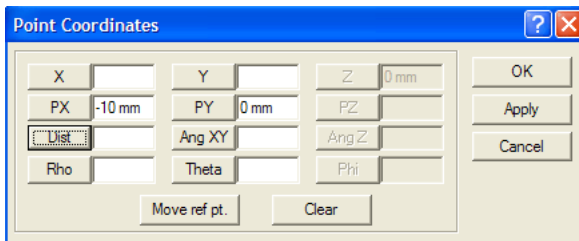
Observe that this change is reflected on all the other 3 holes.

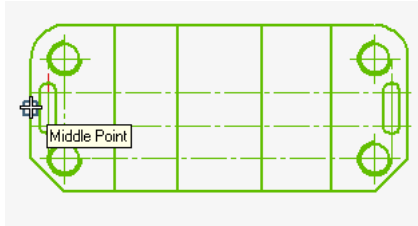
Next, we'll create a section line to represent the side-view cross-section.

- To hide the construction axes, open the layers tab and Click on the Light bulb to Switch Off the Layer Axis.
- Go to **Insert** → **Cutting Plane Lines** → **Draw as Group**.

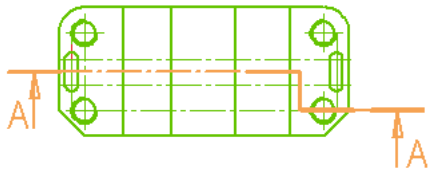


- To enter the first point of the section line, we'll use the **Enable Point Coordinates** snap.
- Enter  $PX = -10$ ;  $PY = 0$





- Click the Move ref pt. button.
- Using the **Mid Point Snap**, select the mid point of the left vertical line.
- Click OK to close the Point Coordinates window.
- Drag the Cutting Plane line to the right, pick a point to the right of the 5th vertical line and then pick the center of circle.
- Again pick a point just outside the rectangle. Now right click and select End of Input.

**NOTE:**

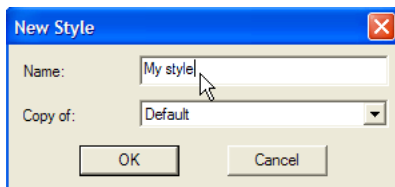
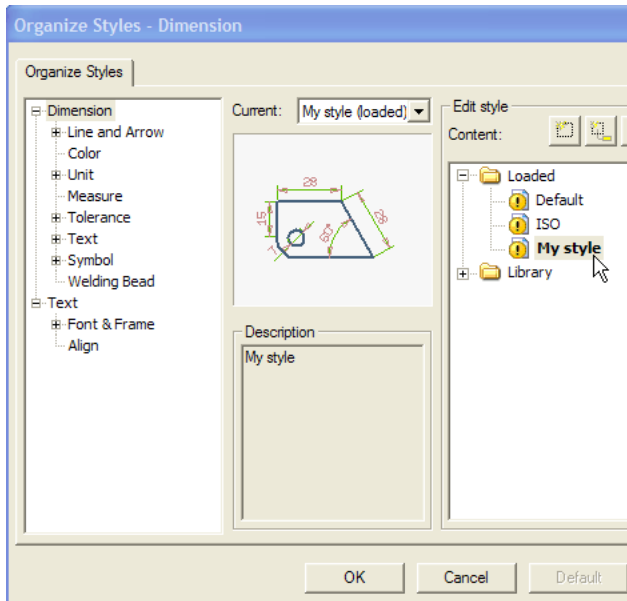
By entering -10 in the PX field, the start point of the section line starts 10 millimeters to the left of the left vertical line..

## 6. Step 6: Dimensioning (Smart and Ordinate )

Now let's dimension the drawing. ThinkDesign allows you to insert dimensions, move them, change their properties, etc. much like any other entity. You use a single dimension command to handle all dimension types. ThinkDesign determines the dimension type from the context of the entity type that is selected.

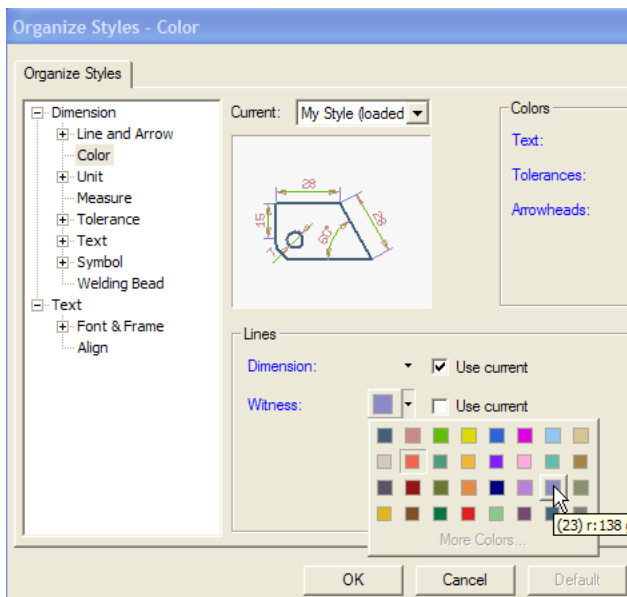
First lets create a custom style.

- Select **Format** → **Organize Styles**.
- Under Organize Styles - select Dimension and then Default
- Click on Default and then click on the New style button as shown in the image below. Name it My Style. Notice that MyStyle is also copied under the Loaded style list.



Now let's personalize the style.

- Now MyStyle is in bold to indicate that is set current.
- Select Dimension -> Color.
- Uncheck the Use current option for Witness lines
- Select color #23 for Witness lines, Text and Arrowheads.
- Click OK.



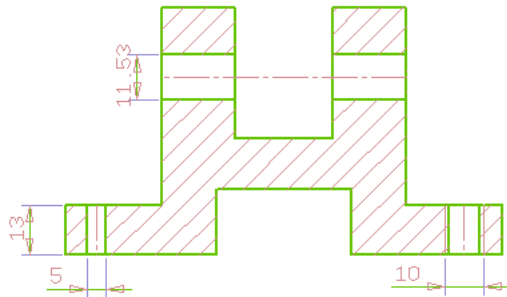
**NOTE:**

Depending on your color palate, your colors may not match the ones that are shown here. For instance, to define your own scheme for color#23 go to **Format**  $\rightarrow$  **Colors and Materials**, select color#23, hit Edit button and select one of the colors from the palette.

The style changes that we performed are saved within the drawing. Normally the style attributes are fixed.

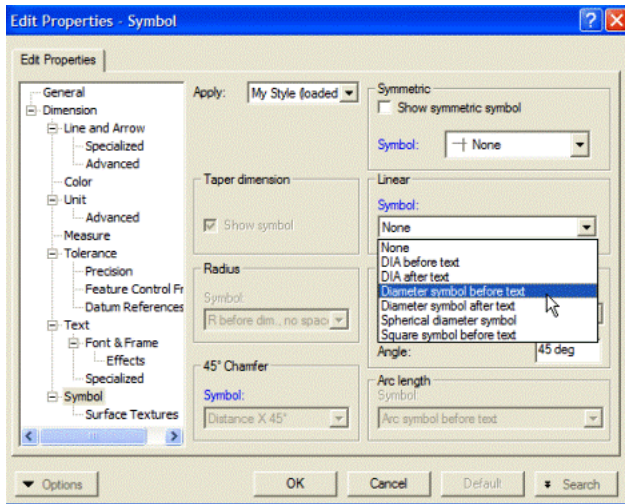
Let's add dimensions to the front section view.

- Start the **Smart Dimension** command.
- Click the lower left vertical line to dimension the thickness of the flange. You may have to zoom in a little bit to prevent snapping to the mid or intersecting point.
- Click somewhere to the left to place the dimension.
- Click the left hole's bottom horizontal line, then place the dimension.
- Repeat this for the hole on the right.
- Click the horizontal hole's left vertical line, then place the dimension.
- Hit **[Esc]** to end the command.



Hmmm, that doesn't quite look like a diameter dimension. Let's fix that as ThinkDesign allows user to set the dimension format providing options like prefix, suffix, tolerance etc.

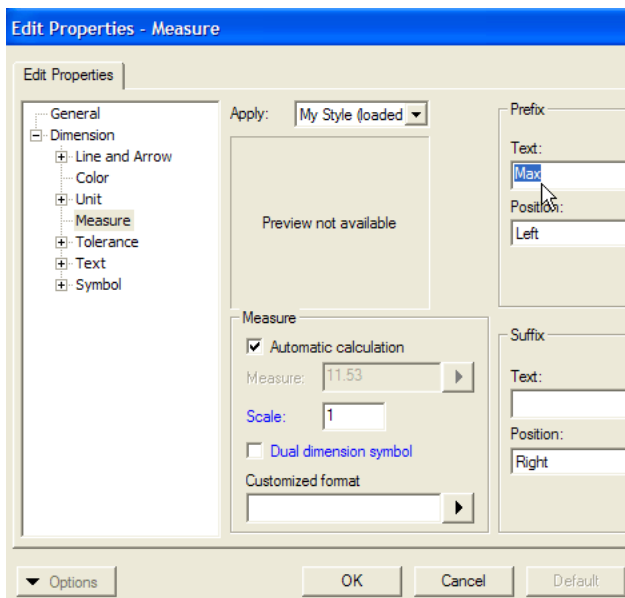
- Right click the horizontal hole's dimension and select **Properties**.
- In the Entity properties dialog - select Symbol and Scroll down the menu under Symbol and select Diameter symbol before text.



- Now select Measure and type Max for Prefix with position left.
- Click OK.
- Was the text placed OK? No - check out the settings against Text - Text Placement.

**Note**

There is another simpler way to move the dimensions text: is enough to point the mouse on the text and drag to the new position.

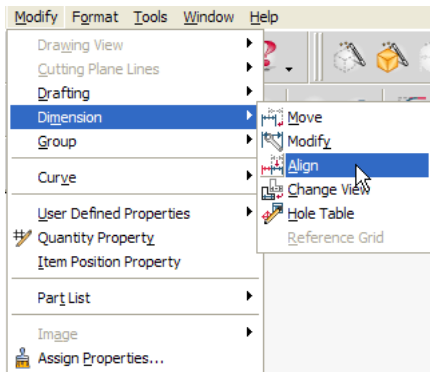


- Using the same method above, right click the right hole dimension and select **Properties**.
- Go to measure and enter M in the Prefix field.

If you want to insert special characters see also: Special characters used in technical drawings

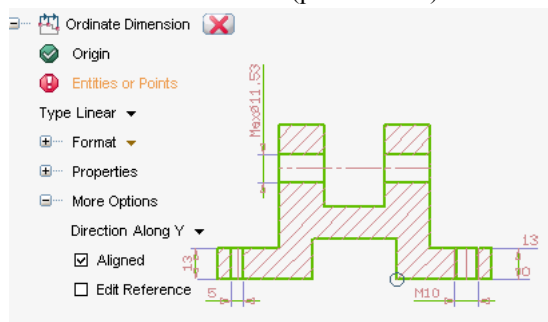
Chances are the two hole dimensions don't line up. This can be easily rectified.

- Start the **Align Dimension** command.
- Pick the left hole dimension as the reference entity.
- Then pick the right hole dimension as the one to be aligned.
- If there are more dimensions to align, you can continue picking them. But since there aren't any, simply right click the workspace, and select End of Selection.
- Hit **[Esc]** to end the command.

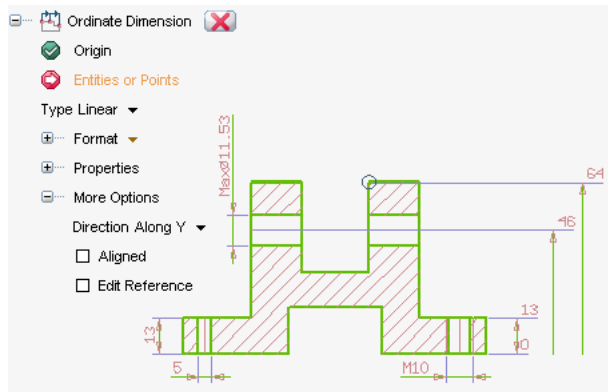


Now we're going to add **Ordinate Dimension** to the view.

- Start the **Insert**  $\rightarrow$  **Dimension**  $\rightarrow$  **Ordinate** command.
- Select type as linear
- Select the origin point for reference
- Now select the entities (point or line) as shown and you get dimensions as 13 and 0 from origin point.



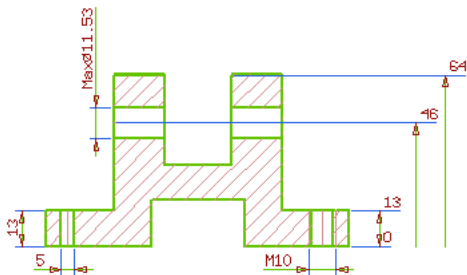
- Now click on each point where you want to put the ordinate dimensions like for values 46 and 64.
- Hit **[Esc]** to end the command.



That's it for Ordinate dimensioning!

A quick example of the use of 'Styles '

- Open **Format** → **Organize Styles** command.
- In the dialog window select Dimension and notice that MyStyle is in bold to indicate that is set current
- Select Dimension -> Color.
- Change the color for Witness lines, Text and Arrowheads.
- Click OK to close the Organize Styles window.



Notice that some of the dimension formatting has changed. This because the style are associative: changing the style settings, all the dimension referred to the style will change.

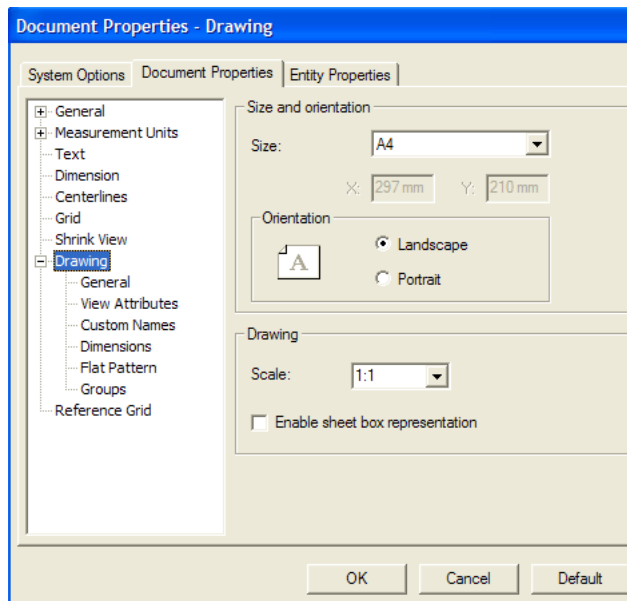
## 7. Step 7: Title Block and Printing

In this step, we will add a title block and then set up the printing properties.

First thing we need to do is to add a title block.

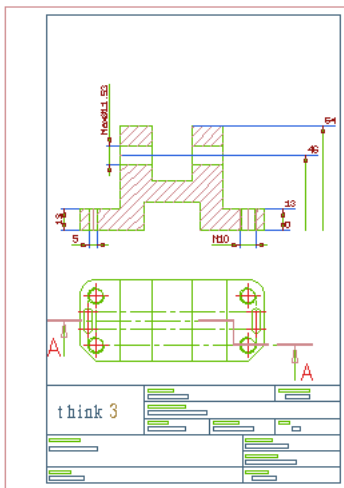
- Right click in the workspace and select **Options/Properties**.
- Go to Document Properties and select Drawing.
- Under Sheet Size and Orientation change the Size to A4 and Orientation to Portrait.
- Click OK.
- Drag the title block into place.





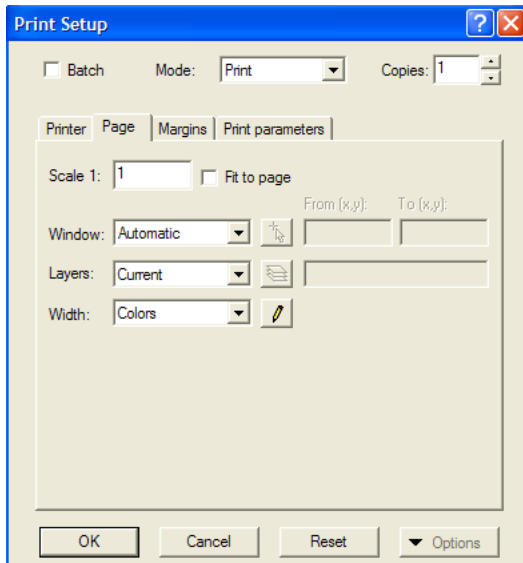
Let's look at the **Print Preview** to see how the default settings are set (see picture).

You can see that the hidden axes are showing up and the entities are all with the same width and color.

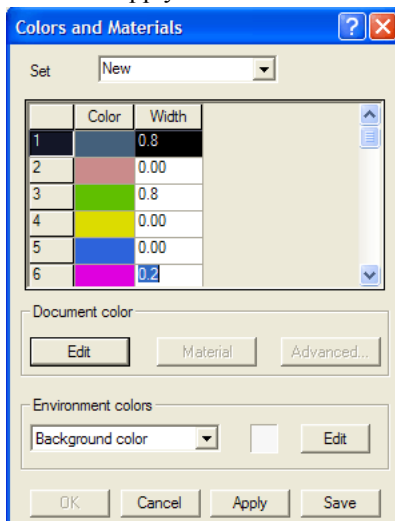


How to set a different mode for printing:

- Open the **Print Setup**.
- Select the Page tab.
- Set the Layers to Current.
- Set the Width to Colors.
- Click the button Pencil icon to open the Colors and Materials window.



- Set Color 1 to 0.8.
- Set Color 3 to 0.8.
- Set Color 6 to 0.2.
- Click Apply.



- Save this new color set by typing a name in the Set field then click Save.
- Click OK to exit the Colors and Materials dialog box.
- Click OK to exit the Print Setup dialog box.

Now open the **Print Preview** again to see the result.

