Alibre Design - Drawing Tutorial 1

Introduction

This tutorial guides you through the creation of your first Alibre Design drawing. You can quickly create drawings directly from parts and assemblies you have already modeled. Changes made to driving dimensions in a part/assembly are automatically applied to the drawing, and vice versa.

It is assumed that you have previously read and worked through *Modeling Fundamentals*, *Part Modeling Tutorial 1*, and *Assembly Modeling Tutorial 1* before starting this tutorial.

You will be using the **Enclosure** part created in *Part Modeling Tutorial 1* to create a simple drawing.

This tutorial will cover:

- Opening a new drawing workspace.
- Selecting a drawing template.
- Setting the drawing scale.
- Inserting standard views into the drawing.
- Adding dimensions to the drawing views.
- Adding notes and annotations to the drawing.
- Saving a drawing to the Repository.

Starting Alibre Design

1 From the Windows **Start** menu, click **Programs > Alibre Design**.

The **Alibre Design Launcher** window appears.

2 Click either **Work Offline** or **Connect Now** to launch Alibre Design. You can complete this tutorial working either offline or online.

The Alibre Design Home window appears.

Opening a New Drawing Workspace

1 To open a new drawing workspace, on the Home window, from the **File** menu, select **New > Drawing**, or click the **New Drawing** icon on the main toolbar.

The **New Sheet Properties** dialog box appears.

Selecting a Drawing Template

Select the template size, set the scale, and enter some title block information.

1 In the **New Sheet Properties** dialog box, select the **Standard** template option.

2 Select the **ANSI B** size template.

3 Set the **Default View Scale** as **1.0** : **4.0**.

4 Click **OK**.

The **Fill In Text** dialog box appears.

5 Select the **Drawn** item in the **Select Tag Field** list and then type your initials in the DRAWN text box.

6 Select the **DWG NO.** item in the list and then type **dwg1** in the DWG NO. text box.

7 Select the **DATE** item in the list and then type today's date in the DATE text box.

8 Click **OK**.

A new drawing workspace opens as well as the **Insert Design** dialog box.

9 From the **Insert Design** dialog box, in the Repository Explorer, select the **Tutorial** folder.

10 Select the **Enclosure** part from the item list.

11 Click **OK**.

The **Standard Views Creation** dialog box appears.

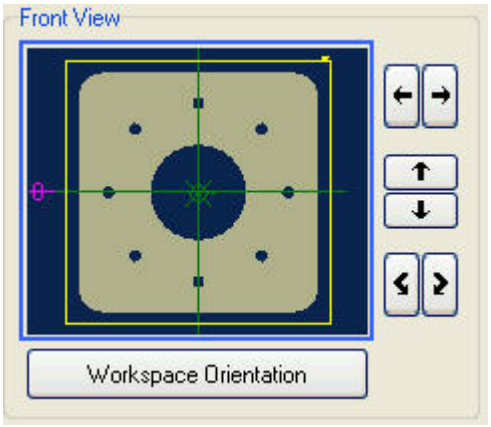
Selecting the Standard Drawing Views

Now select which standard views to insert into the drawing.

1 The first step in selecting the standard views is to select the view that you want to use as the front view. In the **Front View** area of the dialog box, click the **Workspace Orientation** button. The **Orientations** dialog box appears.

2 In the Orientations dialog box, double click **Front(XY)**, or select **Front(XY)** and click **Set.** The **Front View** preview will change in the Standard Views Creation dialog box.

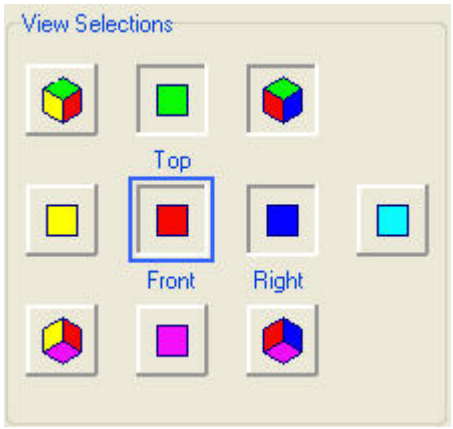
3 Click **Close** on the Orientations dialog box.



Note: Depending on how you created the Enclosure part, your Front View preview may not match the image above. If your preview does not match, click the **Workspace Orientation** button again and select the view that does match the view in the image above.

4 In the **View Selections** area, the **Front, Top,** and **Right** views are selected by default. A view is selected when the corresponding button is depressed. Note that the red face on the symbol corresponds to the front face, the green face corresponds to the top face, etc.

Also select the isometric view in the upper right.

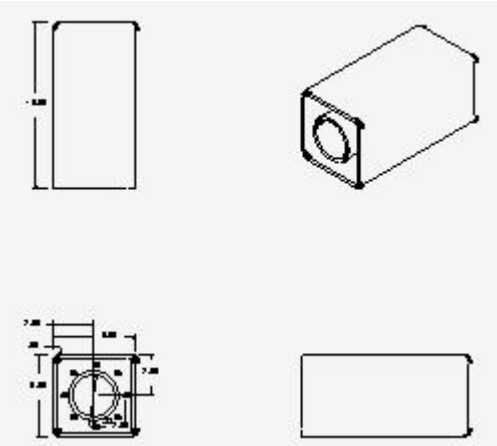


5 Click **OK** to generate the standard views and close the Standard Views Creation dialog box.

The four standard views are previewed in the workspace.

Note that the cursor is essentially tied to the front view. As you move the cursor, the views will all move together.

Also note that the Sketching and Detailing toolbars are displayed on the right side of the work area.

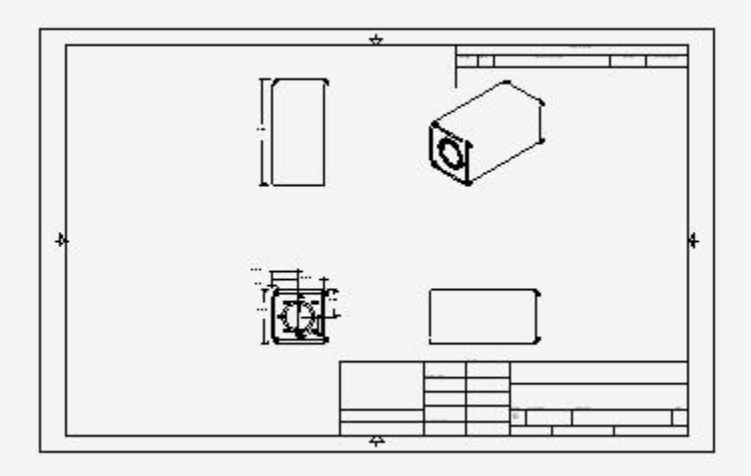


6 To position the views within the template, move the cursor and the views will move.

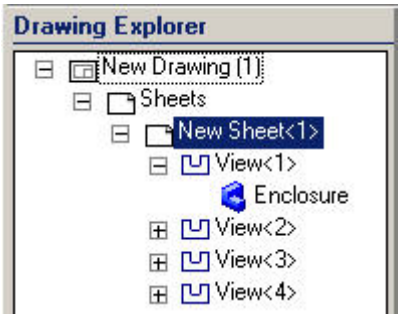
7 When you have the Front view positioned correctly, click to place it in the sheet, the other views will automatically be placed as well.

Note the driving dimensions that were used to create the base feature in the part are displayed automatically. In this example, the majority of dimensions are displayed in the front view. The top view displays the extrusion length dimension.

Any dimension can be deleted and additional dimensions can be inserted as necessary.



The **Drawing Explorer** on the left side of the work area lists the sheets associated with the drawing, as well as the views associated with each sheet. In this example, one sheet and four views are listed. The corresponding design will be listed under a view.



Move the cursor over a view in the Drawing Explorer, and A red boundary will be displayed around the associated view in the work area.

8 Right-click **New Sheet<1>** in the Drawing Explorer and select **Rename** from the pop up menu. Type **dwg1** and press **Enter**.

9 Rename each view in the Drawing Explorer to match its orientation. Right-click on **View<1>** and select **Rename** from the pop up menu. Type **Front View** and press **Enter**.

10 Rename **View<2>** to **Top View**, **View<3>** to **Side View**, and **View<4>** to **Iso View**.

Setting the Drawing Workspace Properties

Check to make sure the units are set appropriately.

1 From the **File** menu, select **Properties**.

The Properties dialog box appears. Click the **Units** tab if it is not already selected.

2 Set **Units** to **Inches**, **Format** to **Decimals**, and **Precision** to **2**.

3 Select the **Show Units for Dimensions** option.

4 All other default options and settings are acceptable for this tutorial. Click **Apply** and then **Close**.

The drawing workspace properties are updated.

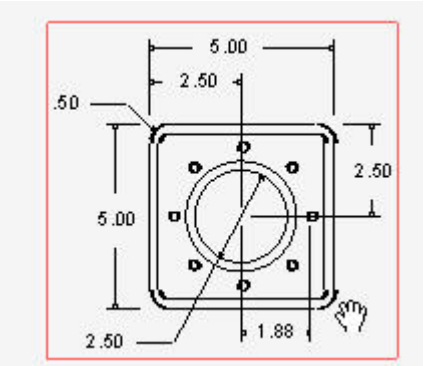
Moving Views on the Sheet

You can move the views after initial placement. In this example, the Top and Side views are initially dependent on the Front View's position. The Isometric view's position is independent of the rest.

1 First, ensure that the **Views** selection filter is on. From the **Tools** menu, select **Selection Filters > Views**.

2 Click the **Select** tool from the View toolbar if it is not already selected.

3 Move the cursor over the **Front** view. A view boundary is displayed around the view extents and the cursor changes.



4 To reposition the views, click and hold the mouse button, and move the cursor. The Front, Top, and Side views all move together, the Isometric view remains stationary.

5 Release the mouse button to place the views.

6 Move the Isometric view as well. Notice that the other views do not move when the Isometric view is moved.

Note: You can break an individual view's dependence on other views by right-clicking on a view and selecting **Break Alignment** from the pop up menu.

Moving Dimensions in a View

After placing the views in the sheet, some dimensions in the Front view may need to be moved to add clarity to the view.

1 Select the **Zoom to Window** tool on the View toolbar, and drag a selection window that encloses the entire Front view. Click and hold to start one corner of the window, move the cursor, and click again to finish the window. The Front View is displayed in the work area.

2 First, ensure that the **Dimensions** selection filter is on. From the **Tools** menu, select **Selection Filters > Dimensions** if it is not already selected. A check mark indicates that filter is currently on.

3 To move a dimension, first move the cursor over it. The dimension lines turn red and the cursor changes to show the Dimension symbol.

4 Click and hold, then drag the dimension to a new location.

5 Release the mouse button to place the dimension.

6 Continue to reposition the dimensions on the Front view until an acceptable level of clarity has been achieved.

Adding Dimensions to the Front View

Next, place additional dimensions in the Front view and then edit one of the dimensions.

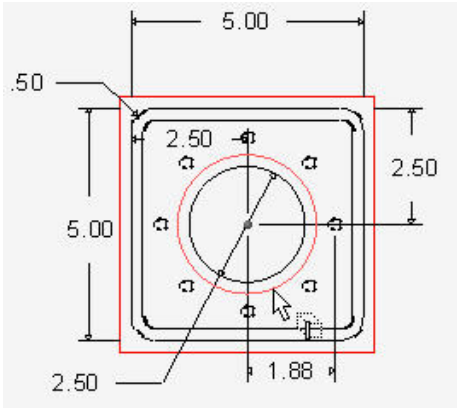
Placing diameter dimensions on the Front View.

1 Zoom in once again on the Front view.

2 First, ensure that the **Segments** selection filter is on. From the **Tools** menu, select **Selection Filters > Segments** if it is not already selected. A check mark indicates that filter is currently on.

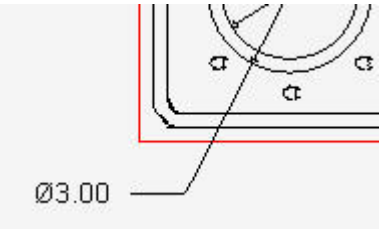
3 Select the **Dimension** tool on the Sketching toolbar.

4 Move the cursor over the large circle in the Front view. A red boundary encompasses the view and the circle turns red.

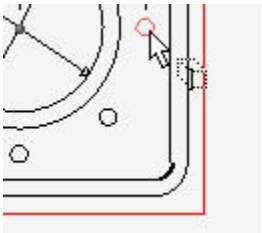


5 Click once, a **3.0”** diameter dimension appears. Move the cursor to drag the dimension away from the circle.

6 Click to place the dimension.

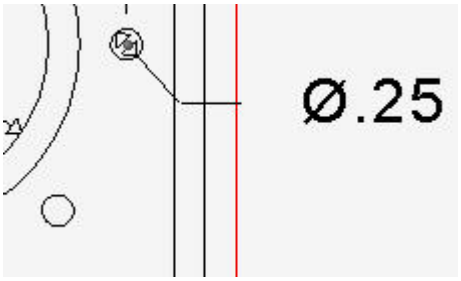


7 With the Dimension tool still selected, move the cursor over the small circle on the right side of the Front view. The circle turns red.



8 Click once, a **.25”** diameter dimension appears. Move the cursor to drag the dimension away from the circle.

9 Click to place the dimension.

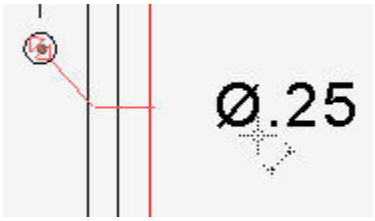


10 Select the **Zoom to Fit** tool from the View toolbar to view the entire Front view. In sketch mode, the Zoom to Fit operation will only display the active view in the work area.

Editing dimensions on the Front View

1 Click the **Select** tool from the View toolbar if it is not already selected.

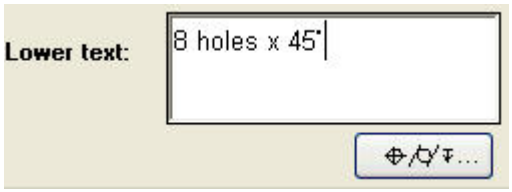
2 Move the cursor over the **.25"** diameter dimension just placed. The dimension line turns red.



3 Right-click the dimension, and select **Properties** from the pop up menu. The **Dimension Properties** dialog box appears.

4 Select the **Text** tab.

5 In the **Lower text** area, type **8 holes x 45°**, click the **Symbols** button beneath the text box to add the ° symbol.



6 Click **OK**. The additional information appears beneath the diameter dimension.

7 Select the **Zoom to Fit** tool from the View toolbar to view the entire drawing.

Saving the Drawing

Save the work you have completed up to this point.

1. Click **Save** on the Standard toolbar, or from the **File** menu, select **Save**.

   The **Save** dialog box appears.

2. In the Repository Explorer, expand your local repository by clicking the plus sign next to it.

3. Select the **Tutorial** folder.

4. In the **Name** field, type **Enclosure**.

5. Select **Alibre** as the **Save as type**.

6. Select the **Make new versions for all** option if it is not already selected.

7. Click **OK**. The Enclosure drawing is saved to your local repository.

Adding Dimensions to the Top and Side Views

Continue to add detail to the drawing by adding linear dimensions to the Top and Side views.

Placing linear dimensions on the views.

1. In the Drawing Explorer, right click **Top View** and select the **Show Hidden Lines** option if it is not already checked. Do the same for **Side View**.

2. Select the **Zoom to Window** tool on the View toolbar, and drag a selection window that encloses the entire Top view.

3. Select the **Dimension** tool on the Sketching toolbar.

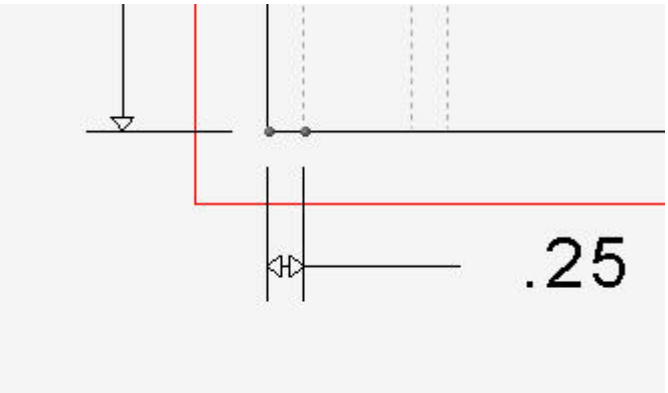
4. Move the cursor over the left most vertical line that represents the outer face of the Enclosure part. The line turns red.

5. Select the vertical line. A vertical dimension appears, do not place this dimension.

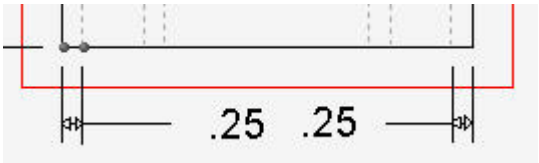
6. Move the cursor over the dashed line that represents the inner face of the Enclosure part. The line turns red.

7. Select the dashed line. A **.25”** horizontal dimension appears.

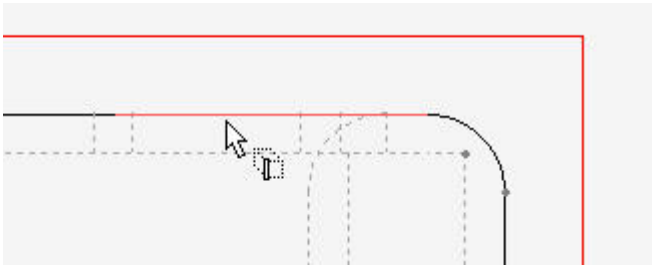
8. Move the cursor beneath the view to drag the dimension. Click to place the dimension.



9 Repeat steps **4-8** to dimension the wall thickness on the right side of the view.

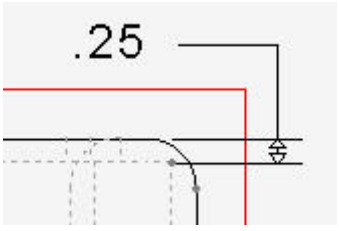


10 With the Dimension tool still selected, select the top horizontal line that represents the outer end face of the Enclosure part. A horizontal dimension appears but do not place it.



11 Select the horizontal dashed line that represents the interior end face of the Enclosure part. A **.25"** vertical dimension appears.

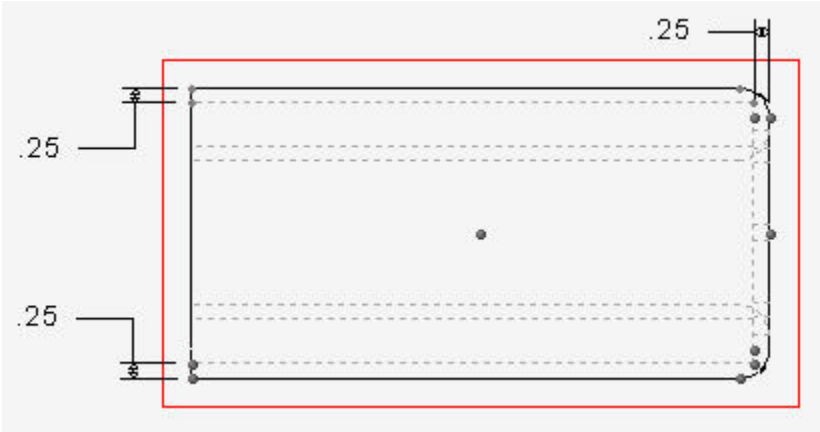
12 Move the cursor to the right of the view to drag the dimension. Click to place the dimension.



13 Select the **Pan** tool from the View toolbar. Move the sheet so the **Side view** is visible in the work area.

14 Select the **Dimension** tool on the Sketching toolbar.

15 On the Side view, place three dimensions to specify the wall thickness in a similar fashion as the Top view.



16 Select the **Zoom to Fit** tool from the View toolbar to view the entire drawing.

Adding Annotations to the Drawing

Finally, add annotations to the drawing. First place a Surface Finish annotation on the Iso view, and then add Annotation Notes to the drawing title block.

Adding a Surface Finish Annotation to the Isometric View

1 Right-click **Iso View** in the Drawing Explorer, and select the **Show Tangent Edges** option if it is not already selected.

2 Select the **Zoom to Window** tool on the View toolbar, and drag a selection window that encloses the entire Iso view.

3 Select the **Surface Finish** tool from the Detailing toolbar, or from the **Insert** menu, select **Annotation > Surface Finish**.

The **Surface Finish** dialog box appears.

4 Select **Basic** as the **Symbol** type, and select **Parallel** as the **Lay Direction**.

5 In the **Roughness** area, enter **16** in the **Maximum** field.

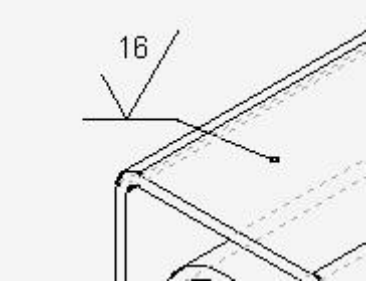
6 In the **Leader** area, select from the **Arrow** list. Select the **Show** and **Bent** options, and change the **Position** to **Right**.

7 Click in the **Face or Point** area, the **Face or Point** labels becomes bold.

8 Move the cursor over the top face in the Iso view.

9 Click once, the Surface Finish symbol and leader appear. Drag the symbol to the left of the view and click again to place it.

10 Click **OK** in the dialog box.

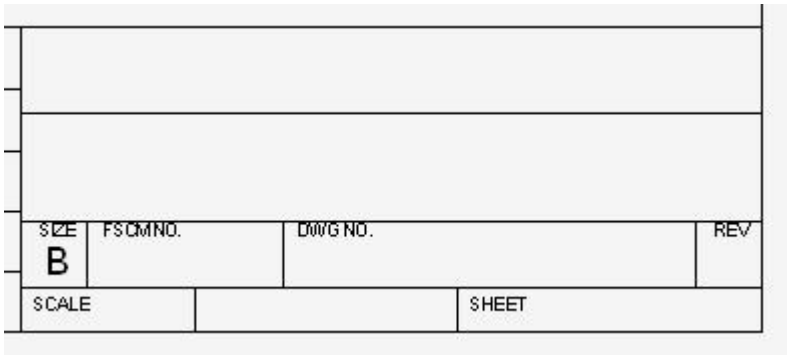


Adding Information to the Title Block

1 Right-click **dwg1** in the Drawing Explorer and select **Sketch Mode** from the pop up menu.

A red boundary appears around the entire sheet.

2 Select the **Zoom to Window** tool on the View toolbar, and drag a selection window that encloses the right side of the title block.



3 Select the **Note** tool from the Detailing toolbar, or from the **Insert** menu, select **Annotation > Note**. The **Note** dialog box appears.

4 Click **Font**. The **Font** dialog box appears.

5 Set the **Font** size at **24**.

6 Select **Arial** as the font type.

7 Click **OK**.

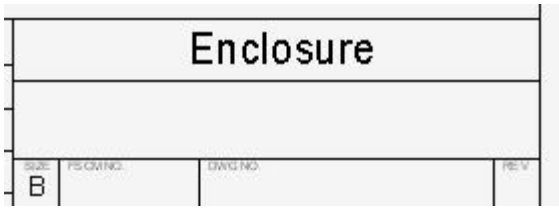
8 Type **Enslosure** in the text area.

9 Click in the **Location** box. The **Location** label becomes bold.

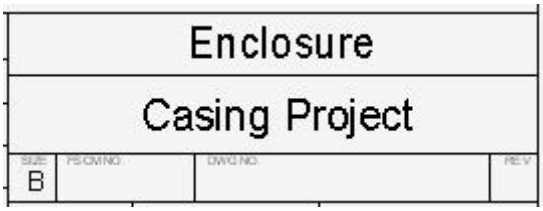
10 Place the note by clicking in the first box in the title block. The note appears.

You can continue to click and reposition the note until it is centered appropriately.

11 Click **OK** in the Text dialog box.



12 Repeat steps **2-7** to place another annotation below the **Enclosure** annotation. Enter **Casing Project** as the text.



13 Select the **Zoom to Fit** tool from the View toolbar to view the entire drawing.

14 Click **Save** on the Standard toolbar, or from the **File** menu, select **Save**. Click **Save** in the Save dialog box.

15 If you have a printer that can print a B size sheet, print the drawing. Select **Print** from the Main toolbar, or from the **File** menu, select **Print**.

16 Close the drawing workspace when finished.

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