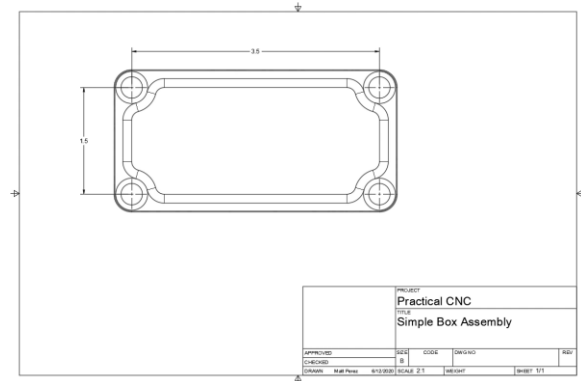


Lesson: Create a Simple Detailed Drawing

In this lesson, you'll create a detailed drawing and add a few details.

Learning Objectives

- Use the drawing workspace to create a simple drawing.
- Identify areas of the drawing workspace.
- Use Dimensions.

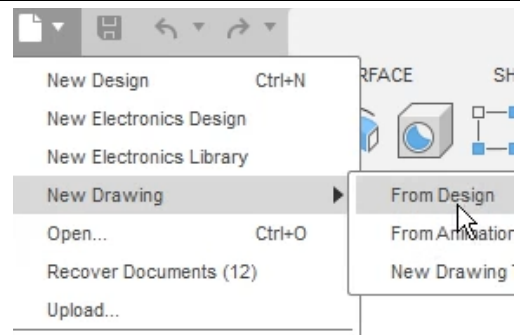


The completed exercise

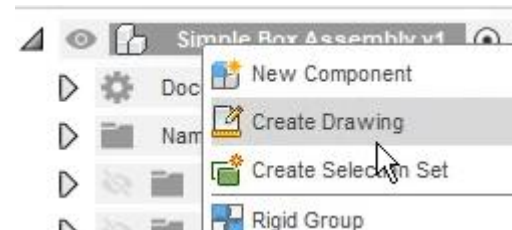
1. Upload and open the supplied *Simple Box Assembly.f3d* file. This file has a base component, a lid component, and four fasteners.



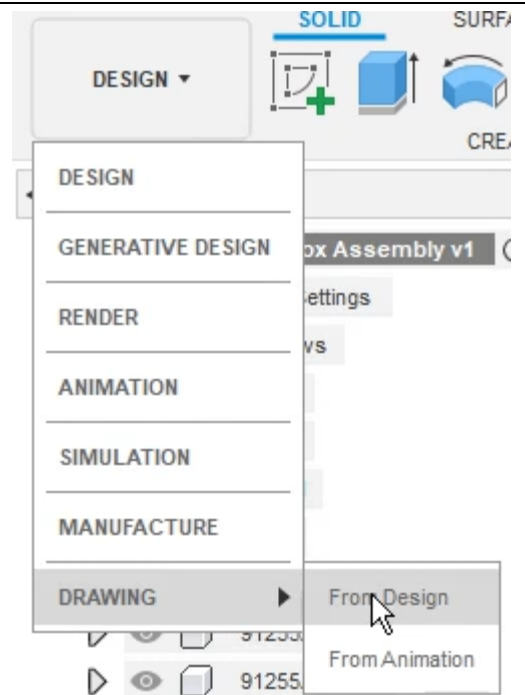
2. Fusion 360 can create detailed drawings using several methods; you could click File > New Drawing > from Design.



3. After right-clicking a component, you could select Create Drawing from the menu.



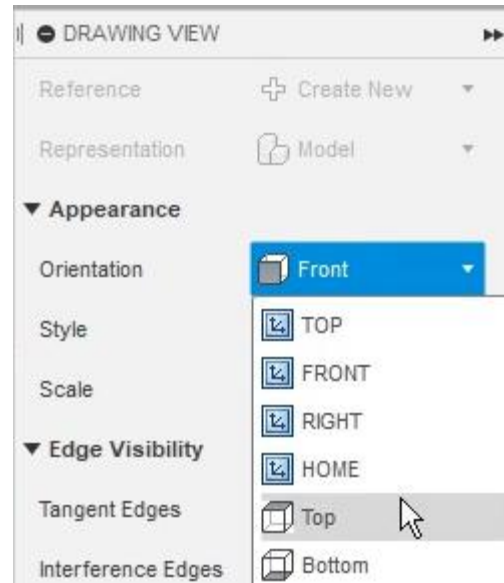
4. A third way to create drawings is by using the drawing workspace. Click Change Workspace and select Drawing> from Design. Use one of these methods to create a drawing based on this current design.



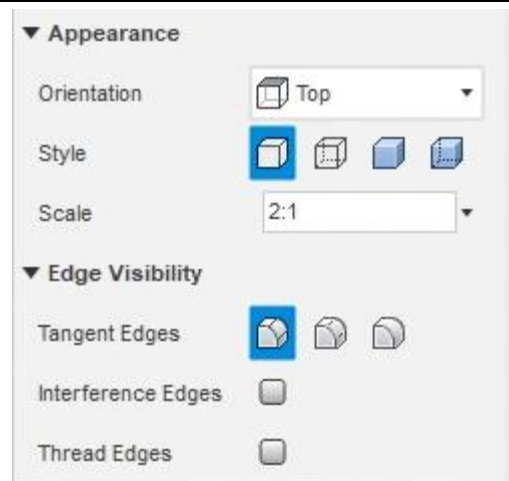
5. In the Create Drawing dialog, deselect the Full Assembly option. Select the lid as the dialog's Components selection. Click OK in the dialog to generate the new drawing.



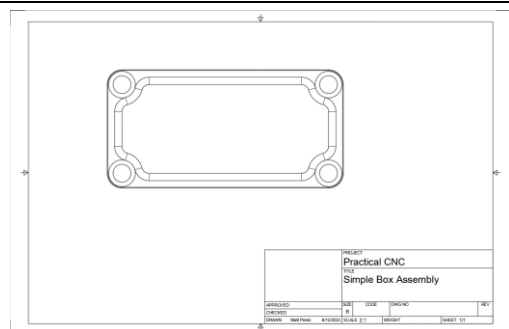
6. Before placing the view onto the sheet, choose the Top option from the dialog's Orientation menu.



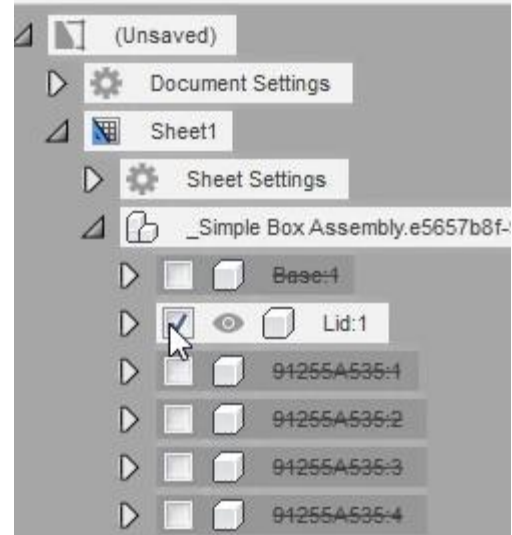
7. Activate the Visible Edges option inside the dialog's Style section, then choose the 2:1 option from the Scale menu. Choose the Full Length option in the Tangent Edges section, then deactivate the Thread Edges option.



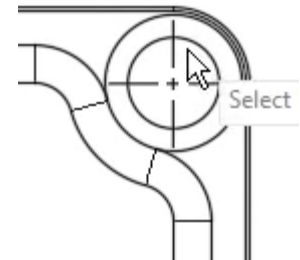
8. Click in the sheet to place the top view of the model, then click OK in the dialog.



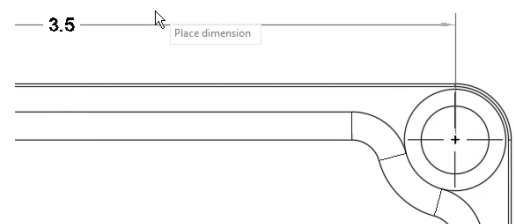
9. The Browser is located on the left side of the screen. The Browser has important information such as the document settings and sheet settings. Also, the individual components can be hidden or shown, suppressed or unsuppressed.



10. Additional details can be added to the drawing view. Click Geometry> Center Mark, then select the hole perimeters to place a center mark over the geometry.



11. Dimensions can also be added to the sheet. Click Dimensions> Dimension. Add a dimension between two of the lid's horizontal holes by clicking on each of the two holes' center points.



12. Add another dimension between two of the vertical holes. Save the file and continue to the next module.

