



Inventor

A Matter of Perspective

Design View and Positional Representation

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Class Overview: Come learn how to take advantage of design views and Positional representations available to you in Inventor. See how Design Views and Positional representations can simplify creation of views in both your assembly, and in your drawing.

You will see how design views can be utilized across your assemblies and drawings to create documentation much more quickly, and how positional representations can better illustrate the assembly's range of motion, creating more detailed documentation.

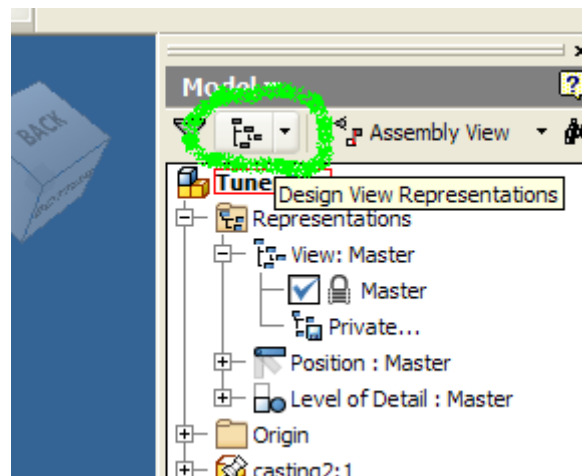
Topics:

- 1. Creating design views.**
- 2. Creating Positional representations.**
- 3. Placing design views and Positional representations in your drawings.**
- 4. Using I-part filters.**

1. Creating design views (view representations):

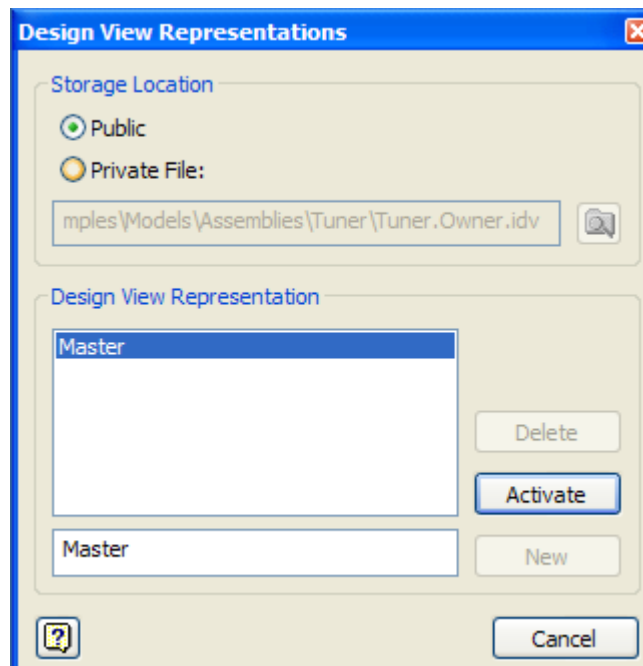
Use the Representations node at the top of the browser to preserve an assembly display configuration so that you can recall it by name when you next work on the assembly. You can define components as visible or not visible, enabled or not enabled. You can also change component colors, zoom magnification, viewing angle, and browser expansion state. For access see figure 1.1.

Figure 1.1



View representations are saved in the current assembly and are available to any designer who has access to the assembly. Private View representations support legacy behavior and are stored in an external file. They are accessible only by browsing to its location. Once you click on the icon inside the GREEN the “design view representation” dialog box appears. See figure 1.2.

Figure 1.2



On the View Representations dialog box, enter a name for the view in the edit box, and then click New. Go ahead and change view perspective, colors, visibility and then right click on the view representation and “LOCK”. This will prevent undesired design view changes.

Optionally you can use the manual method (my way):

1. Define the assembly viewing characteristics you want to preserve in the View representation.
2. At the top of the browser, click the Representations node to expand, right-click the View node, and then click New.

A new View Rep node is added to the browser, nested under the View node. It becomes the active View representation, as designated by a check mark. The default name is appended to the View node.

3. If appropriate, click the View name to activate the edit box, and then enter a new name.

Always give your View representation a descriptive name to help you remember its content.

4. Do one or more of the following to set additional attributes. Right-click the View representation, and then:
 - o Select Lock to prevent changes to the View representation display characteristics. Components added after a view is locked are not visible while the view is active.
 - o Select Copy to duplicate the current View representation and make it active. The name is appended by an incremental number.
 - o Select Copy to Level of Detail to create a Level of Detail representation that keeps the visible components in memory, but unloads the invisible components from memory by suppressing them.
 - o Select All Visible to turn visibility on for all components.
 - o Select Nothing Visible to turn visibility off for all components.
 - o Select Remove Color Overrides to return all components to their default colors.

The MASTER design view is created by default by Inventor see figure 1.3 for the master and figure 1.4 for the new one I created “George 1”.

Figure 1.3

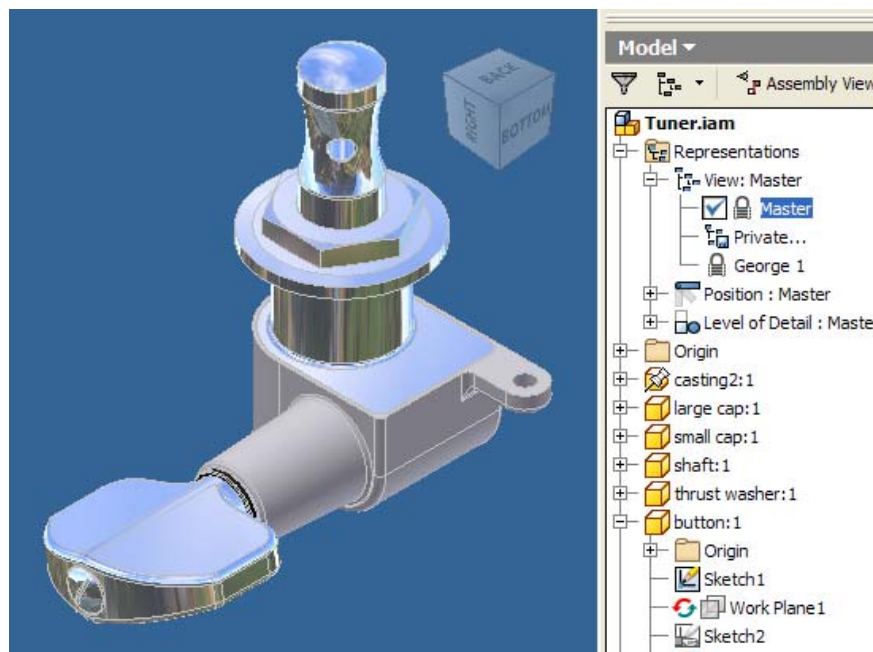
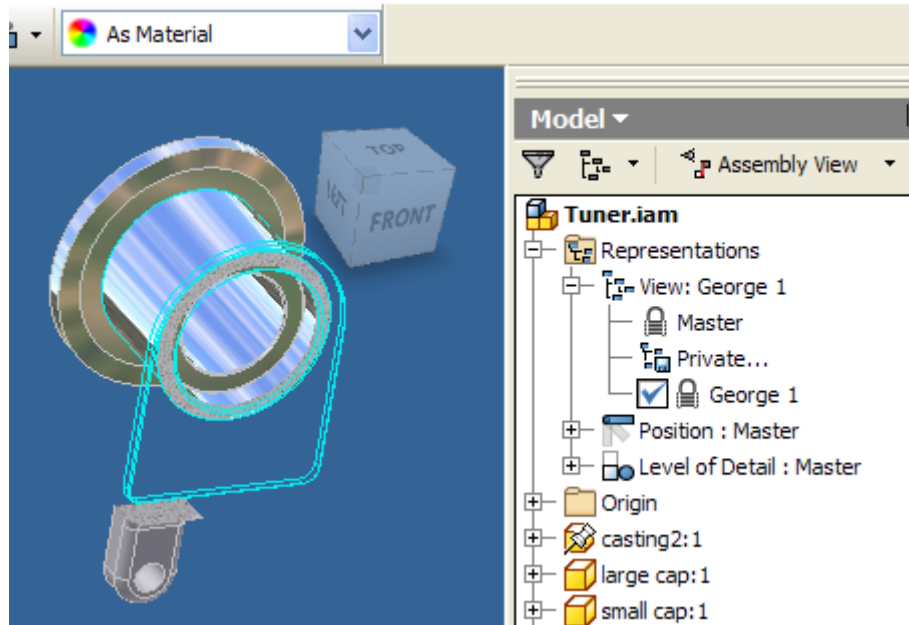


Figure 1.4



2. Creating Positional representations:

Use Positional representations to capture kinematic "snapshots" of assemblies for motion studies and other purposes where you require a specific view. Positional representations are often used in conjunction with flexible assemblies to show component occurrences in various positions within an assembly.

An assembly file has a master Positional representation that represents the default state of the assembly. When you create a new Positional representation, the master is copied.

1. In the browser, click to expand the Representations folder.
2. Right-click Position and then select New. The new Positional representation is activated and nested below the master, using the default name PositionalRep1. See figure 2.1.
3. Additional representation names are incremented unless you change the name.
4. If appropriate, click the Positional representation in the browser, and then enter a new name.

5. Establish overrides to constraints and components, with the desired positional representation active, double click the desired constraint to suppress or change value. See figure 2.2.
6. Click the Save tool to save changes. The Positional representation is saved in the assembly file.

Figure 2.1

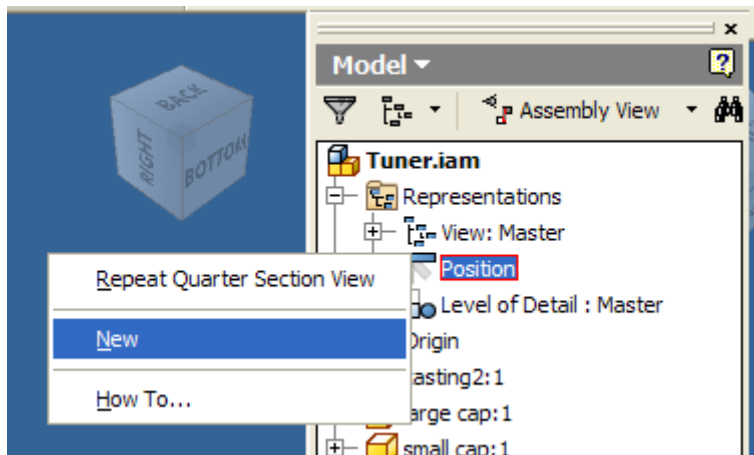
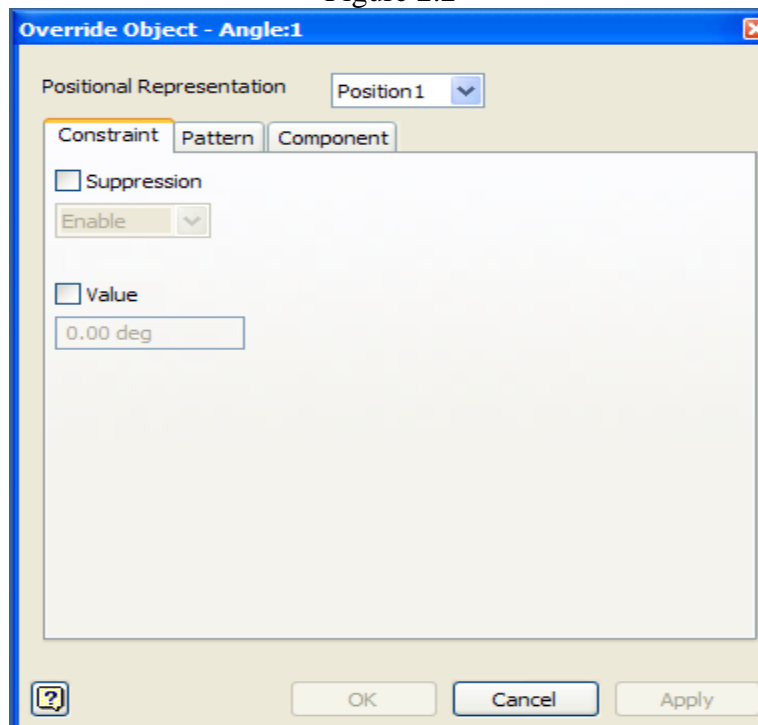


Figure 2.2



As an example I created “Position 1” and overwrote the value of the angle constraint for the knob. See figure 2.3 for the MASTER and 2.4 for the POSITION 1.

Figure 2.3

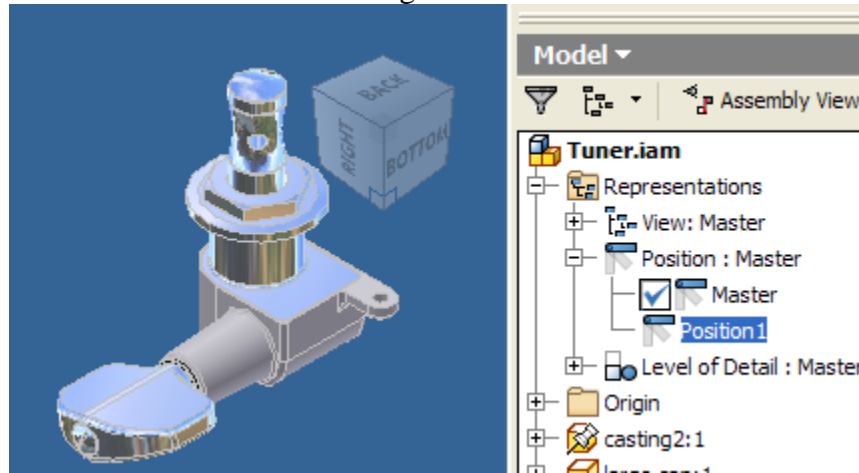
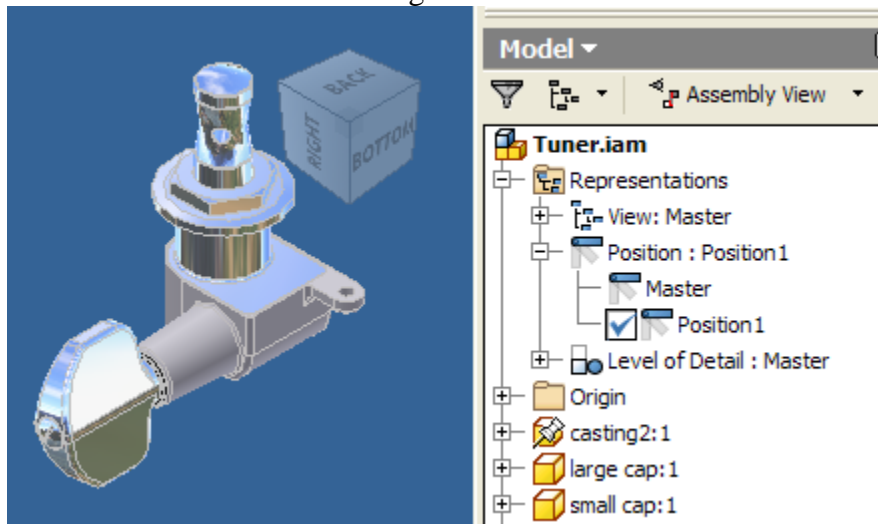


Figure 2.4

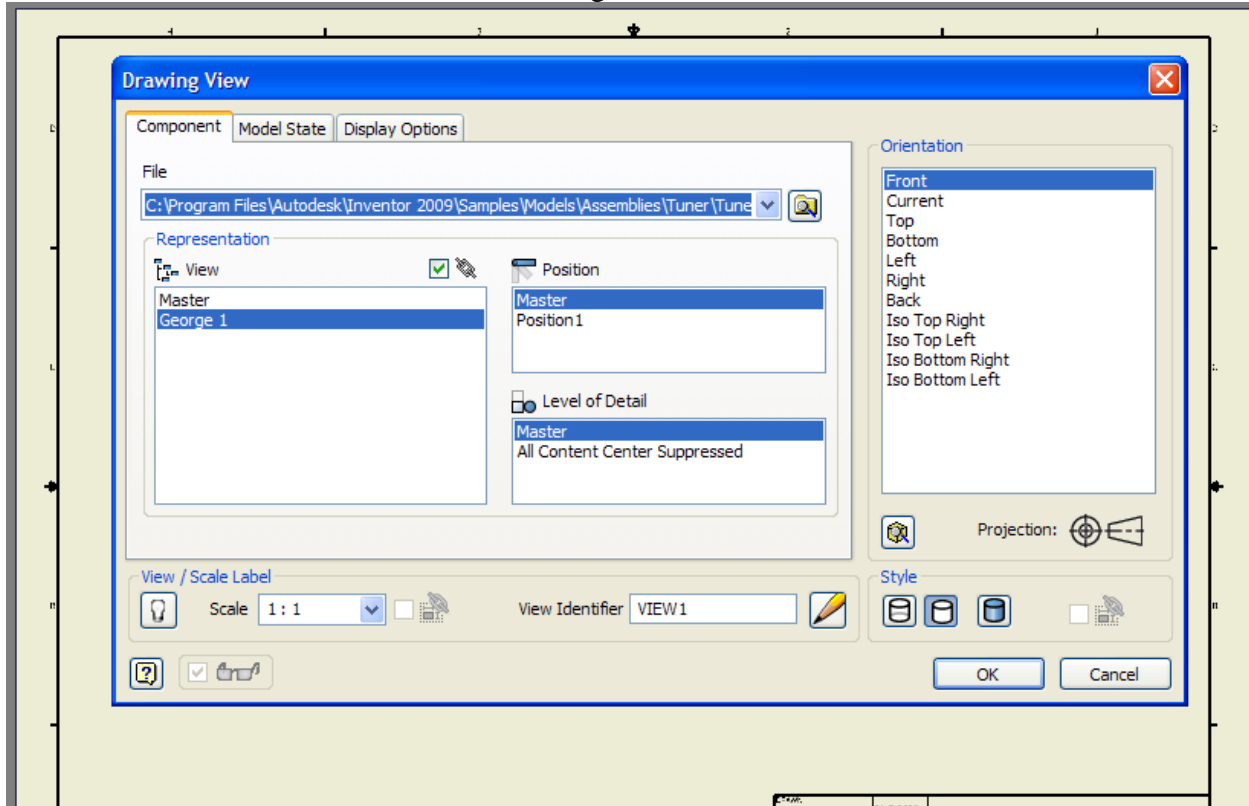


3. Placing design views and Positional representations in your drawings.

After creating design view and positional representations we have the power to add them to our drawings. If an assembly containing either Design views and/or positional representations is

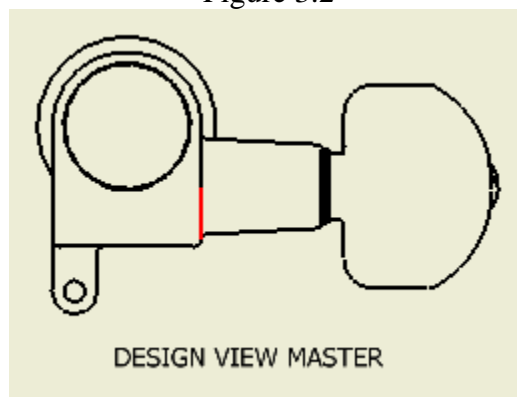
selected while placing a base view, Inventor automatically lists them for easy selection. See figure 3.1

Figure 3.1



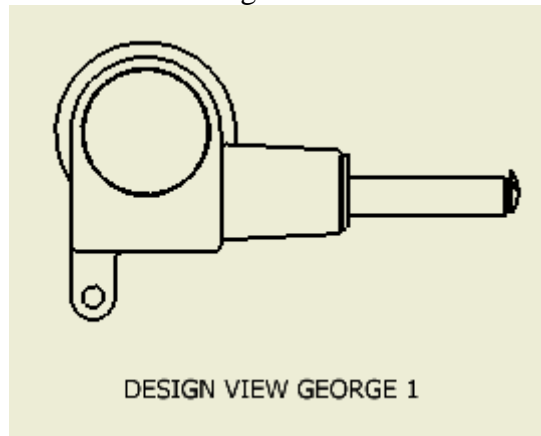
See figure 3.2 for the front view of the master design view.

Figure 3.2



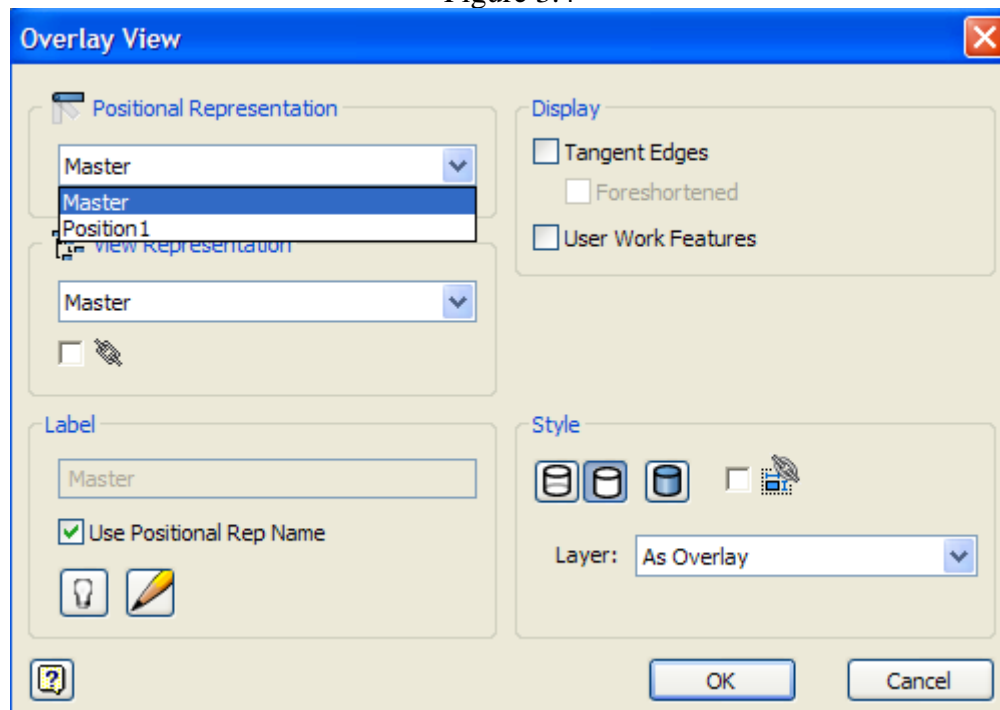
See figure 3.3 for the front design view of George 1.

Figure 3.3

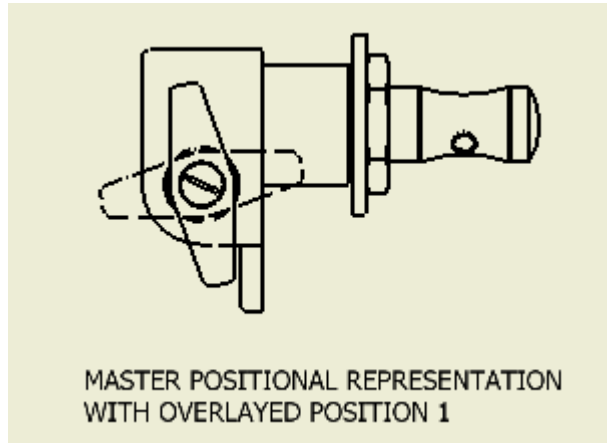


To harness the power of Positional Representations we will use a Overlay view, see figure 3.4.

Figure 3.4



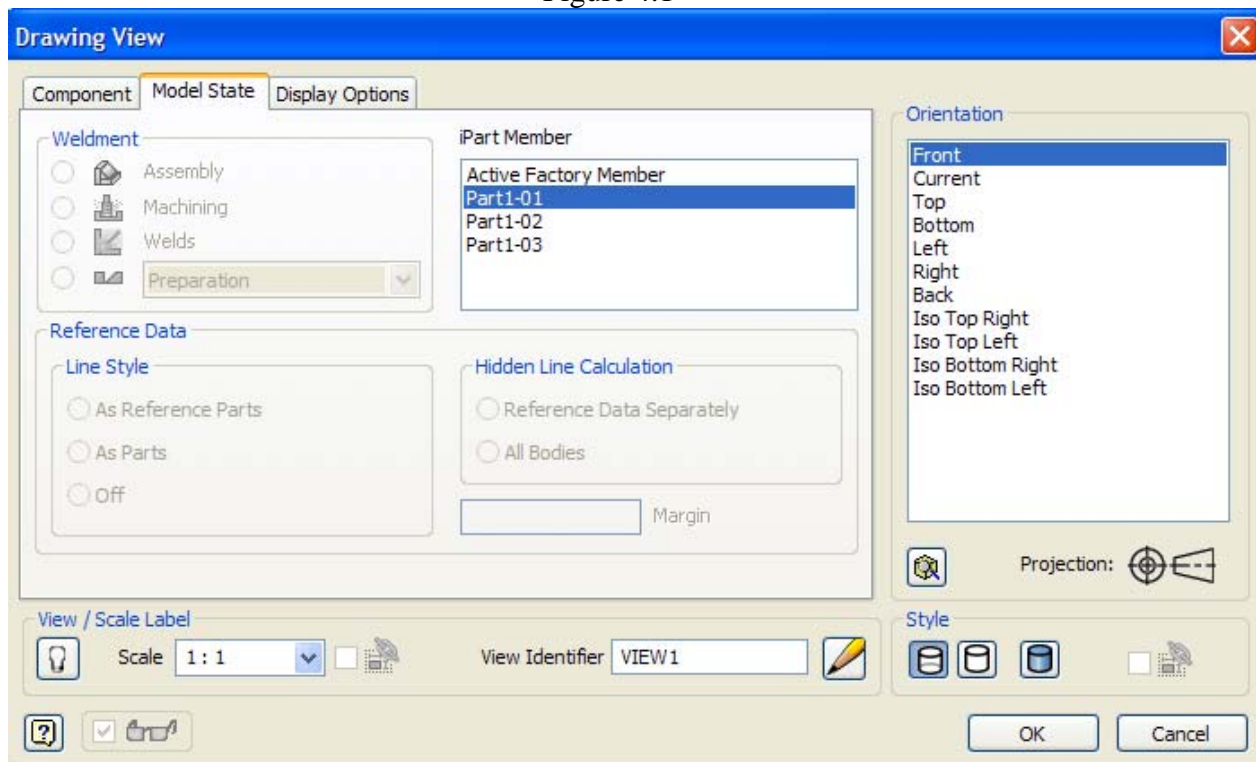
See figure 3.5 for the overlaid view showing the MASTER and “Position 1” positional representation, showing the world that the knob rotates.



4. Using I-part filters.

Additionally to design views and positional representations, there is a way to document all the different versions of an I-part inside a drawing. See figure 4.1.

Figure 4.1



See figure 4.2 for a sample of and I-part in a drawing, and figure 4.3 for the same I-part in an assembly.

Figure 4.2

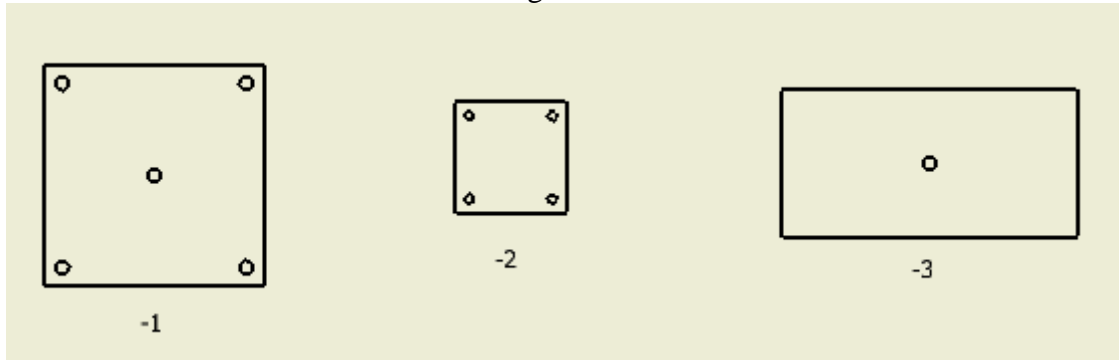


Figure 4.3

