Making the Transition from 2D

By Neil Munro

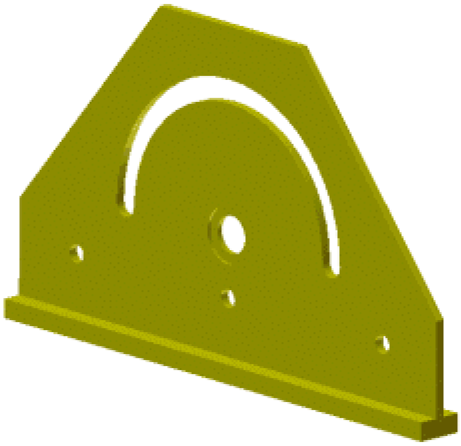

Figure 1: Completed part.

One of the challenges when moving from a 2D environment, such as the one in AutoCAD®, to 3D modeling in Autodesk Inventor™ is learning the different thought process required to *build* rather than *document* a model. In fact, you may have an easier time of it if you've got little or no 2D AutoCAD experience because you lack the ingrained habits of good 2D designers. The accompanying tutorial, New to Autodesk Inventor™, leads you through planning and creating a simple part model in Autodesk Inventor. The tutorial highlights some of the areas where you may need to think outside the AutoCAD "box" to be proficient with Autodesk Inventor.

Three Words: Planning, Planning, Planning
As with most things, a little planning goes a long way. The process of building a 3D part model typically involves a series of operations that either add material to, or remove material from, the model. There are many exceptions to this process, but the following steps comprise much of the workflow for many part models.

1. Create a closed-profile 2D sketch.
2. Add some rules (constraints and dimensions) as to how the sketch entities should behave.
3. Create a 3D solid from the sketch by either extruding the sketch perpendicular to the sketch plane, or revolving the sketch about an axis on the sketch plane.The resulting solid can add or remove material from the part.
4. Repeat as necessary.

The biggest hurdle is often where to start. Well, since the part will be modeled as a series of features, break it down into Autodesk Media and Entertainment features. Identify the one shape that best describes the part's form. That shape is usually a great candidate for your first sketch and feature.

Examine the part in Figure 1. What would your choice be for the feature that defines the part? To me, the plate with the chamfered corners is the most obvious feature and would be a reasonable place to start. If you look at the part from the end, an inverted "T" shape might define the part just as efficiently.

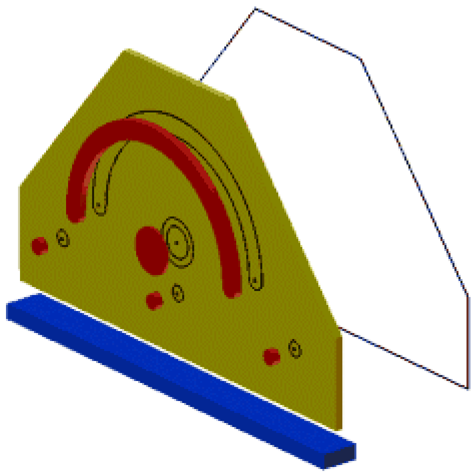

Figure 2: Feature breakdown.

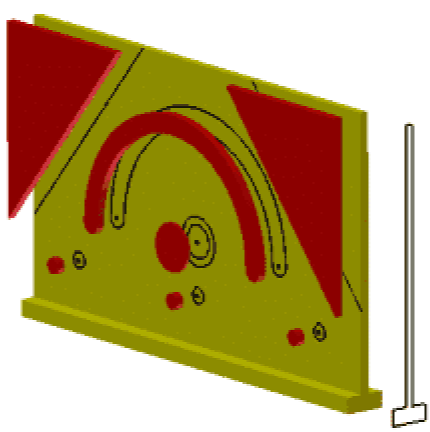

Figure 2 shows the part exploded into features. The image on the left shows the part starting with a sketch and extrusion of the plate outline. The image on the right shows the part starting with an extrusion of the T-shaped sketch. The feature created from the initial sketch is shown in gold; features that remove material are shown in red; and the feature that adds material is shown in blue. Note that the bottom rail is not included in the first extrusion in the left image; that's because it's a different thickness than the plate. There are advanced techniques you can use to overcome this, but better keep it simple when creating your first models.

If you were using AutoCAD software to document this part, you would likely start with the view of the face with the chamfered corners. In addition, the view would probably be drawn entirely in the positive XY quadrant of the WCS. It is also likely that you would draw entities from left to right and use symmetry where appropriate. When building a 3D model, using symmetry is important for efficient and robust designs.

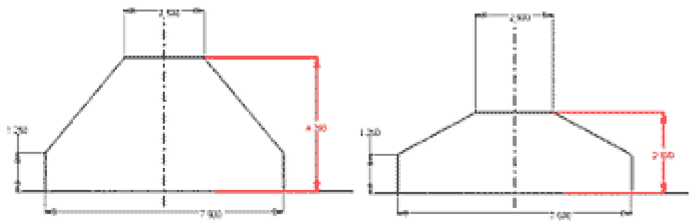

Figure 4: Dimension value drives sketch geometry.

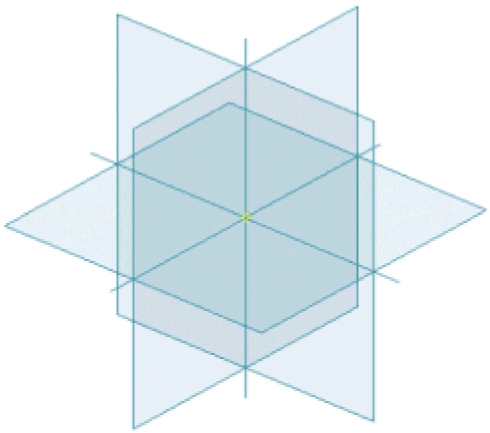

Figure 3: Reference geometry.

A new part file contains default reference geometry (planes, axes, and center point). It is good practice to build models around this geometry, aligning critical geometry with the reference axes or center point. You must *project* reference geometry into your sketch before it can participate in the sketch. Adopting a more "center-out" design philosophy will lead to the creation of robust 3D models.

AutoCAD 2D vs. Autodesk Inventor 2D
A drawing view created in 2D AutoCAD consists of precisely created geometry representing edges, points, and so on. To reduce the chance of being frustrated during sketch creation, you have to let go of the good 2D habit that everything must be drawn exactly. Unlike AutoCAD associative dimensions that can track changes to the underlying geometry, dimensions added to Autodesk Inventor sketches drive the size of the sketch geometry. Editing a sketch dimension value changes the size or position of the referenced geometry (Figure 4).

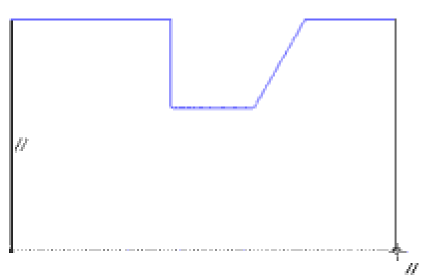

Figure 5: Parallel constraint and horizontal alignment.

Because changing a dimension value modifies the geometry, you can get some interesting results if the value change is large. Sketching roughly to scale reduces the chance of odd editing behavior. Use the following as a guide to editing sketch dimensions.

Modify dimensions in the following order:

- Big dimensions that will be made bigger.
- Small dimensions that will be made smaller.
- Small dimensions that will be made bigger.
- Big dimensions that will be made smaller.

In addition to dimensions, other rules govern the relationship between sketch entities. Geometric constraints, such as parallel, concentric, and coincident, are added among lines, curves, and points as you sketch. You can infer horizontal or vertical alignment to points, or change the entity referenced in a geometric constraint, by moving the cursor over desired geometry (without clicking it). See Figures 5 and 6.

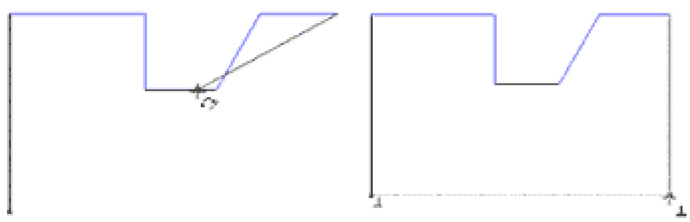

Figure 6: Infer other geometry (left); perpendicular constraint (right).

You can manually add geometric constraints. You can also delete constraints that may be enforcing rules that are no longer required. Where possible, add geometric constraints instead of using multiple dimensions.

Three More Words: Testing, Testing, Testing
Autodesk Inventor indicates the state of sketch geometry by color. As rules (constraints and dimensions) are added to a sketch, the orientation and size of entities become fixed. As long as you are renouncing AutoCAD habits, you might as well pick up this Autodesk Inventor habit: drag underconstrained geometry (curves, lines, end-points of those lines, and so on) to determine how the geometry can change. The additional constraints and dimensions required are usually readily apparent. See Figure 7.

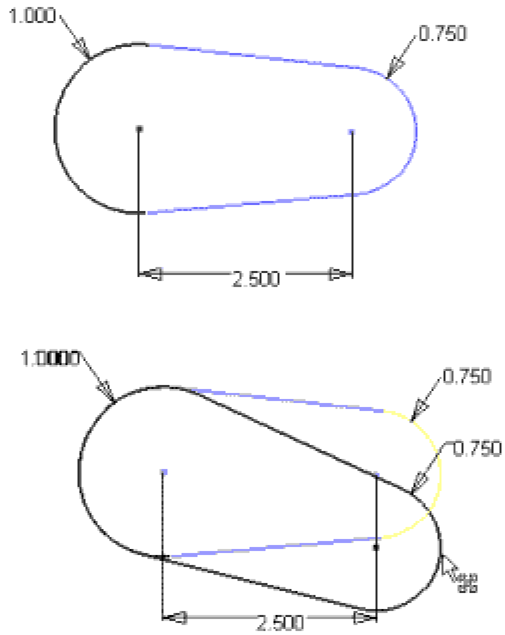

Figure 7: Dragging underconstrained geometry.

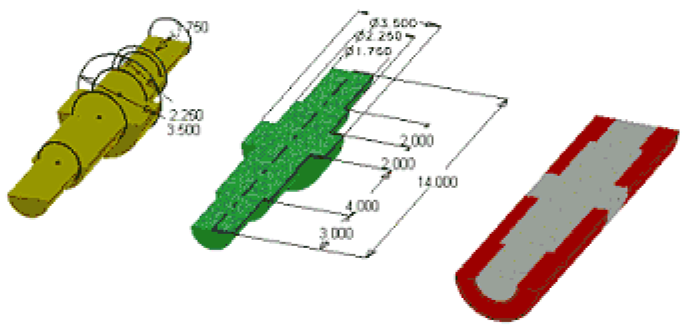

Figure 8: Modeling methods.

There are instances where some sketch geometry is intentionally left in an underconstrained state (e.g., when using adaptive features). This usually involves leaving out one or more sketch dimensions.

Keep It Simple
Because an AutoCAD 2D drawing view may contain all the part edges visible in the view, you may assume it is good practice to add all these edges in the first feature sketch in an Autodesk Inventor part file. It is not. Resist the urge to define every possible feature in the first sketch. Simple sketches are far easier to constrain and tend to create models that are easier to maintain. As you gain experience you will learn where and when to add complexity to a sketch.

Summary
To sketch efficiently using Autodesk Inventor:

- Keep your sketches simple. Once you are comfortable with the sketching process,

investigate more complex sketches and the concept of sharing sketches.
- Sketch features roughly to size and shape but don't try to be too precise. This is especially true for the first feature sketch in a part. It is easier to judge the scale of subsequent features relative to existing geometry.
- Use construction geometry to help position and size normal sketch geometry. Construction geometry is not considered when a solid is created from the sketch. It can reduce the number of dimensions required and the need for equations between sketch dimensions.
- Infer relationships while sketching by moving the cursor over the point, line, or curve you want to reference.
- Keep track of the geometric constraints placed as you sketch. Add additional geometric constraints as required.

Tip: You can disable automatic sketch constraints by holding down the Control key.

- If adding a constraint or dimension distorts the sketch, undo the action and place other constraints or dimensions to stabilize that portion of the sketch before proceeding.
- Fully constrain sketches unless the feature derived from the sketch will need to adapt.
- Experiment—it's fun!

OK, Now, Think About This...

Examine the three versions of the shaft shown in Figure 8. The left image has the shaft modeled as a series of stacked cylinders (all shafts have been split for clarity). The middle image shows a single sketch revolved about an axis. The image on the right mimics the real-world process: a piece of cylindrical bar stock is machined into the finished form (red tubes subtracted). Which approach would you use? Perhaps you have some other method?