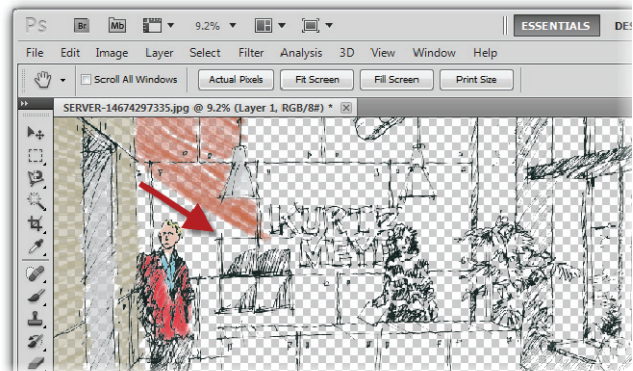
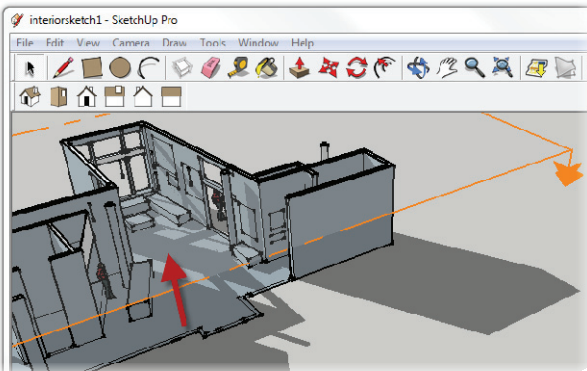
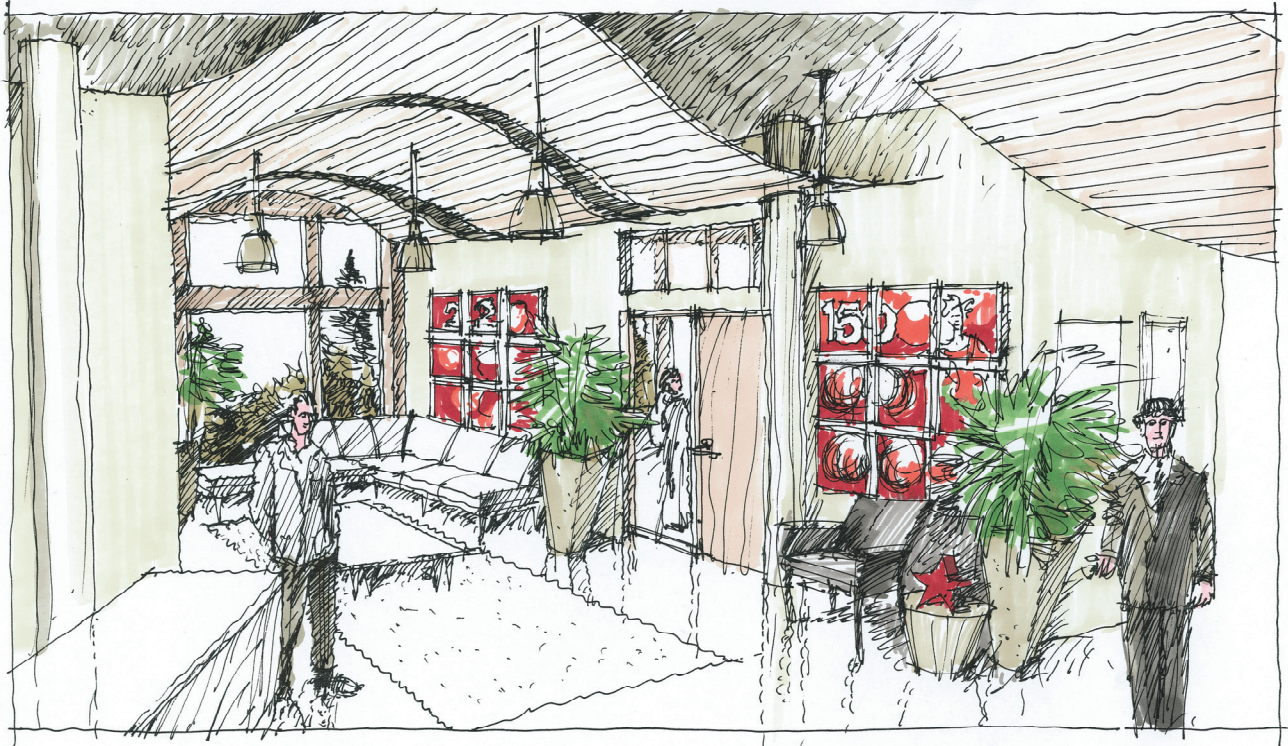


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SketchUp Introduction Part II

This chapter will build upon what was covered in the previous chapter. You will begin to learn how to create geometry and modify it. A very basic introduction will be presented, with additional information provided later in the book as the need arises.

Section 4-1 **The Basic Entities**

Given the amazing images one can create using SketchUp, it is somewhat surprising that there are mainly just eight types of entities that can be added to a model.

They are:

- Edges
- Surfaces
- Annotation
 - Dimensions
 - Text
 - 3D Text
- Components
- Groups
- Guide (reference line)

The next few pages will provide a brief overview of each of these entity types.

Edges

SketchUp is a face-based program, and all surfaces (i.e., faces) must be defined by an edge. This is the fundamental building block of a SketchUp model.

Edges are created with one of the draw tools:

- Line
- Circle
- Arc
- Rectangle
- Polygon
- Freehand

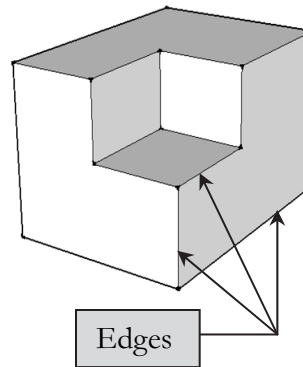
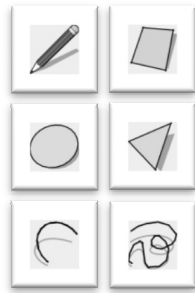


FIGURE 4-1.1 Edges pointed out

Edges can be created with very specific length (or radius) or arbitrarily by clicking anywhere within the drawing window. It is easy to snap to one of the three planes (axes) while drawing lines. This makes it possible to draw 3D shapes from a single 3D view (more on this later).

A basic cube has twelve edges. Figure 4-1.1 has 21 edges and Figure 4-1.2 has two edges.

Even circles are made up of small edges. When one is being created, the *Value Control Box* (on the Status Bar) lists the number of sides that will be used to approximate the circle. This number can be increased to make larger circles smooth, or decreased to make smaller circles less complex (which can be a burden on model performance).

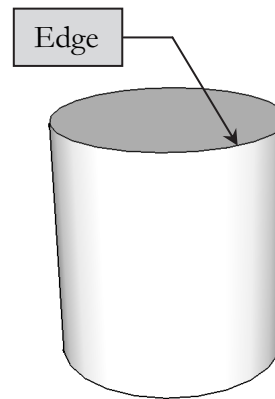


FIGURE 4-1.2 Edge pointed out

Later in this chapter you will get some practice drawing edges and editing them.

Edges can be modified with a number of tools. For example, an edge can be scaled, rotated, divided, copied, offset and erased. These tools are accessible from the *Tools* menu, toolbars (which may not be visible yet), right-click menus (when the edge is selected) and keyboard shortcuts.

Edges can also be placed on *Layers* in order to control visibility. A *Layer* can be turned off, making everything assigned to that *Layer* invisible.

Surfaces

A surface is the second most significant type of entity in SketchUp. You might be surprised to learn that no tool exists to create a surface! They are created automatically when the conditions are right.

What are the conditions in which a surface is automatically created? The simple answer is **when a series of edges form an enclosed area**. When the last edge is drawn which defines an enclosed area, a surface is created. This can be as few as three edges – forming a triangle.

The image to the right (Figure 4-1.3) shows two examples of three connected edges defining a *surface*. Note the edges can be a combination of straight and curved lines.

If an *edge* is erased, the *surface* is also erased, seeing as it no longer has a boundary.

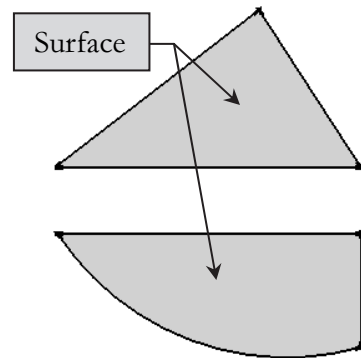


FIGURE 4-1.3
Surfaces defined by at least three edges

In addition to *edges* forming a closed perimeter, there is another important requirement a new modeler needs to be aware of, That is, **all the lines forming the enclosure must be coplanar**.

If you don't already know, the easiest way to describe coplanar is to think of all the edges as lines drawn on a flat piece of paper. As long as all the lines are in the same plane (i.e., on that flat piece of paper) a *surface* will be created.

Surfaces may have materials painted on them. They can also be placed on *Layers* in order to control visibility.

A surface can be deleted; simply select it and press the **Delete** key on the keyboard. The only way to get another surface is to draw a line directly on top of one of the existing *edges*. SketchUp will then create a surface and delete the extra line, as it does not allow two lines to exist directly on top of each other.

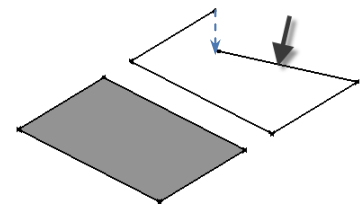


FIGURE 4-1.4
Coplanar on left; not on right

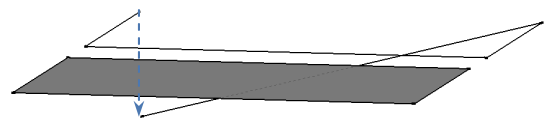


FIGURE 4-1.5 Another angle of Figure 4-1.4

Dimensions



Dimensions can be added to your SketchUp model. These are smart entities; they are not sketched lines and manually entered text. A *dimension* entity becomes a permanent part of the model, unlike the *Tape Measure* tool (which is used to list distances without drawing anything).

To place a dimension you simply pick three points; the first two are what you want to dimension and the third is the location of the dimension line and text. SketchUp automatically displays the correct length.

The dimensions are associative, relative to the first two points picked. The dimension will grow or shrink if the geometry is modified. However, if the geometry is deleted, the dimension will remain (but is no longer associated to anything).

Dimensions can be tricky in that they may appear correctly and legible from one angle (Figure 4-1.6a) but not another (Figure 4-1.6b). But the visibility of a *dimension* can be controlled by *Layers* or by *Scene* (more on what scenes are later).

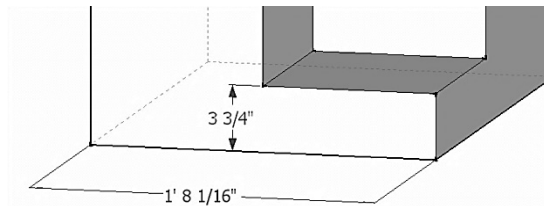


FIGURE 4-1.6A Dimensions added while viewing the model from this angle

To adjust various settings related to how dimensions are created, go to **Window** → **Model info** and then click *Dimensions* in the list on the left.

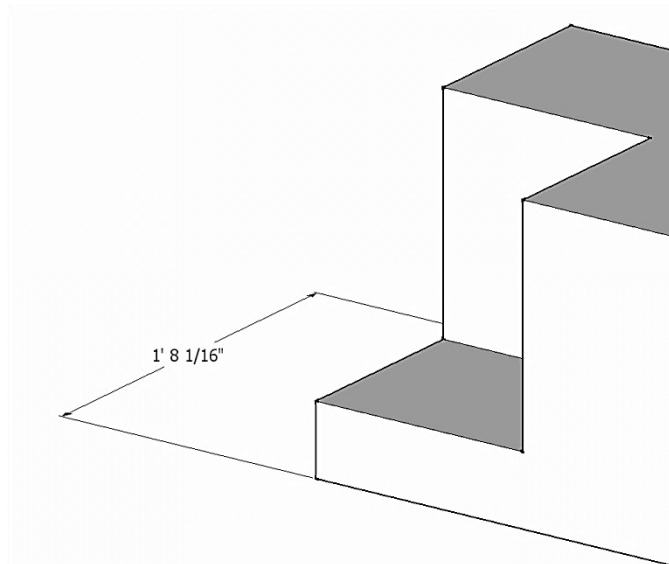


FIGURE 4-1.6B One dimension still visible when view angle is changed using orbit

Text

SketchUp has a tool which allows you to add notes with leaders (i.e., pointing at something). To place a *Text* entity you make two clicks and then type (or accept the default value). Default value? If you point to a *surface*, SketchUp will automatically list the area of that surface. If an *edge* is pointed to, SketchUp will list its length. An example of each can be seen in the image below (Figure 4-1.7).

If you don't want a leader, simply click in empty space and you can just type text. The text "Option A" is an example of text without a leader (Figure 4-1.7).

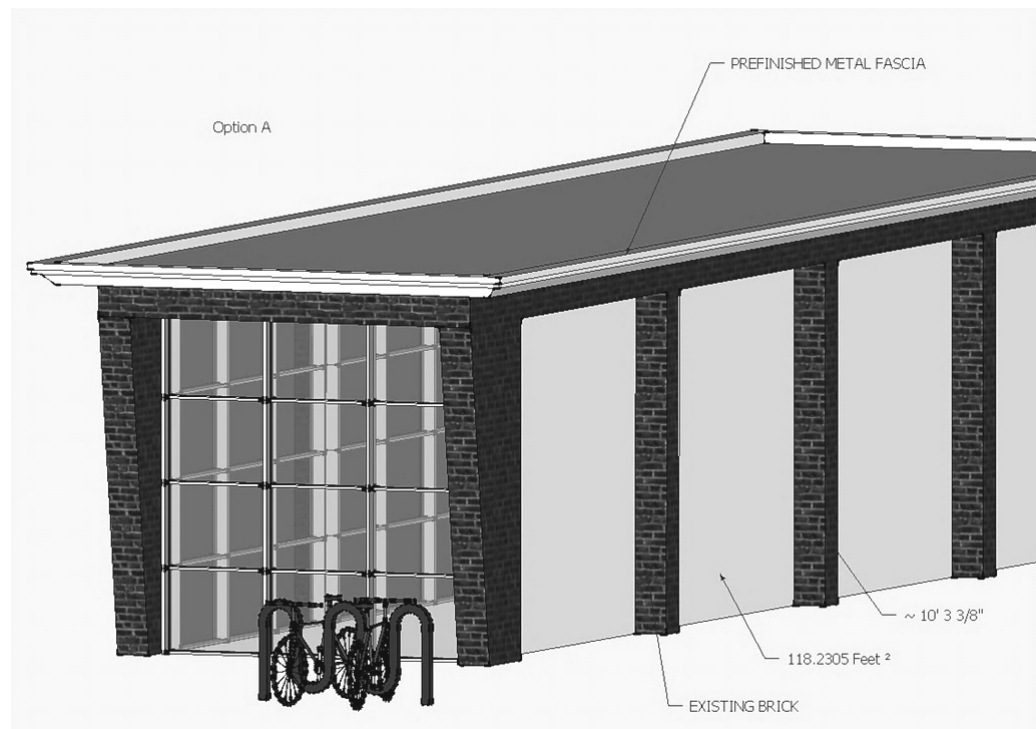


FIGURE 4-1.7 Notes added using the Text tool

Similar to dimensions, text entities remain visible when the vantage point is changed, using *Orbit* for example. As you can see in the Figure 4-1.8, this can get a bit messy. *Text* visibility can also be controlled with *Layers* and *Scenes* like *dimensions*.

For the most part, notes and dimensions are left until the end of the modeling or added outside of SketchUp – in LayOut or a CAD program such as AutoCAD or Revit.

INTERIOR DESIGN USING HAND SKETCHING, SKETCHUP AND PHOTOSHOP

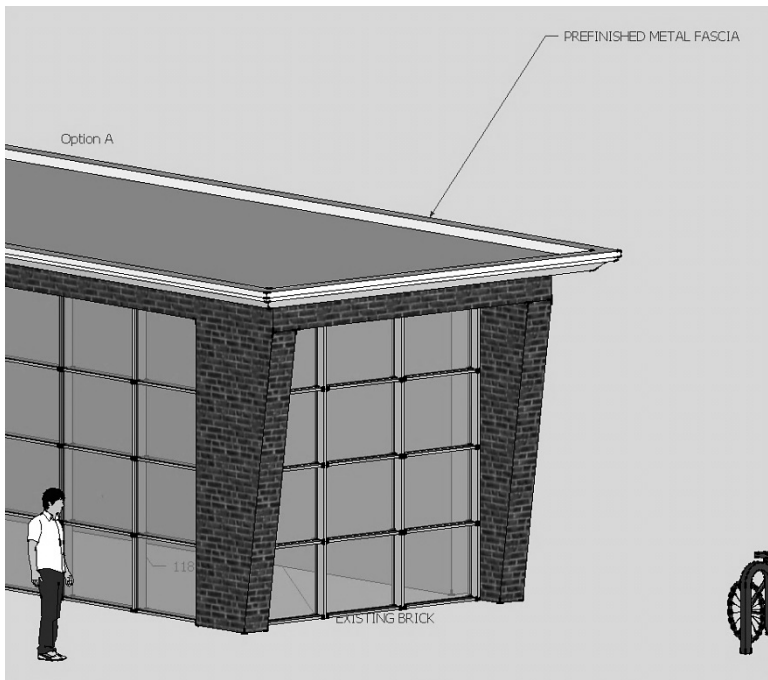


FIGURE 4-1.8 Notes still visible when model rotated using orbit

When text is right-clicked on, a menu pops up which allows you to change the arrow type and leader (Figure 4-1.9). These options, plus the ability to change the font, are available via the *Entity Info* dialog. This can be turned on from the Window pull-down menu.

The default settings for text can be changed via **Window** → **Model Info**, and then selecting *Text* from the list on the left.

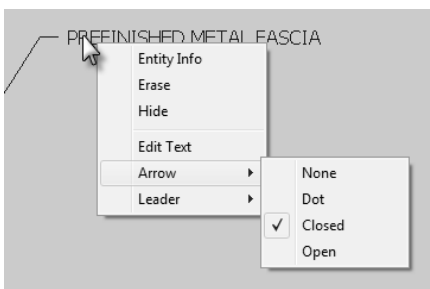


FIGURE 4-1.9
Right-click options for text

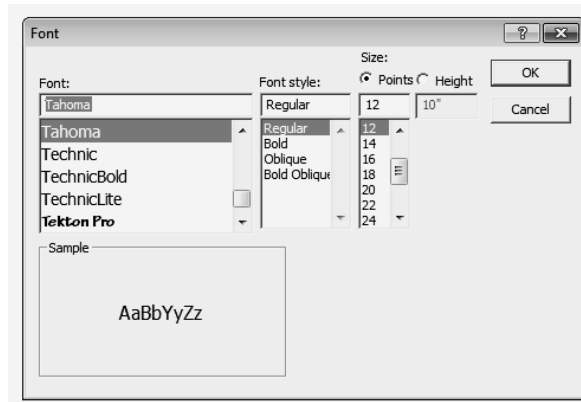


FIGURE 4-1.10 Changing font for text entity

3D Text



The *Text* tool, just covered, is meant for notes and comments about the model. *3D Text* is meant to be part of the model. This tool is used to model text on signs or letters on the face of a building. Unlike notes created with the *Text* tool, *3D Text* stays right where you put it.

Placing 3D Text is easy. Select the tool, and the *Place 3D Text* dialog appears (Figure 4-1.11). Enter your text and select the options desired for font style, height and thickness (i.e., extruded). Click **OK** and then pick a location on a face to place it.

Once the 3D Text is placed, the *Paint Bucket* tool can be used to apply a material.



FIGURE 4-1.11 Adding 3D Text



FIGURE 4-1.12 3D Text placed

Once the text is created, it becomes a component that cannot be easily edited (in terms of typing new words).

It is possible to see the properties for 3D Text, or anything else selected, using the *Entity Info* dialog (Fig. 4-1.13). This can be turned on from the Window pull-down menu. The information presented varies depending on what is selected. This dialog can remain open while modeling.

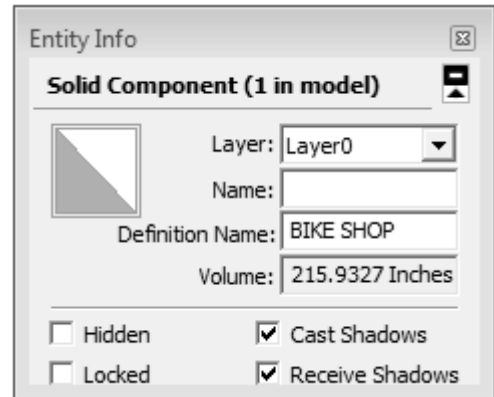


FIGURE 4-1.13 Entity Info dialog

Components

In SketchUp one can think of *Components* being something like clipart in a word processing program – but clipart on steroids! They are pre-built models which can be reused in your SketchUp model. Some components are flat 2D models while others are complex 3D models. The simple, flat components reduce the resources required of your computer, making it easier to smoothly orbit and inspect your model. For example, many of the trees designers use in SketchUp are 2D due to the number typically needed. If 3D trees were used, the file would be large and unmanageable. The 2D components can be setup so that they always face you – plus they cast shadows (see Figures 4-1.14 and 4-1.15).

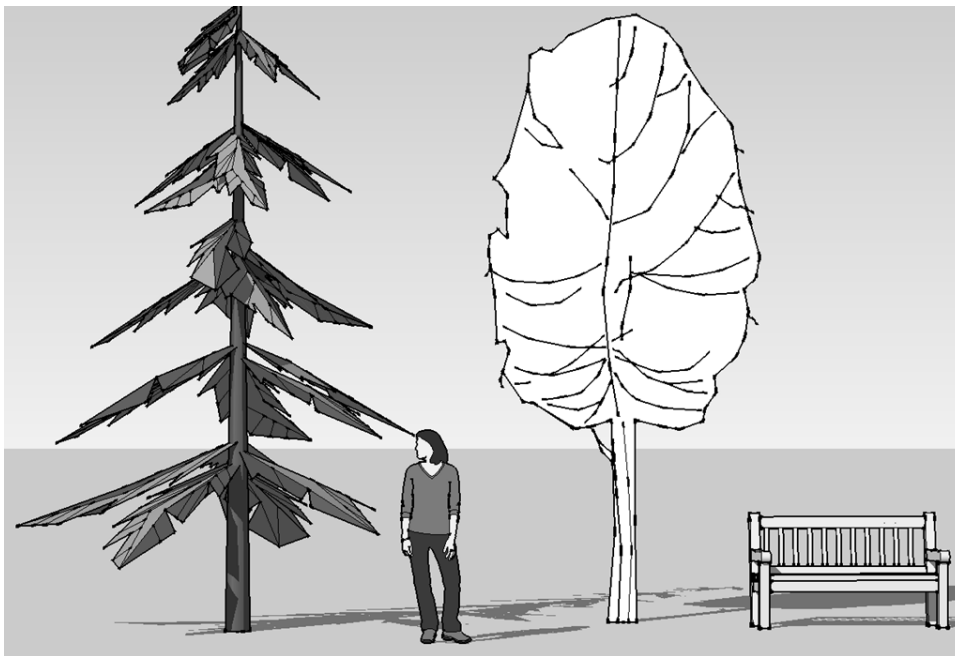


FIGURE 4-1.14 2D vs. 3D components; two items are 2D and two are 3D.



FIGURE 4-1.15 Rotated view of previous image

Right-clicking on a *component* allows you to edit it, explode it (reduce it down to individual entities) and add parameters and parametrics using the advanced ***Dynamic Components*** functionality.

Editing a *component* causes all instances of that component, in your model, to instantly update. You will see an example of this in the next section.

One of the truly great things about using SketchUp is the amount of content the designer has access to. Google hosts a site called *Google 3D Warehouse* which has thousands of components ready for the taking.

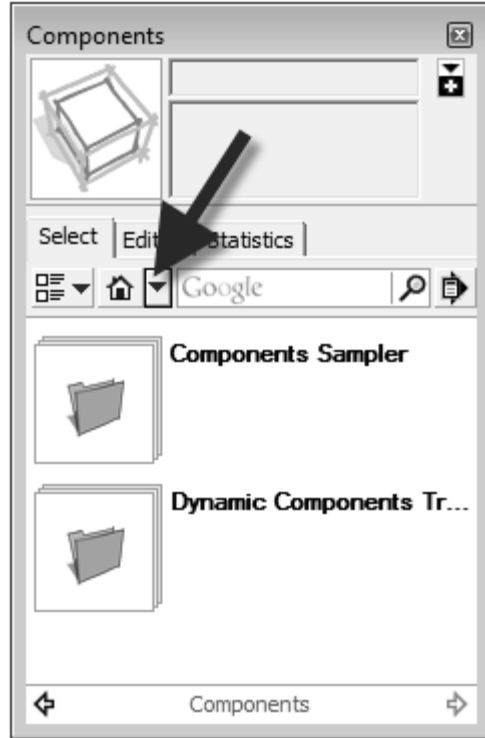


FIGURE 4-1.16 Components dialog

Some of the content found on *Google 3D Warehouse* is provided directly by Google, while other content comes from manufacturers of products (who hope you will ultimately buy or specify their products) or from end users like you.

Of course, *users beware* on anything one downloads and uses in their design. As a design professional (or would-be, someday, design professional) you are responsible for code and performance compliance. So you cannot just assume the toilet or the door you downloaded is the correct size. You need to double check it with the manufacturer’s data sheets. Now, if the content was created by the manufacturers, it is highly probable that it is the correct size.

The *Components* dialog (Figure 4-1.16) is the easiest way to add components to your model. This can be accessed from the Window menu. The down arrow highlighted reveals a menu which provides shortcuts to groups of content, such as Architecture, People, Playground, etc.

It is also possible to search for components. You may be surprised at what you can find. Figure 4-1.17 shows some of the results when searching for “**pizza**”! Notice the author of the component is listed directly under the name.

Try a few searches to see what you can find – maybe try goat, newspaper, or snowboard.

See the next section for more on *components*.

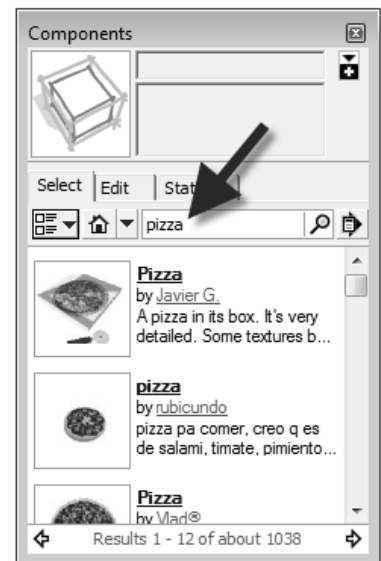


FIGURE 4-1.17 Components search

Groups

A *Group* is similar to a *Component* in that you can select one part of it and the entire representation is selected (selecting potentially hundreds of entities with a single pick). However, that is about all that is the same between them.

Groups are meant for one-off type items. That is, a unique reception desk, a built-in entertainment center, etc. A *Component* is used when your model will contain many instances of an object.

Both *Groups* and *Components* are easy to create. You simply model something, select it and then right-click (on it). At this point you can select either **Make Group** or **Make Component** (Figure 4-1.18).

Both *Groups* and *Components* can be copied around the model (using the *Move* tool and holding down Ctrl). They both can also be edited by right-clicking and selecting “edit” from the pop-up menu.

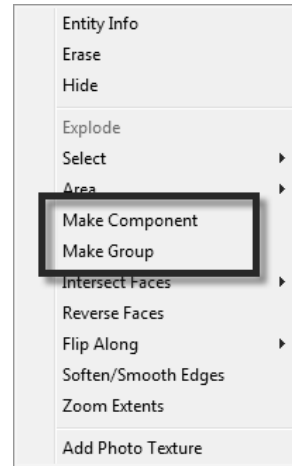


FIGURE 4-1.18
Right-click menu

It is important to note that editing a *Group* only changes the specific *Group* you are editing. But editing a *Component* instantly causes all instances of that *Component* to update (see Figures 4-1.20 and 4-1.21). This means SketchUp duplicates all information required to define each copy of a *Group*. A single definition is all that is needed for multiple instances of a *Component*. Of course, this means a file with many copies of a *Group* will be larger than one with many copies of a *Component*.

The main thing to keep in mind is that *Groups* are quick and require minimal decisions. *Components* can be much more sophisticated and take a lot of time setting up (creating parameters and parametric relationships, and adding formulas).

When you right-click and select *Make Group*, SketchUp just makes it without asking any questions. It can be selected and named via the *Entity Info* dialog if you wish.

When creating a *Component*, the *Create Component* dialog appears (Figure 4-1.19). Notice the various options:

- Glue to – does the tree stick to the ground or float in the air?
- Always face the camera – this is ideal for flat two-dimensional items.
- Replace selection – turn the current selection into one of the *Components* you are creating.

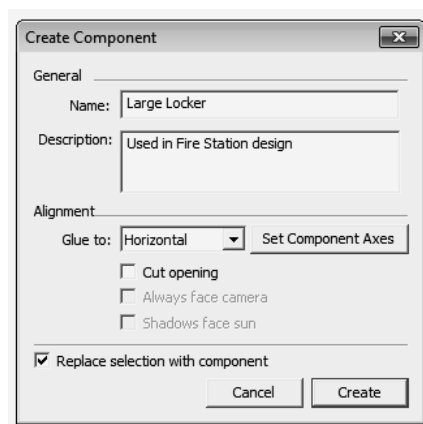


FIGURE 4-1.19 Right-click menu

The images below compare what happens when a *Component* is edited versus a *Group*. Notice all instances of the *Component* are updated, whereas only the selected *Group* being edited is updated (even though the other *Groups* are copies of the one being edited).

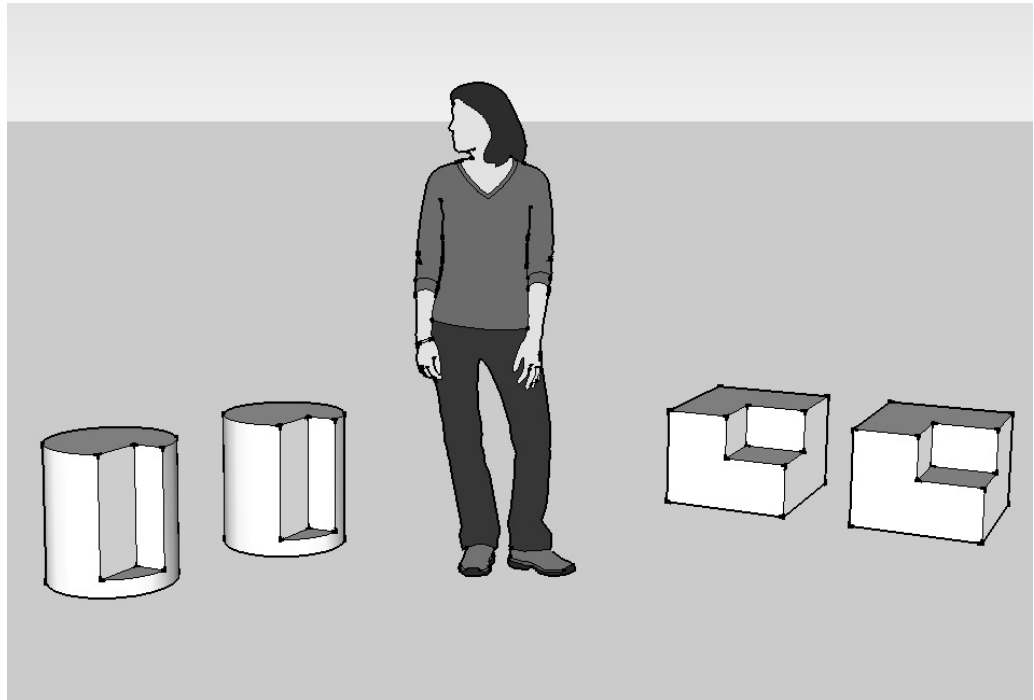


FIGURE 4-1.20 Components vs. groups – components on the left, groups on the right

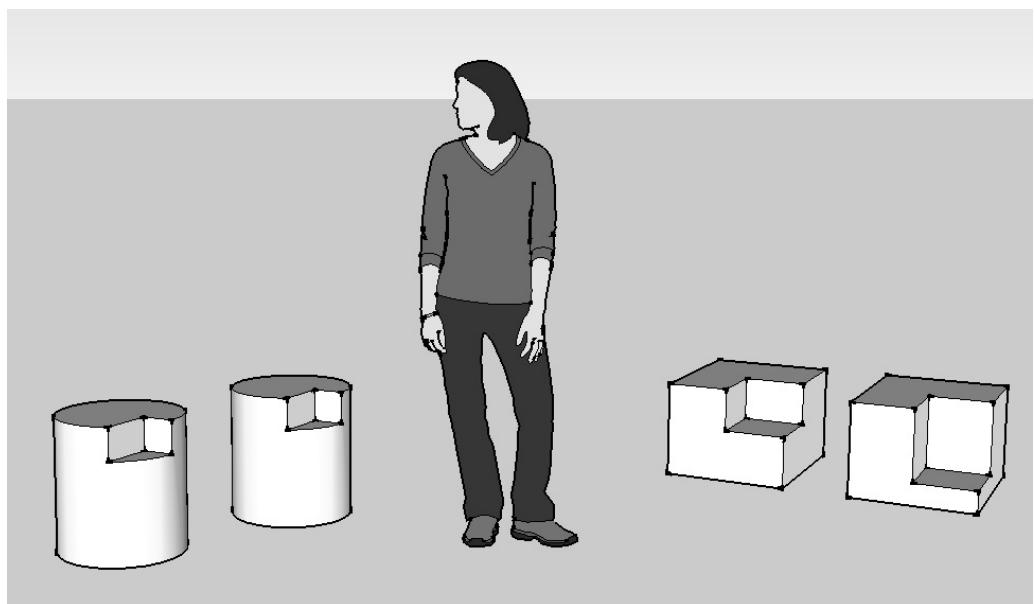


FIGURE 4-1.21 Components vs. groups – all components update, only selected group updates

Guide

Guide lines (or construction lines) are useful for new users and for a general design reference grid. The image below (Figure 4-1.22) shows the main *Axes* and a few *Guides* at 5'-0" intervals. These lines are parallel to the main *Axes* and are infinite in length. Note how they converge at the horizon line.

The *Tape Measure* tool is used to create *Guide* lines. Follow these simple steps to create one:

- Start the *Tape Measure* tool.
- Click on the *Edge* of any shape or *Axes*
FYI: Clicking on an endpoint creates a Guide Point.
- Drag the cursor perpendicular to where you want the *Guide*.
- Release the mouse to locate the *Guide*.
- Type in a length to (retroactively) adjust the *Guide* location.

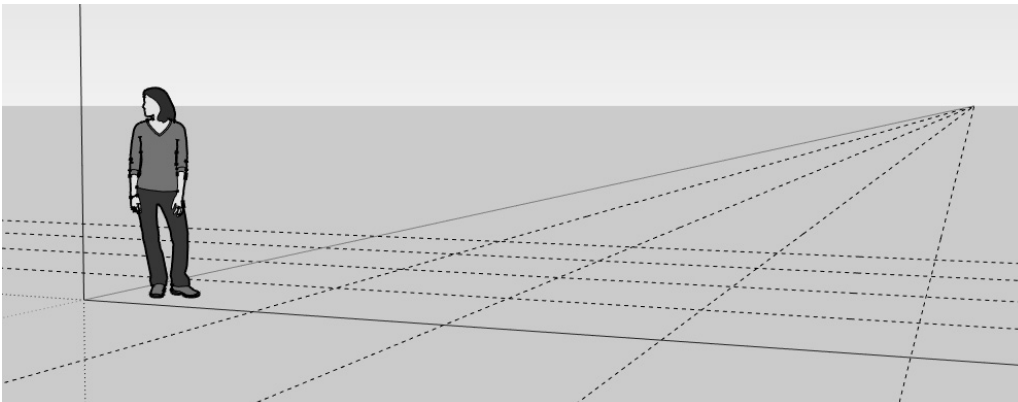


FIGURE 4-1.22 Guides added at 5'-0" intervals

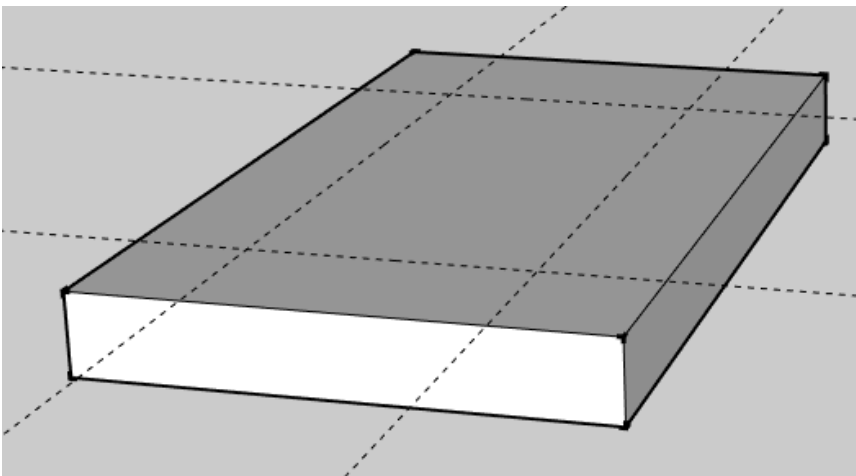


FIGURE 4-1.23 Guides added on top of a surface

Guides can be selected and deleted. They can also be relocated with the *Move* tool. They can be rotated with the *Rotate* tool as well.

You can quickly hide the *Guides* via the *View* menu (Figure 4-1.24). Notice the *Axes* can also be toggled off/on here as well.

Guides can also be placed on a *Layer* and hidden. This would allow you to hide some *Guides* while leaving others visible. Simply create a *Layer* using the *Layer* dialog (*Window* → *Layers*). Then select the Guide(s) and switch them to another *Layer* via the *Entity Info* dialog (see page 4-7).

If you can see *Guides* on the screen, they will print. You need to hide them before printing if you do not want them to print.

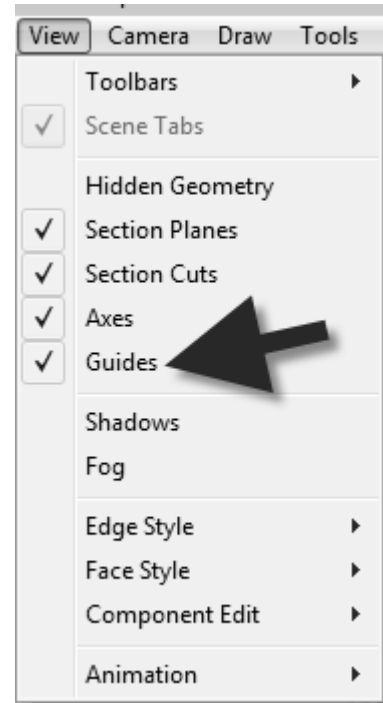


FIGURE 4-1.24
Guides visibility

Section 4-2

Beginning with the Basics

In this section you will practice sketching basic 2D lines and shapes to get the hang of using a few of the draw and modify tools, as well as specifying specific dimensions. In the next section you will circle back and see how easy it is to turn these 2D sketches into 3D drawings.

Setting up the model

The first thing you need to do is start a new model and make a few adjustments. You will complete these steps for each drawing in this section, unless noted otherwise (UNO).

1. Start a new SketchUp model using the **Architectural Design – Feet and Inches** template.
2. Select the person *Component*, and press the **Delete** key on the keyboard.

To break things down into the simplest terms, you will change to a non-perspective plan (or top) view.

3. From the *Camera* menu, select **Parallel Projection** (Figure 4-2.1).
4. Also from the *Camera* menu, select: **Standard Views → Top**.

You can also go to *View → Toolbar → Views* to turn on a toolbar which provides quick access to the standard views (top, front, iso, etc.).

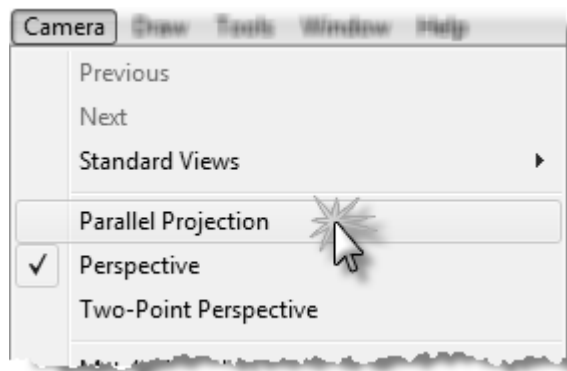


FIGURE 4-2.1 Parallel Projection mode

You are now looking at a plan view (Figure 4-2.2). This view is similar to what you would see on a printed out floor plan (aka, blueprints or construction documents). Use caution not to press and drag your center wheel button, as this action will activate the *Orbit* tool and throw you out of *Top* view; you would still be in “parallel” mode however. If you accidentally do this, simply select “top” again from the *Camera* menu. Selecting *Undo* does not help.

Notice how the axes are centered on the screen.

You can adjust which part of the model you are looking at using the *Pan* tool. When finished panning, click the *Select* icon. Do not pan at this time.

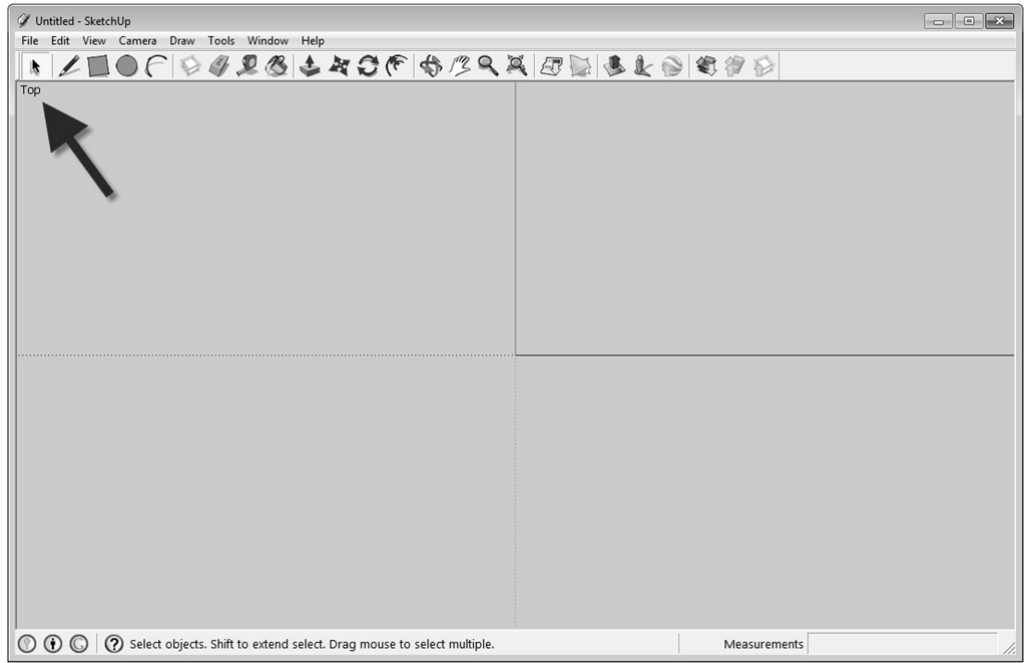
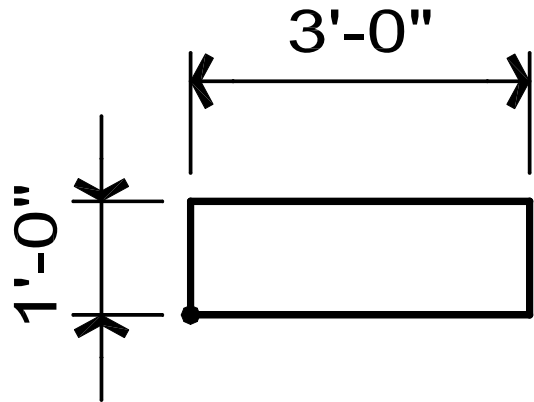


FIGURE 4-2.2 Top view in parallel projection mode

file name: **Bookcase**

This is a simple rectangle that represents the size of a bookcase. The black dot represents the starting point, which should align with the intersection of the axes. **Do not draw the black dot.**



5. Select the **Rectangle** tool.

6. **For your first point**, click the intersection of the axes (Figure 4-2.3).
 - a. Be sure your cursor snaps to the *Origin*; you will see a yellow circle and a tooltip appear.
7. **Select your second point** approximately as shown in Figure 4-2.3).
 - a. You can keep an eye on the dimension box in the lower right, but do not worry about getting the number exact as that will be done in the next step.

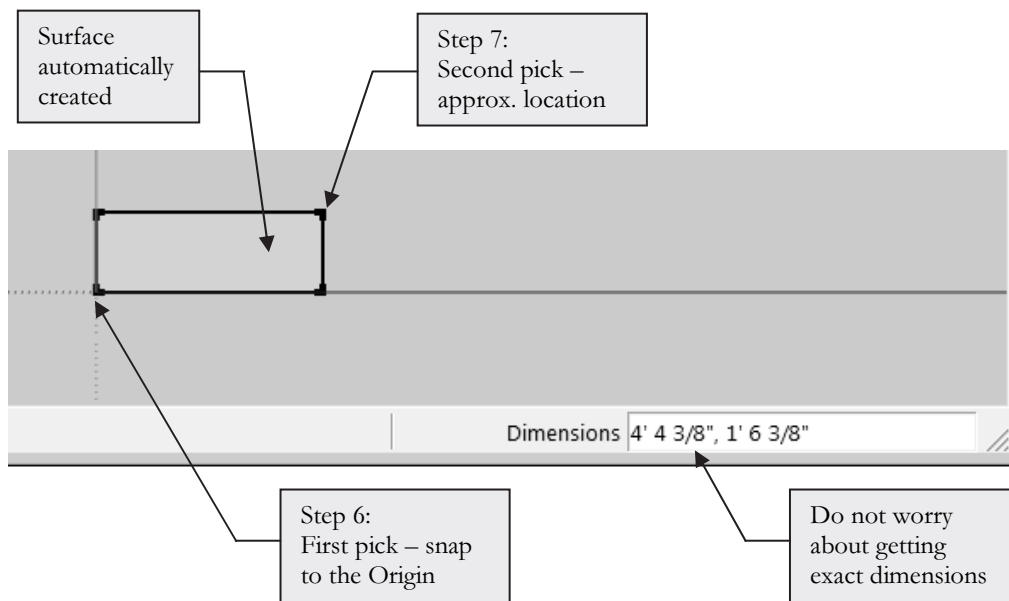
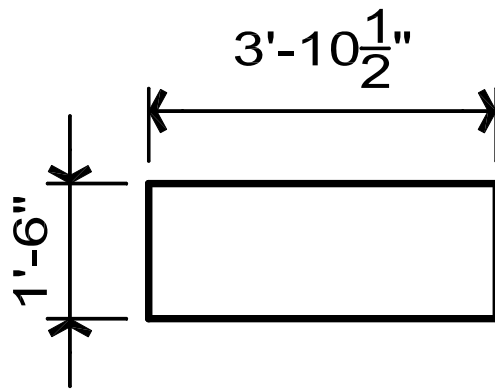


FIGURE 4-2.3 Sketching a rectangle

8. After clicking the second point (step 7) and before doing anything else, simply type **3',1'** and then press **Enter**.
 - a. You do not need to click in the *Dimensions* box; just start typing.
9. Save your file as **Bookcase.skp**.

Notice the surface which was automatically created once an enclosed area was created. You are done with this file for now. You will come back to it later and turn it into a 3D bookcase.

file name: **Coffee Table**

Next you will start a new file, using the steps previously covered (Setting up the Model; steps 1-4). The previously created file can be closed and set aside for use in the next section.

The *Coffee Table* drawing will introduce you to entering fractional values.

10. Start a new model, following steps 1-4.

It would be fairly easy to use the *Rectangle* tool again to draw this item, however you will use the basic *Line* tool so you can see how it works and get practice entering specific lengths.



11. Select the **Line** tool from the toolbar.

12. Snap to the *Origin* (the intersection of the red and green axes).

13. Begin moving your cursor to the right (Figure 4-2.4):

- a. Ensure your cursor is “snapped” to the horizontal.
- b. When horizontal, you should see the “**On Red Axis**” tooltip.
- c. Once you are pointing in the correct direction and snapped to the horizontal, you may type in a length (see the next step for this).

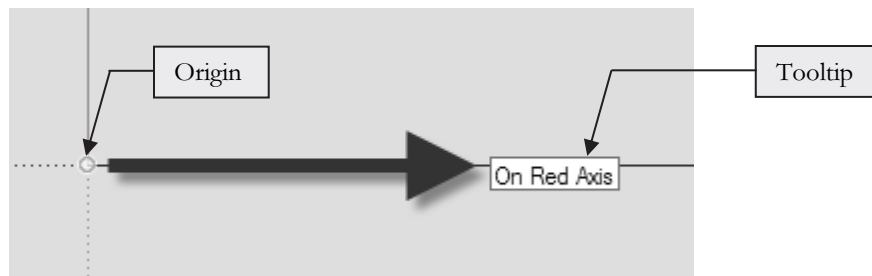


FIGURE 4-2.4 Sketching a line

14. Without moving the mouse, type **3'10.5** and then press **Enter**.

You always have to enter a foot symbol if feet are needed, however the inch symbol never needs to be typed as it is assumed when nothing is specified.

The *Line* tool will remain active until you pick the *Select* icon or another tool. Next, you will draw one of the vertical lines.

15. While the **Line** tool is still active (Figure 4-2.5):
 - a. Start moving your cursor straight up.
 - b. Ensure the “**On Green Axis**” tooltip is showing; meaning vertical.
 - c. Type **1'6** and then press **Enter**.

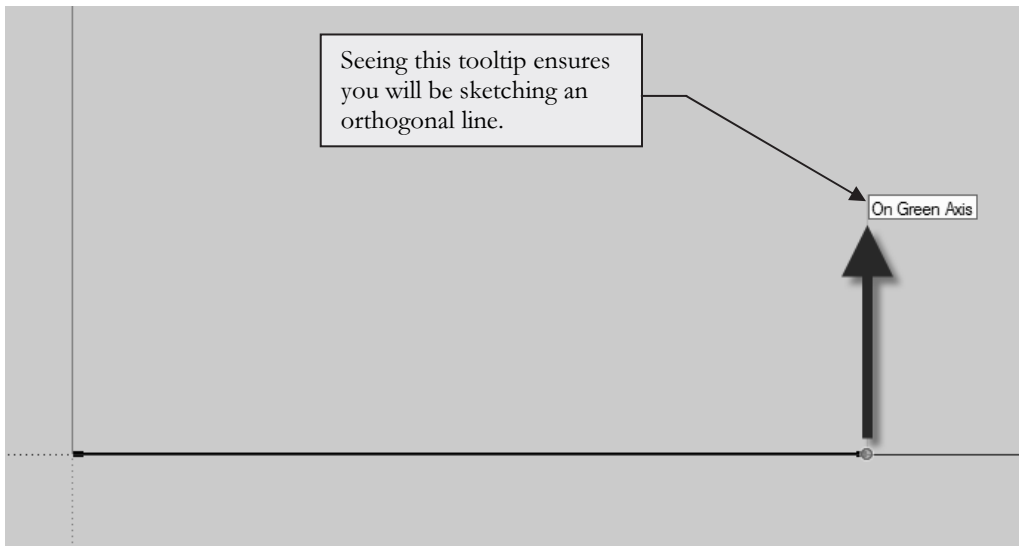


FIGURE 4-2.5 Sketching another line

16. Using one of the alternative methods of entering 3' 10 1/2" (see below), sketch the top horizontal line from right to left.

Entering fractions: the 3'-10½" can be entered several ways.

- 3' 10.5 *Notice there is a space between the feet and inches.*
- 3'10.5 *Notice space can also be left out.*
- 3' 10 1/2 *Note the two spaces separating feet, inches and fractions.*
- 3'10 1/2 *The second space is always required.*
- 0' 46.5 *This is all in inches; that is, 3'-10½" = 46.5".*
- 46.5 *SketchUp assumes inches if nothing is specified.*

TIP: Even when you are in a model using imperial units you can type a metric value and SketchUp will automatically convert it. For example; typing **150mm** draws a **57/8"** line.

To draw the last line you could type in the value but you can more quickly snap to the endpoint of the first line drawn. This will complete the rectangular shape.

17. Snap to the **Endpoint** of the first line drawn (Figure 4-2.6).

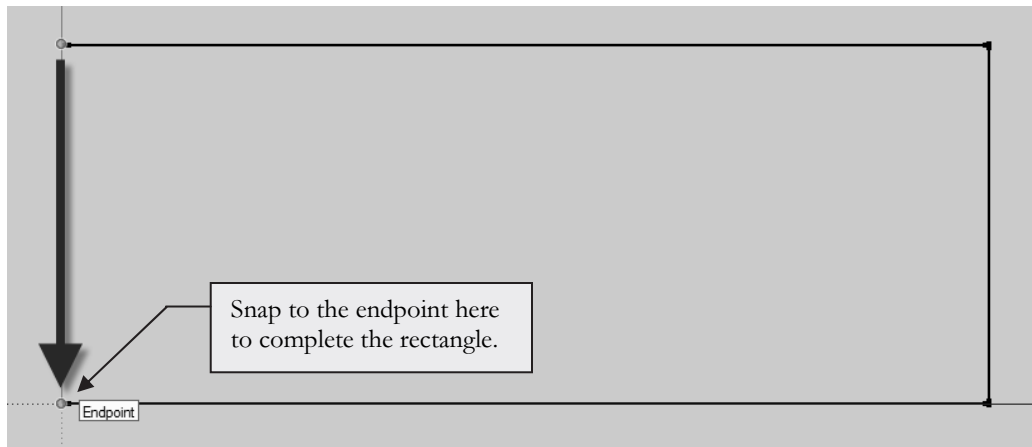
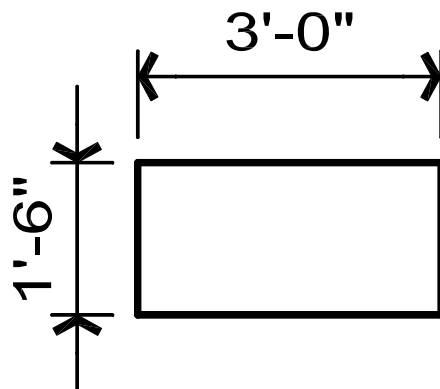


FIGURE 4-2.6 Completing the rectangle

Once a closed perimeter is defined, as your rectangle, a surface is automatically created. You should use the *Tape Measure* tool too occasionally to double check your lengths. Simply select the *Tape Measure* tool from the toolbar and then pick two points in the model. The *Value Control Box* in the lower right of the application lists the measurement. Give it a try!

18. Save your file as **Coffee Table.skp**.

file name: **SmallDesk**



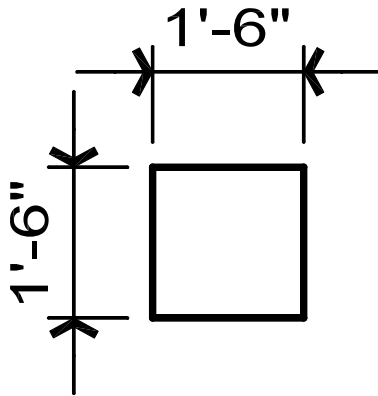
Next, you will start another new file and create this rectangular shape.

19. Create a new model (per steps 1-4) and create this small desk using either of the methods just covered.

20. Save your file as **Small Desk.skp**.

Don't worry, things will get more challenging. These steps are laying the groundwork for all of the 3D modeling you will be doing! So make sure you take the time to understand this material.

file name: **Night Table**



Obviously, you could draw this quickly per the previous examples. However, you will take a look at copying a *file* and then modifying an existing model.

You will use the *Move* tool to stretch the 3'-0" wide desk down to a 1'-6" wide night table.

21. With the *Small Desk* SketchUp model still open, select **File** → **Save As**.

22. Type **Night Table** for the *File name* and click **Save** (Figure 4-2.7).

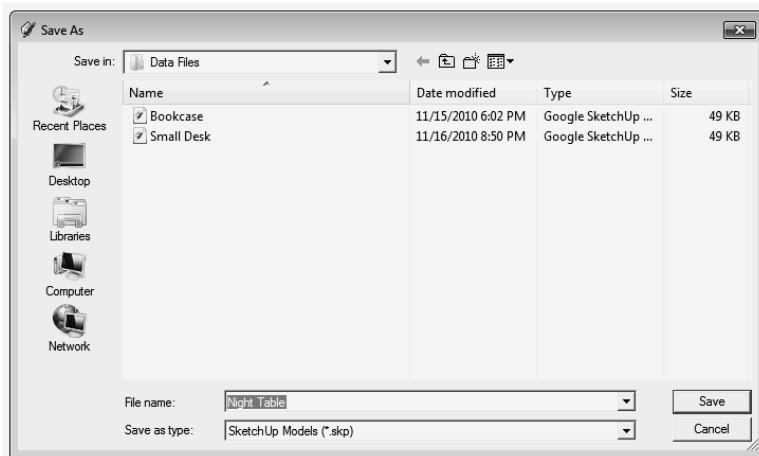


FIGURE 4-2.7 SaveAs dialog

You are now in a new file named *Night Table.skp*, and are ready to manipulate the file. The original “small desk” file is now closed and will not be affected.

You will use the *Move* tool to change the location of one of the vertical lines, which will cause the two horizontal lines to stretch with it. SketchUp’s *Lines* automatically have a parametric relationship to adjacent lines when their endpoints touch each other.

23. Click the vertical line on the right and then select the **Move** icon on the toolbar.

24. Pick the mid-point of the selected line; move the mouse towards the left – while locked to the vertical (Figure 4-2.8). **Do not click yet.**

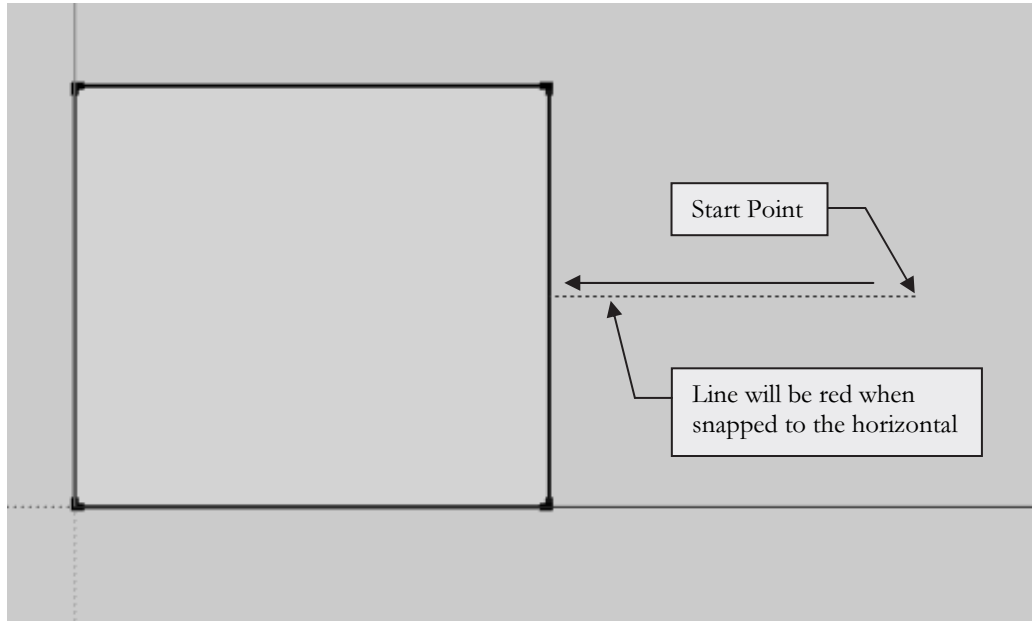


FIGURE 4-2.8 Moving an edge

25. While in the *Move* command and snapped to the horizontal (i.e., red axis), type **1'6** and then press **Enter**.

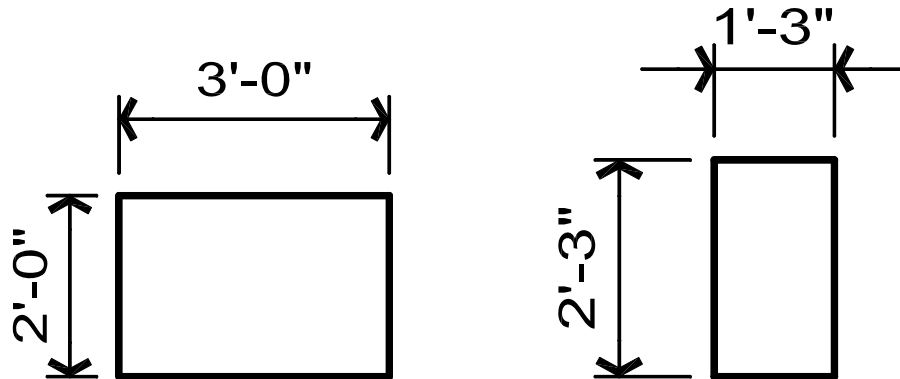
That is it! You essentially just stretched the rectangle. The two horizontal lines automatically shrunk in length and the surface resized itself as well.

26. **Save** your *Night Table.skp* file.

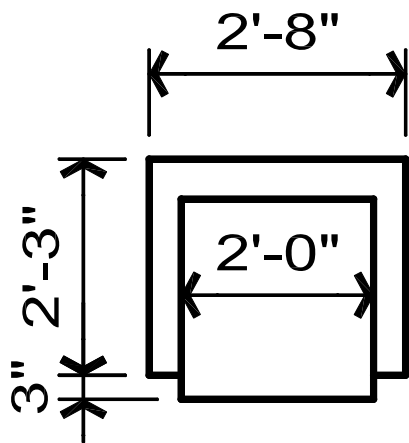
file name: **Small Dresser**

file name: **File Cabinet**

27. Draw the *Small Dresser* and the *File Cabinet* per the previous instructions.



file name: Square Chair



Next you will draw this squarish-styled chair. You could draw this by setting up a few *Guides*, but another method will be shown. It is good to know several ways to accomplish the same thing as one solution may be more efficient than another in certain situations.

28. Start with a 2'x2' square aligned with the origin.
29. Draw the backrest and armrests as separate rectangles, near the square (Figure 4-2.9).

30. Use the **Move** tool to move the rectangles into position (Figure 4-2.10).

SketchUp does not allow overlapping lines. Therefore, when you moved the rectangles into place the lines were merged, with any endpoints remaining. This will allow you to delete the extra lines identified in Figure 4-2.10.

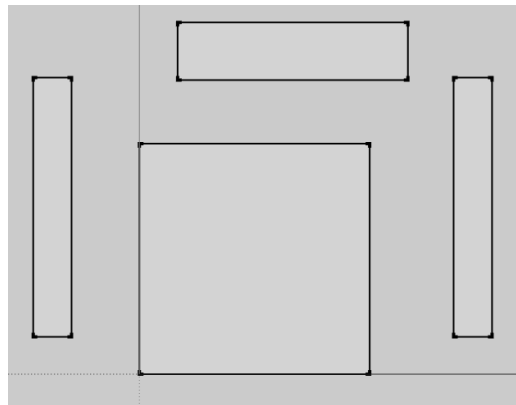


FIGURE 4-2.9 Creating the chair

31. **Delete** the extra lines identified in Figure 4-2.10; select it and press the **Delete** key on the keyboard.

When the two lines were removed, the surfaces automatically joined together within the newly enclosed area (Figure 4-2.11). At this point you only have two surfaces defined, each of which can be extruded into separate shapes, which you will do later in the next chapter.

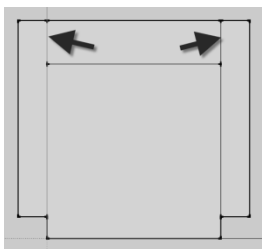


FIGURE 4-2.10

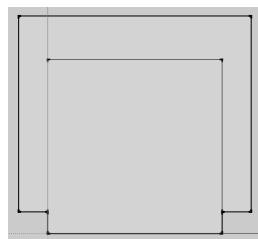


FIGURE 4-2.11

You have completed the square chair – for now anyway. You can save your file and move on to the next one.

Selecting Objects:

At this time we will digress and take a quick look at the various techniques for selecting entities in SketchUp. Most tools work the same when it comes to selecting elements.

When selecting entities, you have two primary ways to select them:

- Individually select entities one at a time
- Select several entities at a time with a window

You can use one or a combination of both methods to select elements when using the *Select* tool.

Individual Selections:

When using the *Select* tool, for example, you simply move the cursor over the element and click; holding the **Ctrl** key you can select multiple objects. Then you typically click the tool you wish to use on the selected items. Press **Shift** and click on an item to subtract it from the current selection set.

Window Selections:

Similarly, you can pick a *window* around several elements to select them all at once. To select a *window*, rather than selecting an individual element as previously described, you select one corner of the *window* you wish to define. That is, you pick a point in “space” and hold the mouse button down. Now, as you move the mouse you will see a rectangle on the screen that represents the windowed area you are selecting; when the *window* encompasses the elements you wish to select, release the mouse.

You actually have two types of windows you can use to select. One is called a **window** and the other is called a **crossing window**.

Window:

This option allows you to select only the objects that are completely within the *window*. Any lines that extend out of the *window* are not selected.

Crossing Window:

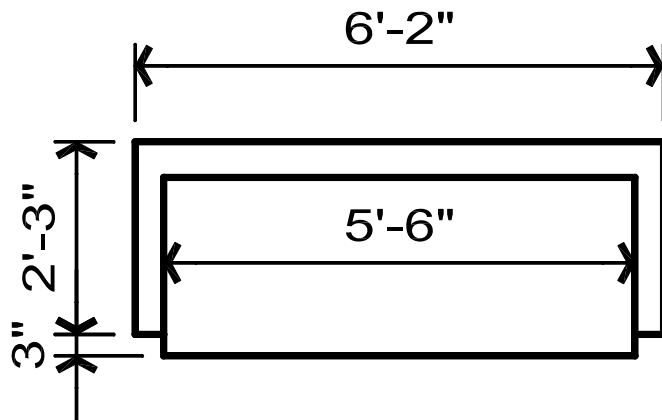
This option allows you to select all the entities that are completely within the *window* and any that extend outside the *window*.

Using Window versus Crossing Window:

To select a *window* you simply pick and drag from *left to right* to form a rectangle.

Conversely, to select a *crossing window*, you pick and drag from *right to left* to define the two diagonal points of the window.

file name: Square Sofa



Similar to how you turned the *Small Desk* into a *Night Table*, you will turn the *Square Chair* into this *Square Sofa*.

This is a little trickier as you have to select multiple lines as part of the *Move* command.

32. Once you are sure you saved the *Square Chair* file, do a **Save As** to create the **Square Sofa** file.

33. Select the three lines shown in Figure 4-2.12.

Moving these lines will cause the horizontal lines to follow – or stretch, thus growing the chair into a sofa!

34. Select **Move** from the toolbar.

35. Pick anywhere in the drawing area and begin moving the cursor to the right, snapped to the horizontal (i.e., on red axis).

36. Without clicking a second point, type the desired length you wish to move the lines (i.e., stretch the sofa); this is the difference between the chair and the sofa.

Notice how the horizontal lines extended because they have a parametric relationship to them. The surface has also updated.

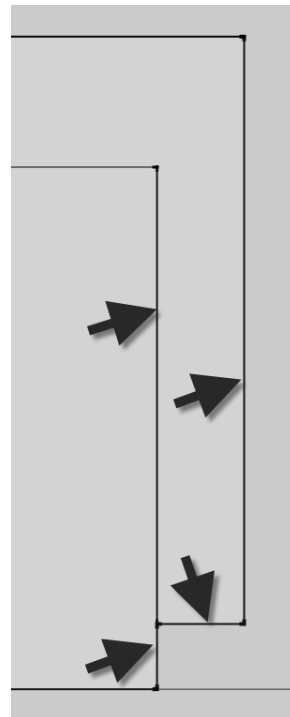
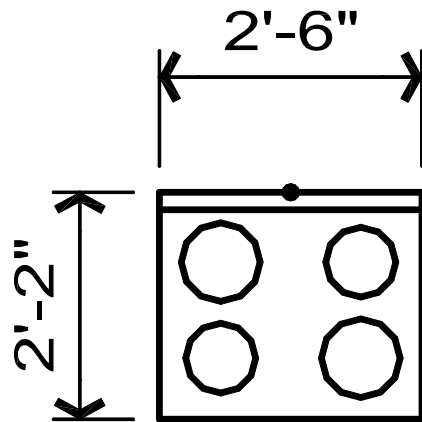


FIGURE 4-2.12

37. Use the **Tape Measure** tool to double check your dimensions. Make any corrections needed before moving on.

Keep in mind that all these basic steps will be directly applicable to the more advanced 3D modeling coming up.

file name: **Range**



Now you will draw a kitchen range with four circles that represent the burners.

In this exercise you will have to draw temporary lines, called *Guides*, to create reference points needed to accurately locate the circles. Once the circles have been drawn, the *Guides* can be erased.

38. In a new file, **Draw** the range with a 2" deep control panel at the back; refer to the steps previously covered if necessary.

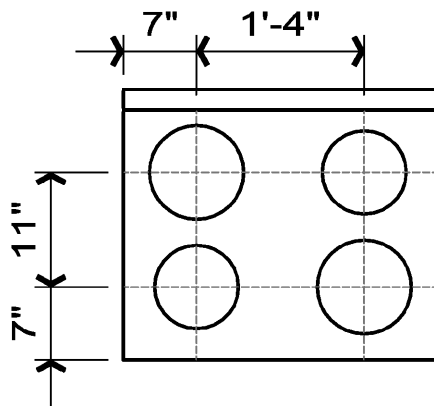


FIGURE 4-2.13 Adding guides

39. Use the **Tape Measure** tool to create the four guides dimensioned in Figure 4-2.13.

- a. With the **Tape Measure** tool active, click and drag on an edge to create a *Guide* parallel to that edge.
- b. Once you let go of the mouse the *Guide* is created.
- c. Once placed, type the distance and then press **Enter** to position it.

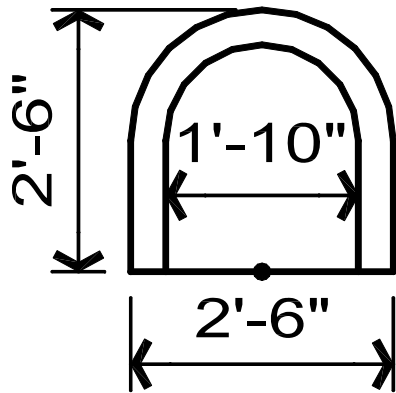
40. Using the **Circle** tool, draw two 9½" Dia. circles and two 7½" Dia. circles using the intersections of the *Guides* to locate the centers of the circles (Figure 4-2.13).



41. Using the **Select** tool, select each of the *Guides* and press the **Delete** key on the keyboard to remove them from the model.

Circles are made up of several straight line segments. The default number is 24. If your circle is large, the number of edges needs to be increased to maintain the look of a circle. This can be done just before clicking to locate the circle's center, by just typing a number and **Enter**. This number, the radius and *Layer* can all be changed at any time using the *Entity Info* dialog.

file name: **Rounded Chair**



These last two drawings will involve using the *Arc* tool. The *Offset* command will also be utilized.

You will start this chair drawing by sketching the perimeter and then offsetting it inward.

42. Draw the three orthogonal lines shown in Figure 4-2.14.

- a. The horizontal line is centered on the green axis and aligned with the red axis.
- b. The two vertical lines are 1'-3" long.



43. Use the **Arc** tool to add the rounded backrest.

- a. Pick the three points shown in Figure 4-2.14.
- b. When picking the third point, you should see a “half circle” tooltip before clicking the mouse button.

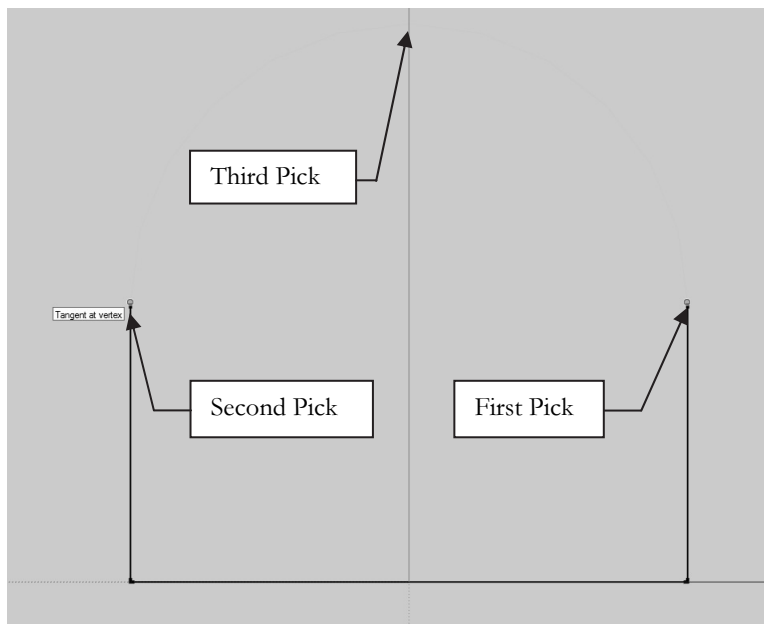


FIGURE 4-2.14 Sketching an arc

You now have the perimeter of the chair defined. Next, you will use the *Offset* command to create the backrest.

44. Select the **Offset** tool from the toolbar.



45. Select the arc – just the line, not the surface.

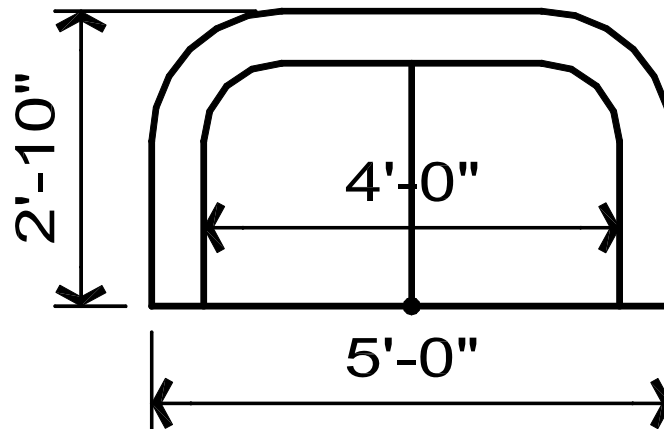
46. Start moving your cursor towards the middle of the chair and then type **4** and press **Enter**.

47. Draw the two remaining vertical lines.

- a. For the first end of the line, snap to the end of the arc.
- b. For the second end of the line, snap to a perpendicular point on the horizontal line.

The 2D version of the chair is now complete. The drawing now has two surfaces, because two enclosed areas exist: the backrest and the seat area. This will make the 3D extrusion process go smoothly.

file name: **Love Seat**



This last 2D exercise will round things off by making quarter round arcs.

One thing that is a little tricky here is setting up points to pick before sketching the arc. When adding an arc it is easiest if the two endpoints are defined by other line work. Thus, you just have to snap to endpoints for your first two picks and the third defines the radius. In this example you will need to add two temporary lines to define the start and end points of the arc.

The radius of the two arcs is 1'-3".

48. Add the orthogonal lines shown in Figure 4-2.15.

- a. All dimensions can be deduced from the information given.

The two 1'-3" lines are temporary and have only been added to aid in sketching the arcs (which is done in an upcoming step).

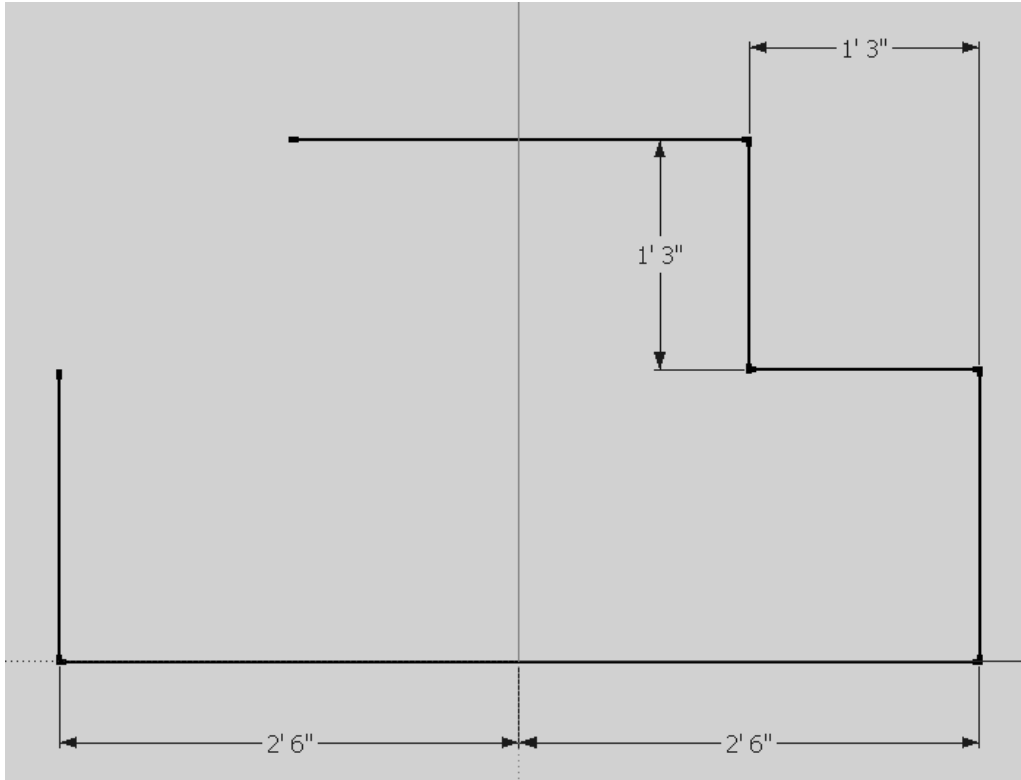


FIGURE 4-2.15 Setting things up for the arc tool

49. **Delete** the two 1'-3" lines.

50. Add one of the perimeter arcs.

- a. Pick the two endpoints provided by the orthogonal lines.
- b. Move the cursor until the arc is purple and the tooltip reads "Tangent at vertex" – this will produce a quarter round arc.

51. Add the other arc per the previous step.

Notice that you did not have to specify the radius of the arc because of the preparation done.

52. Offset the two arcs inward 4" – similar to the rounded chair (Figure 4-2.16).

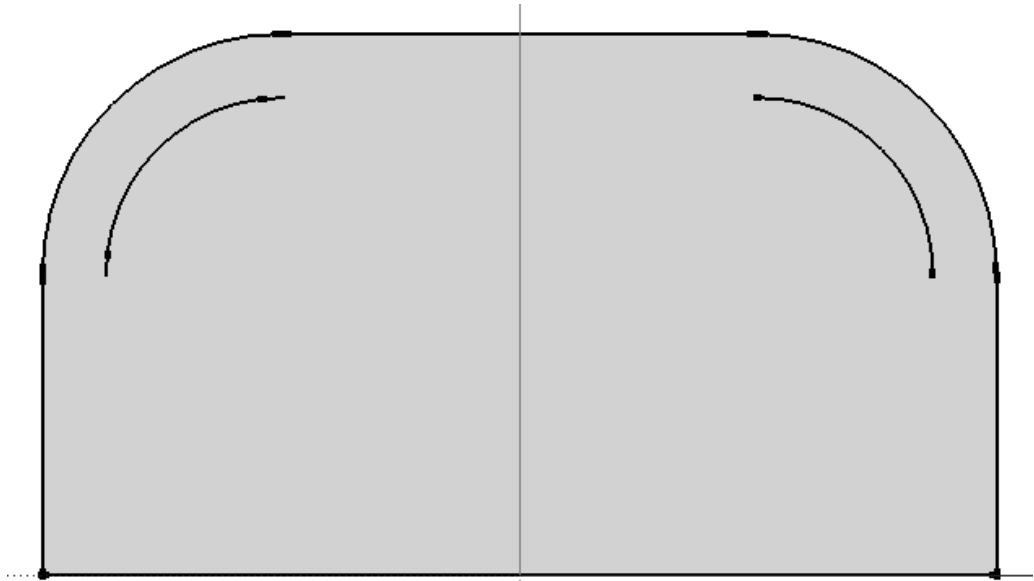


FIGURE 4-2.16 Offset perimeter arcs inward 4 inches

53. Complete the backrest and armrests by “connecting the dots” via the *Line* tool.
54. Draw the vertical line down the middle, representing the cushions.

Sometimes it is better to extrude a shape into the third dimension before adding extra line work such as the cushion line. This issue will be explored more in a later lesson.

Your drawing should be complete (Figure 4-2.17). Note that the drawing has three surfaces based on the line work you sketched. Don't forget to save your file.

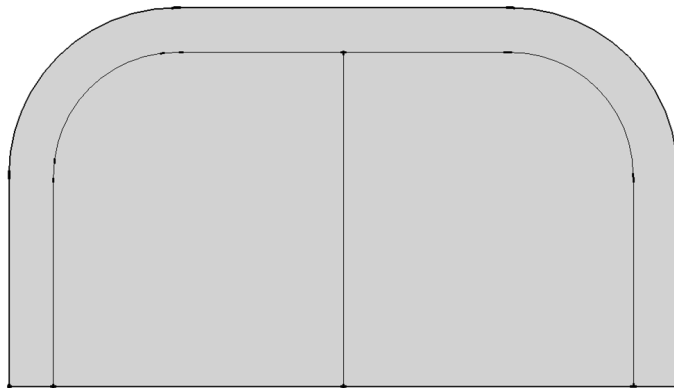


FIGURE 4-2.17 Completed 2D love seat

Self-Exam:

The following questions can be used as a way to check your knowledge of this lesson. The answers can be found at the bottom of this page.

1. Surfaces are created automatically. (T/F)
2. Edges cannot be deleted. (T/F)
3. Circles are made up of several straight lines. (T/F)
4. A _____ is similar to clipart in a word processing program.
5. One of the ways you can type 2'-6 1/4": _____.

Review Questions:

The following questions may be assigned by your instructor as a way to assess your knowledge of this section. Your instructor has the answers to the review questions.

1. Surfaces cannot be deleted. (T/F)
2. A model can only be viewed in perspective. (T/F)
3. If the “foot” or “inch” symbol is not entered, inches are assumed. (T/F)
4. For repetitive items it is better to use a *Group* rather than a *Component*. (T/F)
5. Text and dimensions can be a little tricky to maintain in a 3D model. (T/F)
6. _____ lists a length without actually drawing anything.
7. When reference lines are needed, you can add _____.
8. *Layers* help to control visibility of entities in your model. (T/F)
9. Used to see the properties of 3D text: _____.
10. Three picks are required to sketch an arc. (T/F)

Self-Exam Answers:

1 – T, 2 – F, 3 – T, 4 – Component, 5 – 2'6.25