

[www.sketchup.com](http://www.sketchup.com)

User Guide

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# Welcome to SketchUp 5

SketchUp for Microsoft Windows is a deceptively simple, extremely powerful tool for creating, viewing, and modifying 3D designs. SketchUp was designed to offer the elegance and spontaneity of hand drawing with the speed and flexibility of digital media.



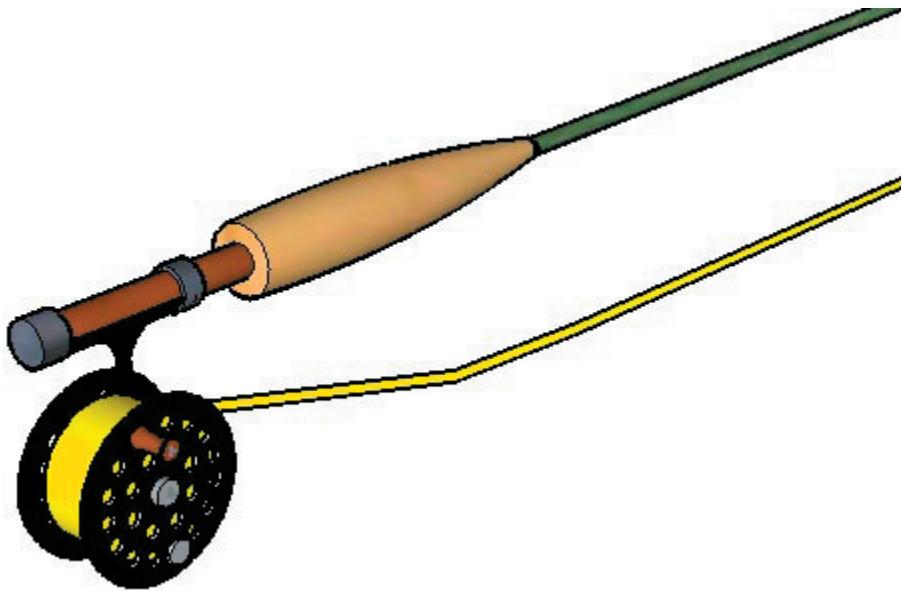
From the entire SketchUp team, thank you for trying SketchUp, and welcome to the SketchUp user community.

## Using this Guide

Familiarity with computer basics is all you need to use SketchUp. Continue with the Learning SketchUp section of this guide for assistance with learning SketchUp. Or, read the What's New in This Release section of this guide to become familiar with SketchUp's new features.

# Learning SketchUp

As with any software tool, there is minimum level of learning you must do to attain proficiency with SketchUp. The following information can help you learn how to use SketchUp.



## Video Tutorials

@Last Software has created several video tutorials to facilitate learning of the SketchUp tools and drawing procedures. View these tutorials through the View Tutorials menu item in the Help Menu (Help > View Tutorials > Video Tutorials).

These video tutorials will also play directly from the CD-ROM if you have a SketchUp CD-ROM inserted into your drive. If the CD-ROM is not in your drive, your computer will attempt to access streaming versions of the tutorials from the SketchUp web site. You can also download the online tutorials from the Web site to your hard drive to ensure they play smoothly.

## Online User's Guide

This online user's guide contains a Concepts section for users who are new to working in three dimensions or new to SketchUp. This section is meant to be read sequentially before using SketchUp for the first time.

The remainder of the user's guide consists of reference information for SketchUp users. Refer to this user's guide to find explanations of each SketchUp feature or to learn how to perform common tasks using the tools. The user's guide uses these special symbols and conventions:



**Note** - The Note icon is used to describe additional detail for a feature or aspect of SketchUp.



**Video** - Used to identify a Video Tutorial that facilitates the text by explaining how to use a feature of SketchUp.



**Caution** - The Caution icon is used to identify issues that might cause you problems.



**Tip** - The Tip icon is used to identify a tip to help you optimize the way you use SketchUp.

*Menu > Menu Item* - Used to identify a location for a specific menu item.

*italics* - Used to identify a term that is defined in the glossary.

*<a\_value>* - Used to identify a value you must supply.

## Context Sensitive Help

The Context Help button in the Standard Toolbar is used to access help on individual tools and features in SketchUp. Click on the Context Help button and then click on any tool button, dialog box, or menu command to open the online user's guide to the associated topic. Or, press the F1 key to display information about any open dialog box.

## Menus

Most SketchUp commands are accessible using both tool buttons and drop-down menus. Examine SketchUp's menus to become familiar with the breadth of features.

## Quick Reference Card

The SketchUp Quick Reference Card, available in the SketchUp Help Menu and on the @Last Software web site, contains a list of all of tools and their modifier keys. Examine the SketchUp Quick Reference card to become familiar with using SketchUp's tools.

## The Status Bar

The Status Bar, located at the bottom of the SketchUp Drawing Area, displays tips for the currently used drawing tools, including special functions accessible using keyboard shortcuts. Watch the status bar while you are working in SketchUp to discover advanced capabilities of each of the SketchUp tools.

## User Forum

The SketchUp Forums are a great way to contact others in the SketchUp user community. These forums provide unique environment for you to obtain help, suggest new features, offer advice, and share your models and materials. Visit <http://www.sketchup.com/forum> to begin interacting with other SketchUp users.

## SketchUp Training

@Last Software provides training courses for users who want extra assistance with SketchUp from the experts at @Last Software. Visit <http://www.sketchup.com/training> to see a list of training courses available in your area.



# What's New in this Release?

@Last Software listens to your requests and attempts to implement many into SketchUp enhancements and new features. SketchUp 5 contains the following enhancements and new features:

- **Sandbox Tools**

This release of SketchUp contains new sandbox tools allowing you to easily create and sculpt terrain and/or any geometry created from a *sandbox* or triangulated information network (TIN). Sandbox tools can not only be used to create terrain, but can be used to create organic geometry or geometry that appears as though built by hand, such as a wooden bowl or the face of a person. The sandbox tools are enabled through SketchUp's new Extension Manager.

- **Extension Manager**

The Extension Manager can be used to enable non-core SketchUp features, such as the Sandbox tools. This feature helps simplify the SketchUp user interface for those users who primarily use core SketchUp features.

- **Component and Group Locking**

Component and Group entities can now be locked preventing them from being moved or edited.

- **Edge Rendering Styles**

This release of SketchUp contains Depth Cue and Endpoints edge rendering options. Depth cue allows you to emphasize foreground lines while Endpoints allows you to emphasize the endpoints of your model. Additionally, you can now turn on and off edges completely. Combined, these new edge rendering options provide several new rendering combinations for your models.

- **Component Outliner**

A new Component Outliner is used to view your Group and Component hierarchies as a hierarchical tree. This feature is great for navigating through large models, restructuring the model hierarchy, locating instances of a particular component, and renaming groups and components.

- **User Interface Enhancements**

Cursors and icons have been redesigned so that they are more easily recognizable.

- **New Import and Export File Types**

SketchUp 5.0 contains new 3D Studio (3DS) and digital elevation mode (DEM) importers and object (OBJ), XSI, and FBX exporters

- **Component Manipulation**

The Component Browser now has a Replace Selected context menu option allowing you to replace the selected components in the drawing area with the selected component in the Component Browser. Additionally, the Make Unique menu item converts the currently selected component into a unique component, with new definition.

*SketchUp*

- Walk Tool Enhancements

The Walk Tool now walks up inclines including ramps and stairs and terrain while attempting to maintain eye height. Additionally, the Walk Tool has advanced collision detection to ensure you do not walk through walls or other barriers.

- Dialog Box Behavior

Model settings and manager dialog boxes will snap to the edge of the screen and to the edges of the application window. These dialog boxes will also stick to each other (top or bottom) to create a dialog box stack. Each dialog box in the stack can be closed to only show its title bar, or opened. A new Show/Hide Windows menu item allows you to show or hide all dialog boxes with a single menu item or keyboard shortcut.

- Push/Pull: Create New Starting Faces

The Push/Pull tool, in conjunction with the CTRL key, can be used to create a connected series of volumes (each with their own set of edges). This feature is very useful for creating space planning diagrams for building interiors.

- Ruby Toolbars

New Toolbar and Command classes allow you to create toolbars for your tools in the Ruby programming language.

- Enhanced Documentation

The online SketchUp User's Guide now contains a concepts section, a glossary, and visual cues informing the reader when a companion video tutorial is available to supplement the documentation.

- Fractional Units

The Model Info dialog box now contains fractional units primarily used by wood workers in the United States.



# Technical Support

Click on a topic to the left to browse the SketchUp User's Guide. Or, type in a keyword above to search for topics.

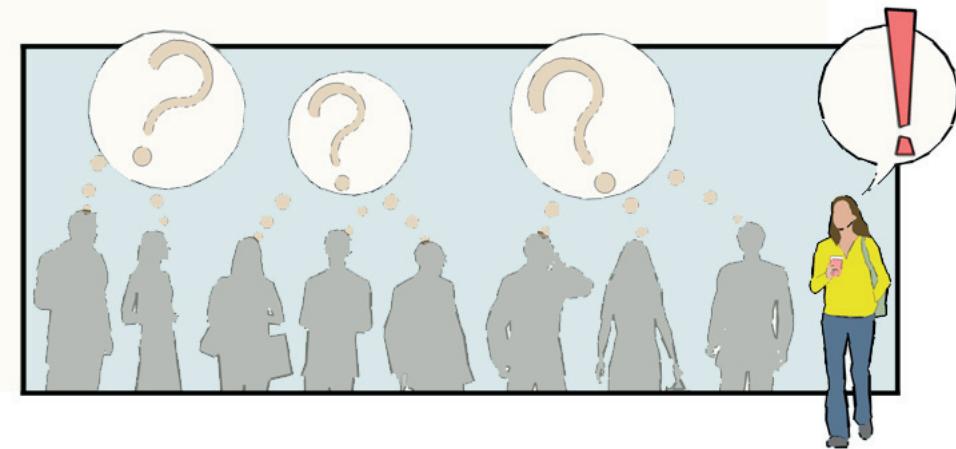
If you cannot find a topic in the user's guide, visit the support area at <http://www.sketchup.com/support> or send an email directly to @Last Software at support@sketchup.com. Please include the following with your email:

- A copy of the SketchUp file you are creating
- A list of your computer system's resources (CPU type, RAM, video card details, and the exact release of your operating system, such as Windows XP Professional with Service Pack 2)
- A detailed description of your question or issue

@Last Software provides free unlimited email technical support to SketchUp end users.

# Concepts

This section of the user's guide covers the numerous 3D design and SketchUp concepts used throughout the SketchUp community. This section was written primarily for users who are new to SketchUp.

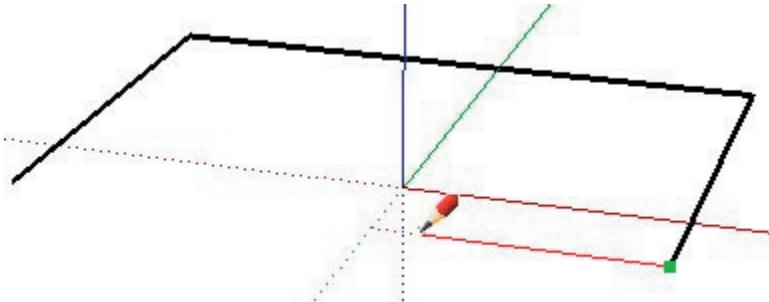


This section is organized into the following topics:

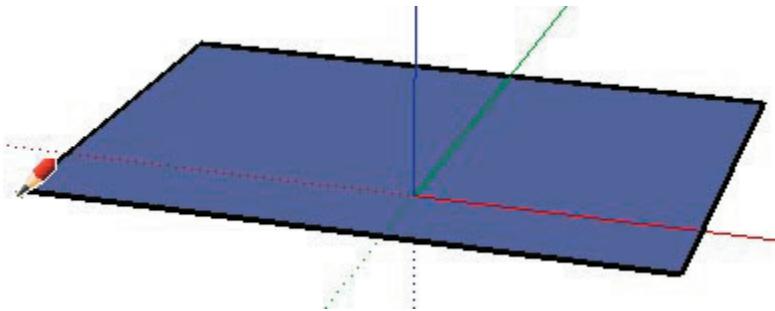
- Designing in Sketchup - Introduces you to the basic principles behind drawing accurately in SketchUp.
- Drawing Quickly - Introduces you to SketchUp tools and concepts necessary to draw quickly.
- Viewing Models in 3D - Presents the concept of the camera and manipulation of a model in 3D space.
- Adding Realism to Your Models - Introduces you to mechanisms to quickly add realism to your SketchUp models.
- Presenting Your Models - Presents concepts related to the presentation of your models to clients.
- Introduction to Terrain Modeling - Presents concepts related to working with mesh tools to create both terrain and organic shapes.

# Designing in SketchUp

SketchUp *models* are fundamentally created by joining lines as the edges of the model. Faces are automatically created when any three or more lines or edges are in the same *plane* (an infinite flat 2D space), or *coplanar*, and they form a closed loop. These edge/face combinations are combined to create 3D models. The following image shows three unconnected coplanar lines. These lines were drawn with the Line Tool (this tool looks like a pencil).

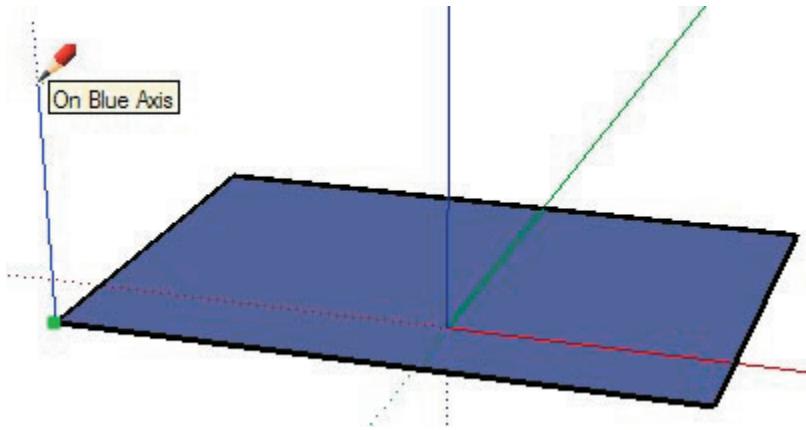


The following image shows four connected coplanar lines and the subsequently created flat, 2 dimensional, face.

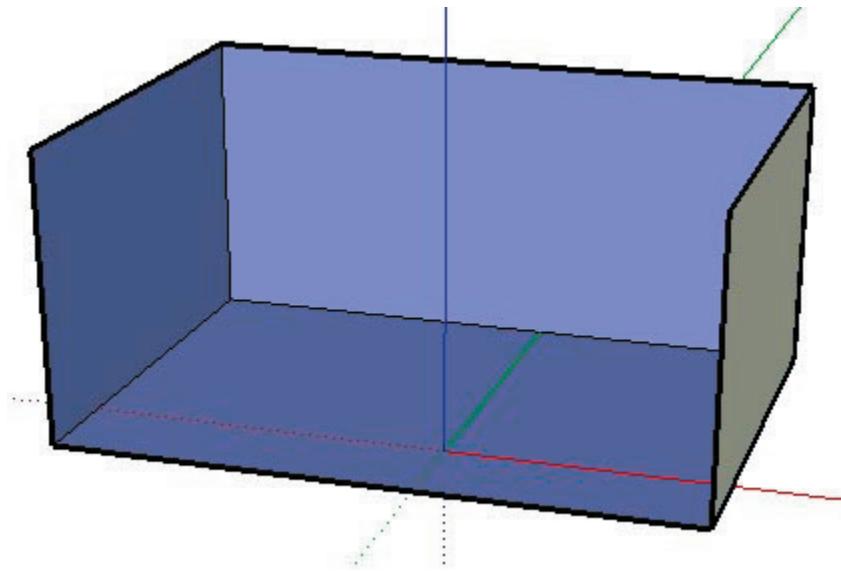


*Note - Everything you draw in SketchUp is generically referred to as geometry.*

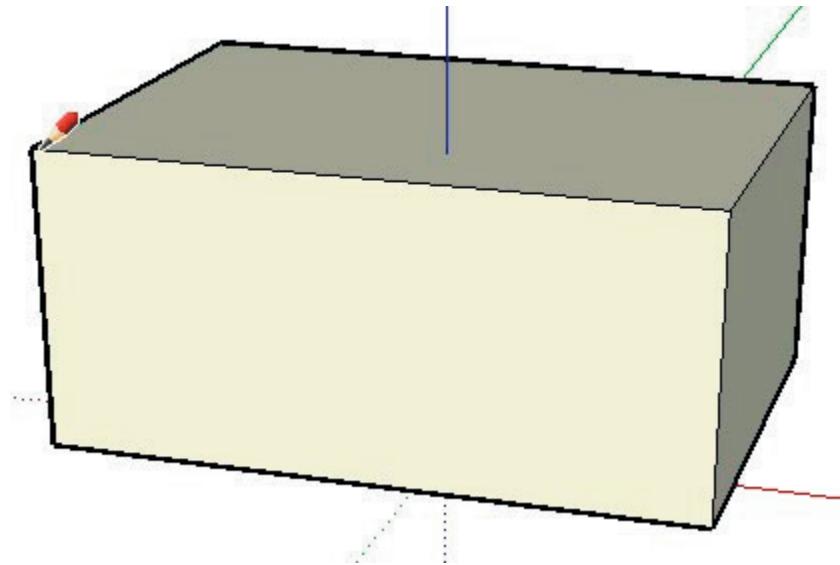
To create a 3D image, simply draw up or down in the blue direction (parallel to the blue axes). The coordinate system (axes) is covered later in this section. The following image shows the first line created in 3D space.



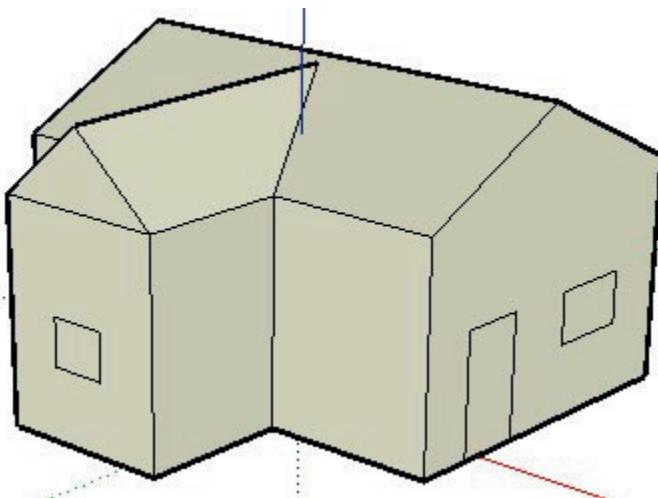
As you continue to draw lines, following the axes, faces are created. The following image shows three faces created simply by drawing lines parallel to the three axes directions (red, green, and blue).



There is one line left to draw to finish a 3 dimensional box. Notice that when this one line is drawn in, two faces are created.



You can do a lot in SketchUp simply by drawing lines to form faces using the Line Tool. And, you can draw lines starting anywhere (on another line, on a face, at a point, and so on). Can you recognize the previous 3D box within the image of the following house?



Look around the room you are in. Notice how everything you look at has faces. Some faces may be rounded, some might be flat. Additionally, everything has edges that bound the face, such as the edge of a shelf in a bookshelf.



*Note - SketchUp is not the same as Computer Assisted Design or CAD. CAD applications are designed specifically for representing concrete information, while SketchUp is for exploration and design of concepts and ideas (though you are not prohibited from designing models that are as concrete or accurate as those designed in CAD). SketchUp files can be imported into several different CAD applications for further processing and several different CAD files can be imported into SketchUp to quickly create 3D models.*

## Introduction to Entities

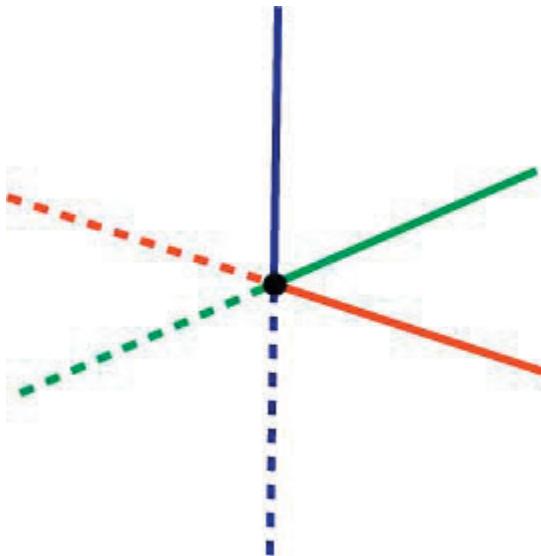
As mentioned previously, lines are combined to create faces in SketchUp. Edges and faces, are just two of the many building blocks (called *entities*) used for creating models in SketchUp. A full list of SketchUp entities follows.

Name	Notes
Line	Lines in SketchUp are straight. Lines, also referred to as edges, are the most basic building block for all SketchUp models.
Face	Faces are created automatically when three or more coplanar edges form a closed loop. Faces have a front side and a back side. SketchUp attempts to put the front side of all faces on the outside (facing out) of all of your models, though sometime you might have to tell SketchUp the direction for your faces.
Circle	Circle, Arcs, and Curves are comprised of several small lines or edges.
Arc	
Polygon	
Curve	
Freehand Sketch	
Group	<i>Group entities</i> are used to combine two or more entities in your model for quick operations such as a copy.
Component	<i>Component entities</i> are like groups but can be saved and reused in all of your SketchUp models. Components are just SketchUp models used within other SketchUp models.
Construction Line	A <i>Construction Line entity</i> is a temporary line used as a drawing guide.
Dimension	A notation indicating length of an edge or a radius.
Surface	<i>Surface entities</i> are the result of combining a number of faces to give the impression of smoothness.
Section Plane	
Image	An imported raster, or pixel-based, image.
Text	Text can be unattached (floating) or attached to a specific entity using a leader line.

## The SketchUp Coordinate Systems

SketchUp uses a 3D coordinate system whereby points in space are identified by position along three drawing axes: plus or minus X, Y, and Z (above ground/below ground) values. In SketchUp, plus or minus X is represented by solid green and dotted green lines respectively; plus or minus Y are represented by a solid red and dotted red axis lines respectively; plus or minus Z (above and below the ground plane) are represented by solid blue and dotted blue lines respectively. The plane where the red and green lie is called the *ground plane*. Finally, the term *origin*, is used to define the place where all of axes lines start or originate.

The following image shows the drawing axes in SketchUp (the lines have been thickened to make the axes easier to read). The black circle represents the origin.

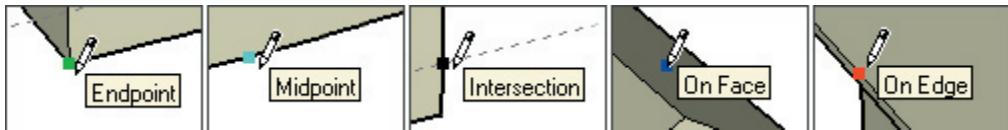


Understanding SketchUp's coordinate system is important because SketchUp's inference engine (explained next) provides you help with drawing accurately.

## Following the Inference Engine

SketchUp has an invisible inference engine to help you draw accurate and realistic models. The inference engine locates or infers points from other points in your model, such as the center of a circle, the midpoint of a line, a line that is perpendicular to the ground plane, a point on a face, a point on an edge, and so on.

SketchUp notifies you of these points by using both color indicators and tool tips, which are on-screen messages indicating the location of the cursor as you draw an entity. For example, SketchUp displays the string "On Face" when the cursor is touching a face. The following image contains five common inference tool tips.



Additional information on the inference engine, including inference types, is in the User Interface section of this guide.

The first step to drawing in SketchUp is to learn how to draw accurately by following the cues of the inference engine. Simply select the Line Tool (it looks like a pencil in the toolbar) and start drawing. Pay attention to the on-screen tool tips from the inference engine as you draw. Most everything you will create in SketchUp can be created by inference using the Line Tool.

You can move on to learning how to draw quickly after you master using the inference engine.

# Drawing Quickly

As mentioned, everything you create in SketchUp can be created by inference using the Line Tool. It is recommended that you use the Line Tool and inference engine to create your initial models.

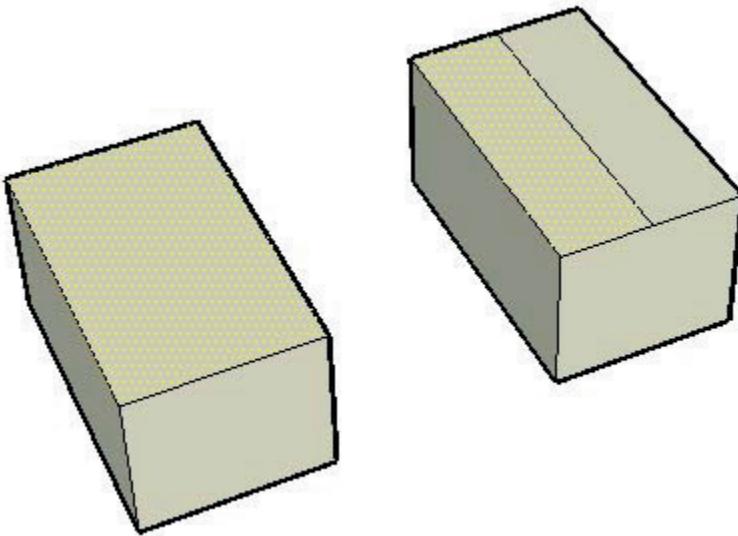


**Tip - Learn to draw accurately before you learn to draw quickly and you will master SketchUp in a shorter amount of time.**

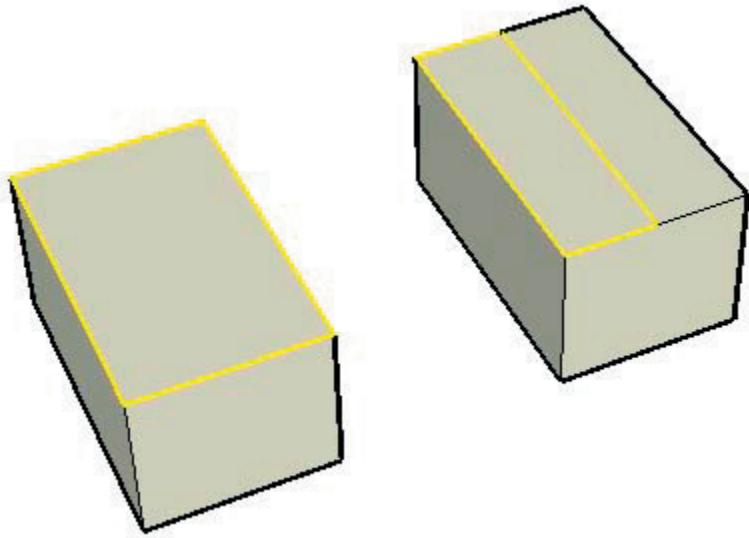
Most entities in SketchUp, including arcs and circles, are actually just a combination of small lines or edges. Thankfully, you do not have to use only the Line Tool to sketch your models. Instead, SketchUp provides many other tools that can be thought of as mechanisms to draw quickly. Some of these tools are for creating common entities such as a circle, arc, polygon, or freehand line. Other tools allow you to quickly draw complex models by modifying your models (splitting, skewing, and even merging geometry together).

## Dividing and Healing Geometry

Some tools speed up design dramatically by allowing you to perform modifications on existing geometry. For example, the Line Tool allows you to *split* faces and edges to create additional independent faces and edges. Simply divide the face or edge with another edge. This concept is similar to cutting a piece of wood in half and having two separate remaining pieces. The following image shows how two faces are created when dividing the face with a line. Notice the image on the left has one solid top face, while the image on the right has two independent faces when split with a line. Note that the dividing line does not cut through the model from top to bottom, but just cuts the top face in half.



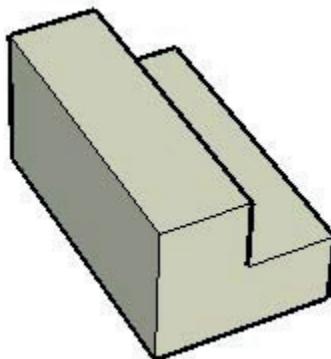
The following image shows how the edges at each end of the face have been split to create additional edges. Note four edges bounding the face in the left image, while four edges bounding each of two faces in the image on the right (the middle edge being common to both top faces).



If you remove the line that shared by the two faces in the image on the right, the two faces will be glued back together or *healed* back into one face. It is common in SketchUp to perform these dividing and healing operations to your model.

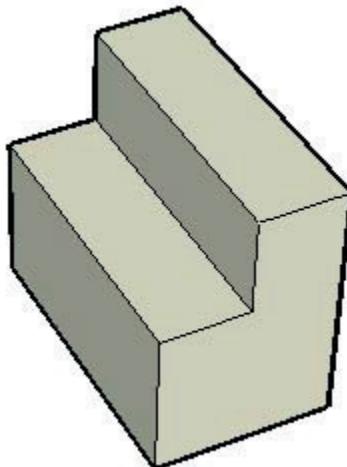
## Pushing and Pulling

The Push/Pull Tool allows you to take any face in SketchUp and push it away or pull it toward a starting point (these operations are also referred to as *extrusions*). Pushing is the process of reshaping a portion of your model by shrinking the portion away from its starting point and along a single axis. The following image shows the right most face of the split face image (shown earlier) having been pushed down toward the ground plane.



In the previous image, there are eight faces that can be pushed, can you find them (some are hidden out of view)?

Pulling is the process of expanding a portion of your model away from the starting point along a single axis. The following image shows the same face having been pulled up toward the sky. Any face in SketchUp can be pushed and pulled (with some minor limitations).

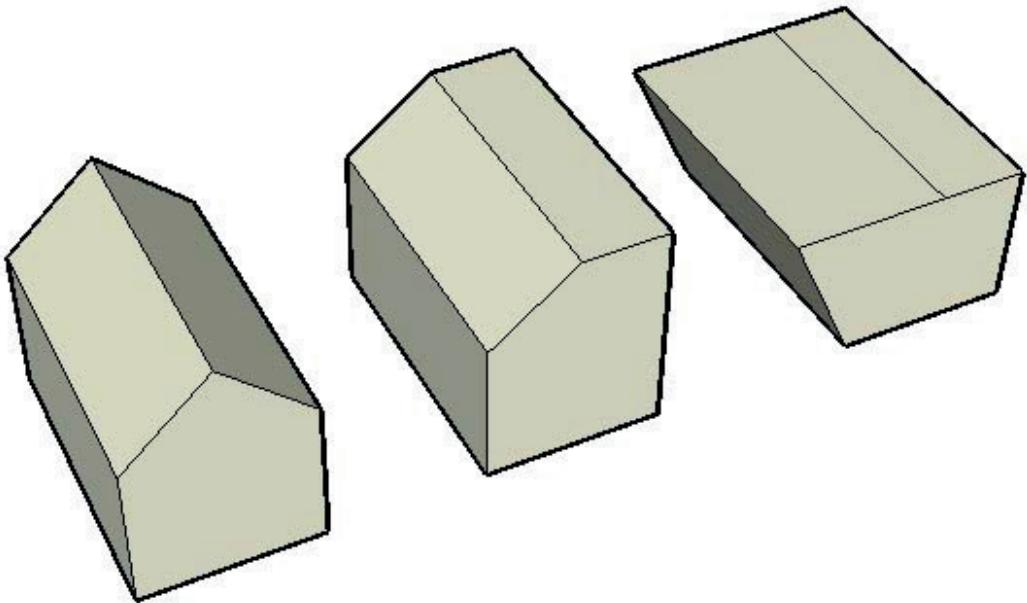


In both of the images, the left side of the cube remained the same size, while the right side was pushed and pulled (shrunk and expanded) independently.

SketchUp®

## Manipulating Connected Faces

Models in SketchUp can be manipulated (skewed, distorted, or folded) by selecting specific points on an edge or face and moving the points with the Move Tool.



The previous image shows three cubes that had their top face cut in half by an intersecting line and then were skewed by moving a single edge or face. From left to right:

- The intersecting line was moved up with a Move Tool. The faces that were connected to the middle line followed the line as it was moved to create a roof-like effect.
- In the middle image, the right-most top face was moved up in the blue direction, causing the left-most top face to angle to create a slant on one side.
- Finally, in the right-most image, the left most top edge was moved to the left to skew the model into a trapazoid-like shape.

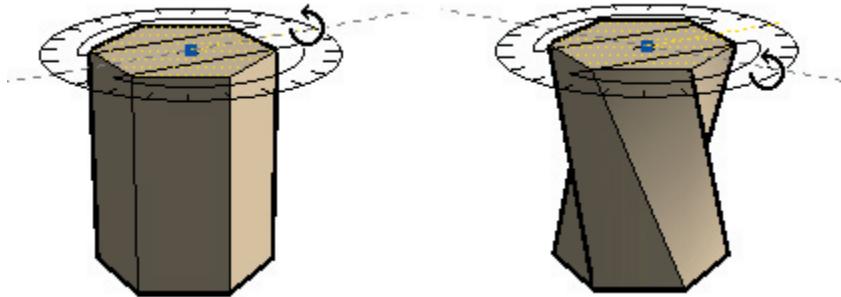
Be sure to use the Move Tool to grab and move edges and faces while you begin to experiment with modeling in SketchUp.



**Tip** - Experimentation and play is highly important in learning how to draw in SketchUp! Learn how to draw accurately first, then learn how to draw quickly using the concepts in this portion of the user's guide.

## Autofold

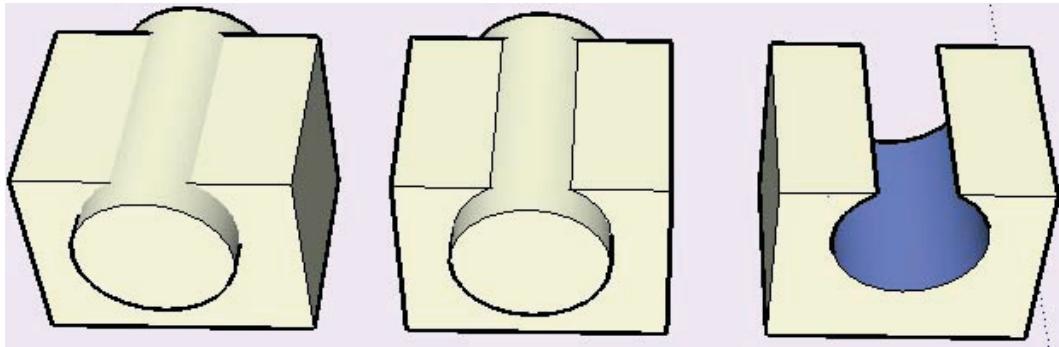
Faces must remain planar at all times in SketchUp. Therefore, SketchUp will Autofold, or *score*, faces as necessary to accommodate any operation that will result in warped faces.



In the images above a six-sided polygon (created with the Polygon Tool) was pulled to create a 3 dimensional shape. Then, the Rotate Tool was used to rotate only the top face of the shape. Thus, because of SketchUp's sticky nature, the sides of the shape, which share common edges with the top face, twisted and folded with the rotate operation.

## Intersections (Boolean Operations)

SketchUp allows you to easily create complex geometry by intersecting two geometrical forms, such as a box and a tube, creating new edges where the elements intersect, merging the geometry, and subtracting parts of the combined geometry.



In the left-most image above, a cylinder shape was moved into a cube shape. Notice, that no lines appear where the two shapes intersect, indicating that the shapes have not truly merged. A special tool, called the Intersect with Model Tool, is used to create intersection lines (the middle image), merging the two shapes together to form one new shape. Finally, the cylinder shape is deleted (using the eraser tool on the cylinder's edges) and because the intersection lines were created, the resulting curved face is left within the cube creating a new complex geometrical shape (right image).

## Tools

Tools in SketchUp are divided into five categories: *principal tools* (tools that are used a lot to select and modify geometry), *drawing tools* (tools used to create geometry), *modification tools* (tools used to modify existing geometry), *construction tools* (tools used to create construction lines or points, and document your model), *camera tools* (tools used to view geometry; discussed later), and *walkthrough tools* (tools to explore your model). The following table lists all of the Drawing and Modification tools:

Tool	Type	Notes
Line	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Arc	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Freehand	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Rectangle	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Circle	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Polygon	Drawing	Can be used to create, intersect, or divide a face or other line (edge)
Select	Principal	Used to select entities
Eraser	Principal	Used to delete geometry and heal faces
Paint Bucket	Principal	Used to apply <i>materials</i> or <i>textures</i>
Position Texture*	Modification	Used to manipulate materials painted on a surface.
Move	Modification	Used to move, distort (autofold), and copy entities
Rotate	Modification	
Scale	Modification	
Push/Pull	Modification	Used to expand and shrink geometry
Follow Me	Modification	Used to duplicate a profile along a path
Intersect With Model*	Modification	Used to create complex geometry by piecing together separate geometry
Offset	Modification	
Tape Measure	Construction	Used to get a measurement and create construction line entities
Protractor	Construction	Used to set angles
Axes	Construction	
Dimensions	Construction	Used to calculate and display a dimension
Text	Construction	
Section Plane	Construction	Used to create section cut effects

\* These menu items available from menus (not available on the toolbar, but are discussed in the tools sections of this user's guide).

Each of these tools is discussed further in this guide. As a recommendation, however, learn the Line Tool and Eraser Tool first, followed by the Orbit Tool (discussed in the next section), Push/Pull, and Move Tools.



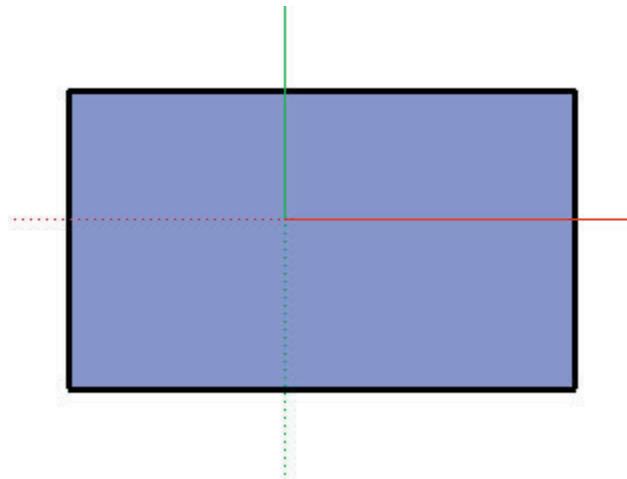
**Note** - Most drawing tools can perform modifications as well, such as the Line Tool being used to draw a line to divide a face.

# Viewing Models in 3 Dimensions

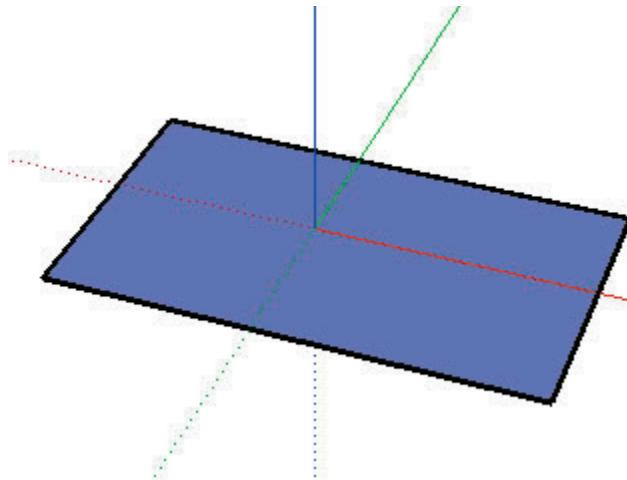
SketchUp implements the concept of a *camera* to represent your *point of view* (POV) of the model. Simply, you (the user) are treated as though you were a camera looking at your model as you work. This concept is particularly important when your model is something that you want to tour, such as a house, as though you were walking through it in the real world. In this case, SketchUp allows you to change your point of view to a specific height and angle to the model and walk through the model as though it were real.

## Manipulating the Camera

SketchUp starts at a camera angle (the angle at which you view your model) that is facing directly down at the ground as though you were looking directly down the blue axis from the sky at the ground plane. This camera angle is the default because the majority of SketchUp models, houses and landscapes for example, begin on the ground or red/green plane. Notice you only see the ground plane in the following figure. This figure suggests you are looking down at the image as though looking down the Blue axes (you, the camera, are hanging in the sky and looking right down at the ground).



However, you will not get the sense of modeling in a 3D space until you *orbit* the camera using the Orbit Tool after opening SketchUp and drawing a starting form. Following is the same model after having been rotated.



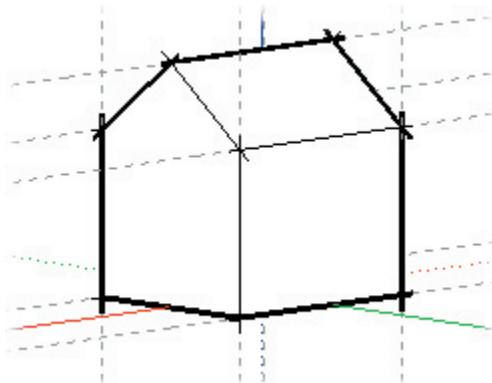
## Camera Tools

There are other ways to manipulate the camera (your point of view) in SketchUp, including zooming in and out and panning. The following table lists all of the camera tools in SketchUp:

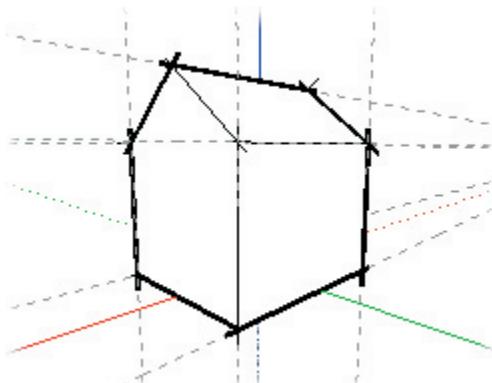
Tool	Type
Orbit	Camera
Pan	Camera
Zoom	Camera
Zoom Window	Camera
Zoom Extents	Camera
Position Camera	Walkthrough
Walk	Walkthrough
Look Around	Walkthrough

## Perspective and Paraline Projections

In addition to your point of view, you can view items in SketchUp in a paraline (also called *axonometric*) projection or in a *perspective* projection. Axonometric means “measurable from the axes” and an axonometric projection is a view of a model in which lines appear parallel in both 3 dimensional and 2 dimensional space. Also, lines have a length that is to some scale in an axonometric projection when printed (for example 4' in SketchUp = 1" on paper). The following image shows a axonometric or paraline projection.



Perspective, however, distorts the view such that it represents the model as though the lines were vanishing to a horizon (certain items appear closer while other items appear to be far away; entities are not to scale). The following image shows a perspective projection.



SketchUp defaults to a perspective view, but can be toggled into a paraline view for those who prefer a point of view that is similar to what you would see when drawing 3D objects in a 2D space (such as when drawing a cube on flat paper).



**Note** - The iso camera view in SketchUp is not the same as an isometric projection in which anything along the x, y or z axis are in proportion. This view simply shows you  $\frac{3}{4}$  of your model and is a standard camera angle for viewing models in a 3D space.

## Layers

SketchUp *layers* are used to control the visibility of geometry within large models. A SketchUp layer is an attribute with a name, such as "Layer0," "Layer9," or "Chairs." Elements can be assigned different layers. For example, all Component entities that are chairs can be assigned to the "Chairs" layer. This layer can be then be hidden temporarily to hide everything on the layer from view.



**Note** - Geometry on one layer is not separated from geometry on another layer. For example, faces sharing a common edge will both be affected by a modification to that edge regardless of what layer the faces are on.

### Default "Layer0"

There is one Layer, Layer0, in SketchUp, by default. All entities are automatically placed on this layer unless you create another layer, designate the new layer as the active layer, and create entities on the new layer.

# Adding Detail to Your Models

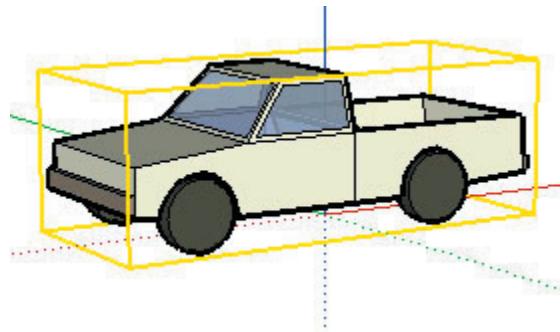
You will likely want to create progressively more detailed models as you become more proficient with SketchUp. SketchUp contains several features allowing you to quickly create detailed or life-like models. These features include components, materials, and shadows.

## Groups and Components

*Groups* and *components* are entities that can hold other entities. Groups are commonly used to combine several entities into a single entity for the purposes of performing a quick operation on the group (such as a copy or move). For example, you might draw a model, group the entities that make up the model and move the entire model. The characteristics of groups are:

- Quick selection. When you select a Group, all elements within that group are selected as well.
- Isolation of geometry. Entities within groups are protected from the rest of the model. Geometry outside of the group does not affect the geometry within the group
- Model organization. Groups can be nested within other groups resulting in hierarchical collection of subgroups.
- Group material. The group as a whole can be assigned a material of its own, which is separate from the materials painted on individual elements within the Group. See the Materials section within this topic for further information.
- Drawing Axes. Groups maintain their own internal drawing axes.
- Alignment and Hole Cutting. Groups have a special behavior that allows them to properly align themselves and stick to faces on which they are placed (as well as cut holes in those faces).

Anything you create can be used as a component. Components are just SketchUp models that have been saved and are into other SketchUp models. For example, you might create a model of a car that you want to bring into other SketchUp models, such as the model of a house.



Components have the following characteristics, in addition to the characteristics of groups:

- Instancing Behavior. Each copy of a component that you insert into a model is considered an *instance* of a *component definition*: the blueprint that defines the appearance and behavior of all component instances. Editing a component instance edits the definition and all instances accordingly. However, some actions, such as scaling an instance, only affects the instance itself.
- Improved Performance. Components allow SketchUp to more efficiently use your computer's resources because the information necessary to describe a component is only stored once, in the component definition, and then referenced for every component instance.
- Drawing Axes. Components display their own internal drawing axes when you are editing the component (group axes are invisible).
- Alignment and Hole Cutting. You can define alignment and hole cutting behavior when creating components (it is automatically defined for groups).



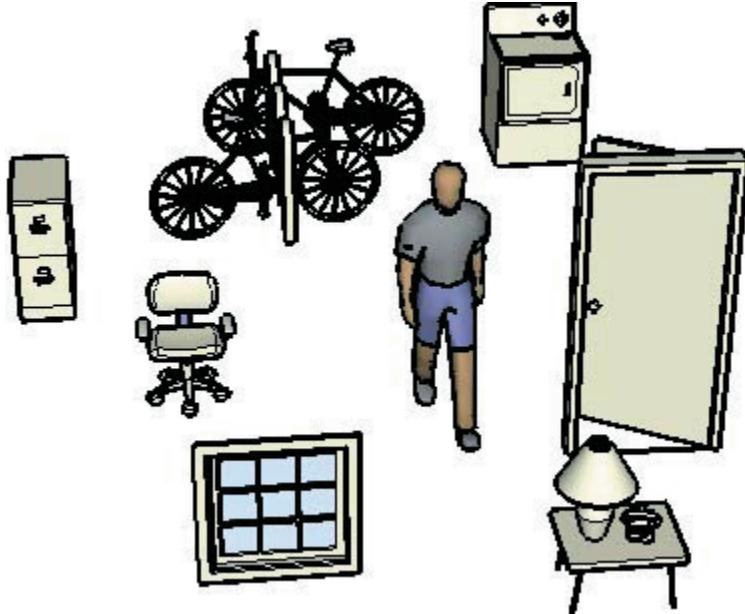
**Note** - You can define the characteristics of the component during component creation.

## Group and Component Context

Entities within a group or component are said to exist within the group or component's scope or *context*. You can modify a group or component outside of the group or component's context (affecting the entire group or component) and inside of the group or component's context (affecting the individual entities that make up the group or component). Additionally, you can embed components within other components, groups within components, components within groups and so on. So, it is always important to know what context you are in to ensure you are editing the correct entity.

## Component Libraries

*Component libraries* have been created and included with SketchUp to allow you to easily add detail to your models. These components range from standard architectural components (doors, windows, and so on) to people, cars, trees, and geometric shapes.



**Note** - Additional industry-specific content libraries are available on the SketchUp web site under the download section ([www.sketchup.com](http://www.sketchup.com)).

## Component Axes

Components have their own axes which, by default, are aligned to the global axes in the drawing area. These axes can be moved affecting component placement in the model.



**Note** - You can change the axes to affect the insertion point or you can move the insertion point independently of the axes.

## Cutting and Gluing Behavior

Components, such as doors and windows, can be designed such that they can be placed on specifically-oriented surfaces such as vertical surfaces. This behavior is referred to as the *gluing behavior* of a component.

Additionally, components can be designed to automatically cut holes in surfaces, such as in the case of a window component being able to cut a hole in a wall. This behavior is referred to as the *cutting behavior* of a component. It is important to note that the location of the red/green axes for the component defines the *cutting plane* or the infinite 2 dimensional surface where the component cuts into the particular surface. The cutting plane is an important concept when working with 3 dimensional cutting components because you will want the red/green plane of the component to be at the exact spot where the component should meet and cut through a surface.

## Layers, Groups and Components

Layers are a mechanism for controlling the visibility of entities within a model. Entities can be assigned to different layers in a model and those layers, and their contents, can be displayed independently. Groups and Components are used to isolate geometry as sub-models within an overall model, such as a component of a chair within a model of a room.



**Note** - Entities within a group or component can be on several different layers.

## Materials

In addition to components, SketchUp allows you to paint colors and *materials* on faces to add detail and realism to your models. Materials are essentially paints that have a *color* and optional *texture* (defined within a image file). For example, a siding material with the color of grey and a appearance or texture simulating to real overlapped composite siding. Following is a building painted with grey composite siding and grey shingle roofing materials. A grass material is also used to the ground.



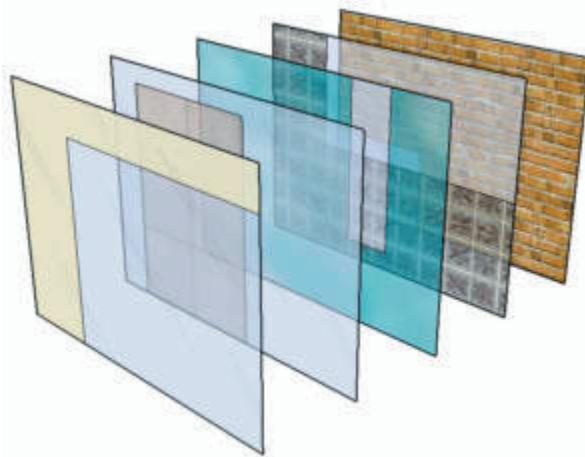
As with components, SketchUp comes with a library of materials. You can paint these materials on your models using the Paint Bucket Tool. Additionally, you can create your own materials within SketchUp (using the color wheel), or by importing texture-like images (such as the image of a wood flooring). In fact, you can import an entire image of a real-world object (such as a picture of a car), and manipulate it over your 3D model to give your model a truly realistic appearance.

### Default Material

Geometry in SketchUp is assigned a *default material* as it is created. The default material can be changed by painting the geometry with a material.

## Material Transparency

Materials also have an opacity property (a number between 0 and 100%) allowing you to create materials that behave like glass. Paint these materials on faces to create windows.



SketchUp materials are normally applied to a single side of a face at a time. Painting a default colored side with transparent material will result in both sides of that face being treated as transparent material allowing the surface to be transparent when viewed from both sides. If the back side of a face has already been painted with some non-transparent material, applying a transparent material to the front side will not cause the back side to also display as transparent. Likewise, if you paint the back side of a face with a different transparent material, it will not effect the front side. Thus, by specifically applying a material to both sides, it is possible to have transparent faces that can have different colors and levels of transparency on each side.

## Shadows

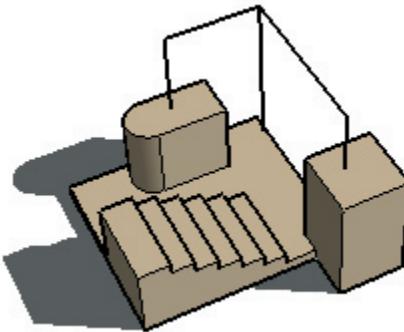
SketchUp allows you to cast shadows on your model as though the model were in a real-world environment. These shadows can be cast based on time of day and virtual location of the model in the real-world. For example, you can set shadows to see exactly what a model of a house would look like at 10:20 am, December 10, in Boulder, Colorado, which is the default location. Finally, SketchUp's shadows are designed to provide dynamic feedback as you change geometry and your camera viewpoint. The shadow casting feature in SketchUp is a great way to give your models a better sense of depth and realism.



**Note** - While SketchUp's projected shadow angles are accurate, the rendering effect is not intended to be photo-realistic. Fortunately, SketchUp can export models to other many other applications that excel at photo-realistic renderings.

## Ground Shadows

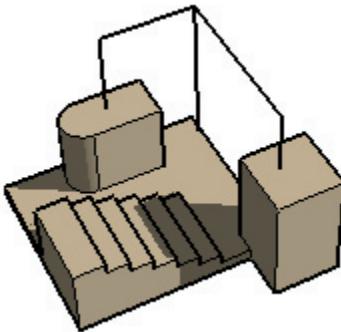
*Ground shadows* use the faces in your model to create a flattened set of faces on the ground plane. These faces are colored and positioned based on the background color and the angle of the sun. Although faster than face shadows, the illusion that ground shadows provide only works on the ground plane. The following figure shows a model whose geometry all lies on or above the ground plane.



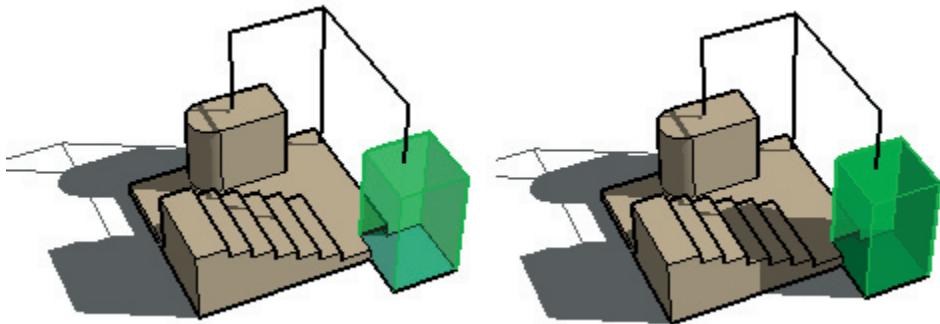
**Note** - Ground shadows create unexpected geometry, called artifacts, when your model contains geometry beneath the ground plane. Ground shadows works well for models with solid color backgrounds and in which all geometry is above the ground plane.

## Face Shadows

*Face shadows* use the sun angle to project shadows based on the location of faces relative to other faces. For example, a shadow is cast on the stairs in the following figure, based on the face of the vertical 3D rectangle. The calculation used to create face shadows is processor-intensive, however, and will slow down performance with large models.



**Note** - Faces with less than 70 percent opacity are considered transparent, and do not cast shadows. Faces with 70 percent opacity or greater cast shadows. Transparent faces cannot receive shadows. The following image shows a transparent (less than 70% opacity) rectangle that does not cast a shadow (left) and a opaque (70% or greater opacity) rectangle that does cast a shadow (right).



The two shadow systems are designed to be complimentary, and you may often wish to have both types enabled simultaneously.

# Presenting Your Models

There are several things you can do after you create a model. For example:

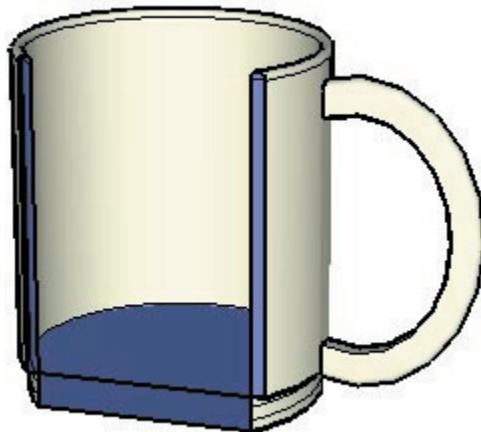
- You can add dimensions, section cut effects, and other entities to your model for use in documenting the actual physical item.
- You can present the model within SketchUp as a TourGuide Tour (a animated tour of your model)
- You can print your model
- You can export all or portions (such as a section slice) of the model for post processing in another application

## Section Cut Effects

SketchUp allows you to create *section cut effects* which are the result of slicing through your model to see and work inside its interior. The following image shows a model of a building with a section cut effect active allowing the designer to work inside the model or present interior detail to a client.

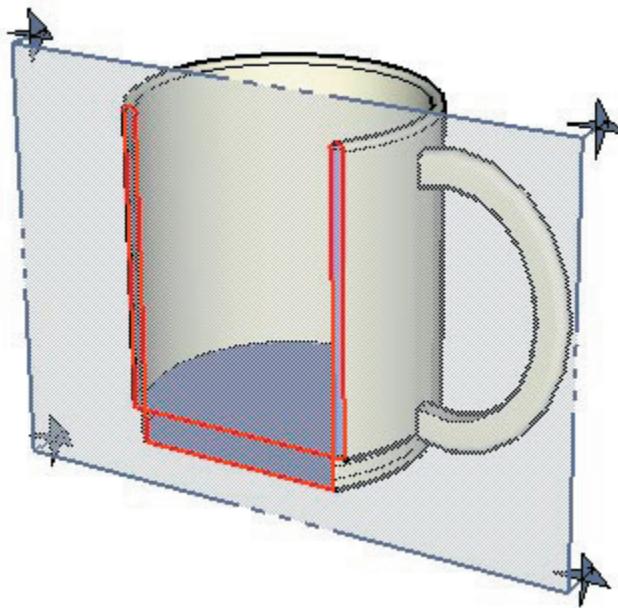


The following model shows the section cut effect resulting from slicing through the model of a cup.



## Section Planes

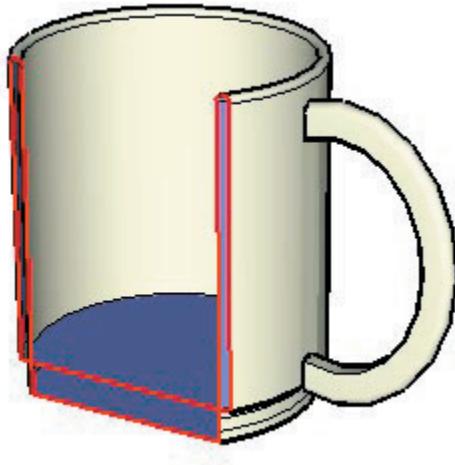
Section cut effects are created by *section planes* which are special entities that are used to control the selection, placement, orientation, direction, of the *section slice*. Section planes are generated using the Section Plane Tool.



The previous image shows a section plane entity intersecting the cup and creating a section slice through the house.

## Section Slices

The term *section slice* refers to the edges that are highlighted after intersecting geometry with a section plane. The following image shows a section slice in red.



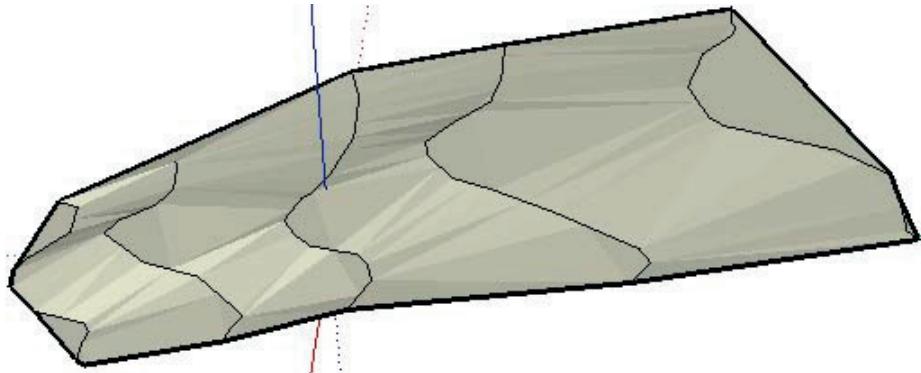
These edges act as dynamic virtual edges in that they continually change as you move the section plane through your model. You can create a group from these edges, such as when slicing horizontally through a house, to create a wire frame of the model (such the outline of a floor plan). Then, export this section slice for use in a CAD program to add additional detail (such as wall construction detail).

## TourGuide Tours

SketchUp allows you to create different *pages*, similar to slides in traditional presentation software, each containing different settings for your model (point of view, shadows, section cut effects, and so on). These pages can be combined and executed sequentially as a TourGuide Tour. The TourGuide engine can be set to gradually transition between effects on different pages for a truly dynamic presentation.

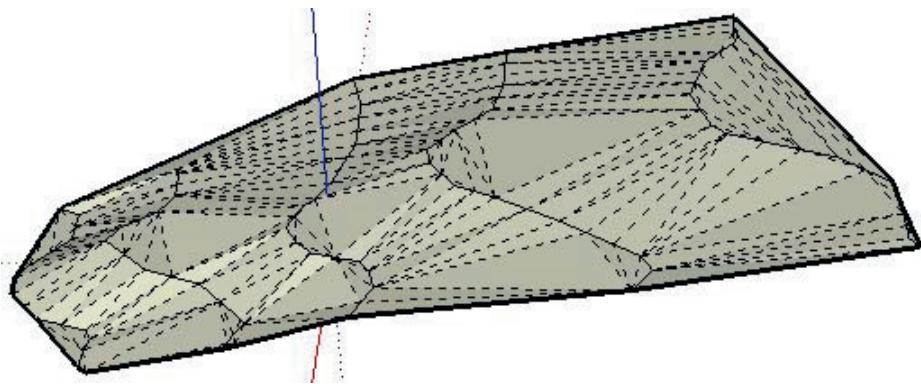
# Modeling Terrain and Organic Shapes

SketchUp uses the concept of a sandbox which refers to a surface that can be generated and manipulated using sandbox tools. A sandbox in SketchUp (and in other 3D modeling tools) is commonly referred to as a *triangulated irregular network* or TIN in terrain modeling terminology. The following image shows a TIN (sandbox) in SketchUp.



**Note** - This documentation uses the term *TIN*, *sandbox*, and *terrain* interchangeably. A *TIN* is like a *sandbox* because it has a boundary and contains a sculpted surface (*sand*). *Sandbox tools* are those tools that are included in SketchUp for creating and manipulating *TINs*.

The following image shows a TIN in SketchUp with hidden lines displayed (so you can see the triangles that are the foundation of the TIN).



There is nothing special about a TIN, it is simply several connected triangular faces that, when smoothed, appear like one continuous smooth surface. Note that you are not limited to creating terrain with the sandbox tools, you can create other *organic shapes* (shapes that appear to be crafted by hand) using these tools. For example, you might create a person's face, a bowl, or a swimming pool using the sandbox tools.

Another type of TIN-like surface, which the sandbox tools can work with, is called a *polygon mesh*. A polygon mesh is similar to a TIN, but contains faces with more than three vertices.



**Note - SketchUp's Sandbox From Scratch Tool and Sandbox From Contours Tool only create TINs, but you can import a polygon mesh or create a polygon mesh (manually) using other SketchUp tools.**

## Triangulation

Triangles in a TIN can be oriented in different directions. The orientation of triangles is referred to as *triangulation*. Notice in the previous image that some triangles are oriented vertically while others are oriented horizontally. This concept is important because some sandbox tools allow you to change the direction of the triangles as a way to further smooth TINs.

## Creating a TIN

There are several ways to obtain a starting TIN as the foundation of your model. These are:

- Create or import contour lines and use the Sandbox From Contours Tool to create a TIN.
- Import an image of a site plan or contour map and trace its contours with the Freehand Tool. Then, adjust the contours to their proper elevation and use the Sandbox From Contours Tool to create a TIN.
- Import a TIN using the SketchUp ArcGIS Plug-in available at [www.http://www.sketchup.com/markets/gis.php](http://www.sketchup.com/markets/gis.php). ArcGIS is an industry standard application suite or Geographic Information System for working with geographical information such as the location of buildings within a world-wide global coordinate system.
- Import a digital terrain model (DTM) file.
- Create sandbox (TIN) using the Sandbox From Scratch Tool.

## Modifying a TIN

SketchUp contains several tools for modifying TINs. The following table lists all of the sandbox tools in SketchUp.

Tool	Type	Notes
Sandbox From Scratch Tool	Create TIN	Creates a flat, rectangular shaped, TIN or sandbox that can be modified by other sandbox tools.
Sandbox From Contours Tool	Create TIN	Creates TIN or a sandbox from contour lines at various elevations.
Smoove	Large Modifications	Allows you sculpt or make large modifications to a TIN by moving groupings of vertices to create hills, berms, or other features.
Stamp	Large Modifications	Allows you to sculpt or make large modifications to a TIN by stamping footprints into the TIN, such as the footprint of a building.
Drape	Large Modifications	Allows you to drape the outlines of roads, paths and buildings, drawn on a flat plane, onto a TIN.
Add Detail	Small, Detailed, Modifications	Allows you to sculpt or make small, detailed, modifications to the a TIN by adding additional triangles to a TIN.
Flip Edge	Small, Detailed, Modifications	Allows you to sculpt or make small, detailed, modifications to the TIN by adjusting the triangulation of a TIN.



**Note -** The sandbox tools can also be used to create organic geometry or geometry that appears to be hand-crafted.

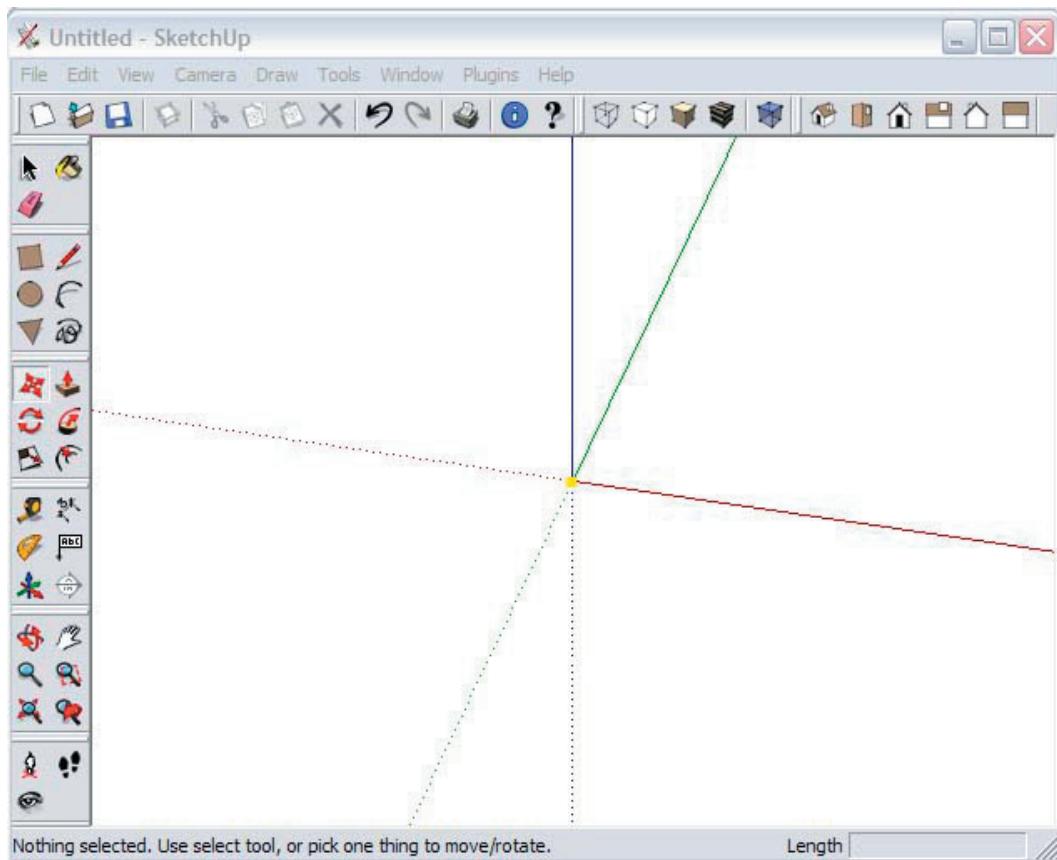
## Functional Terrain

The term *functional terrain* is used to describe terrain that has no portion bending back upon itself creating overhangs, underhangs or caves. If you draw a vertical line through your terrain at any point and the line only touches the terrain at one point, your terrain is functional.

This concept is important because certain tools, such as the Smoove Tool, can create unexpected results when working on nonfunctional terrain (the neighboring points included in the sculpting operation by the Smoove Tool are often undesirable).

# Application User Interface

The SketchUp user interface is designed to be simple and easy to use. The main parts of the SketchUp user interface are the title bar, the drawing area, the status bar, and the value control box.



## Title Bar

The title bar (at the top of SketchUp) contains the standard Microsoft Windows controls (close, minimize, and maximize) on the right, and the name of the currently opened file.

A blank drawing area appears when you start SketchUp and the name of the currently opened file is "Untitled" in the title bar, indicating that you have not yet saved your work.

## Menus

Menus appear below the title bar. The majority of SketchUp tools, commands, and settings are available within these menus. The menus that appear by default are: File, Edit, View, Camera, Draw, Tools, Window, Help.

## Toolbars

The Toolbars, appearing below the menus and along the left side of the application contain a user-defined set of tools and controls. Controls can be added to the toolbars below the menus using the View > Toolbars menu item.

## Drawing Area

The *drawing area* is where you create your model. The 3D space of the drawing area is identified visually by the drawing axes.

## Status Bar

The status bar is the long gray rectangular area at the bottom of the drawing area.



The left side of the status bar displays tips for the currently used drawing tools, including special functions accessible using keyboard shortcuts. Watch the status bar to discover advanced capabilities of each of the SketchUp tools.



**Note** - Use the resize handle to make the drawing area larger so you can see the entire message in the status bar.

## Value Control Box (VCB)

The value control box (VCB) is located on the right side of the status bar. The VCB displays dimensional information while you draw. You can also enter values into the VCB to manipulate currently selected entities, such as creating elements of a specific dimension. Other behaviors of the VCB are as follows:

- You can type a value in the VCB before or after an operation is complete as long as the value is entered prior to the start of a new operation. Detailed information on the VCB values accepted for each tool appears in the Drawing Tools and Modification Tools sections of this user's guide.
- You must press the Enter or Return key to accept a typed value.
- You can change the value of the geometry as many times as you like before you start a new operation.
- The VCB cannot be used again to enter values for a tool once you have exited the tool.

- It is possible to specify values that are not within your drawing's precision parameters. SketchUp will display a tilde (~) before the number to indicate that a number is not precise.
- It is not necessary to click in the VCB before typing. The VCB is always awaiting input from your keyboard.
- You can type values in the VCB using an alternate measuring system than the default system. SketchUp will convert the value to the default system. For example, you can type in 3' 6" even if you are using metric system as your default. Units are set within the Units panel of the Model Info dialog box.

## Window Resize Handle

To the right of the VCB is the window resize handle which is used to change the size of the drawing area.

# Menus

There are eight menus in SketchUp. These menus are the File, Edit, View, Camera, Draw, Tools, Window, and Help menus.

## File Menu

The File menu contains items that relate to SketchUp model files, including commands to create, open, save, print, import, and export model files.

### New

The New menu item is used to close the current document and create a blank drawing area to begin a new SketchUp model. You will be prompted to save your changes if you have not saved changes to the current model before selecting the New menu item. Sketchup will use the settings in the template file to define the initial model state if you have selected a template file in the template panel under SketchUp > Preferences.

### Keyboard Shortcut: Ctrl-N



***Tip*** - Open another SketchUp application if you want multiple drawing areas open simultaneously.

### Open...

The Open menu item is used to launch the Open dialog box, allowing you to open a previously saved SketchUp file. You will be prompted to save your changes if an unsaved model is already open because only one file can be open at a time.

### Keyboard Shortcut: Ctrl-O

**Problem Detection:** Minor problems can occur with your model given the infinite flexibility that SketchUp provides when designing in 3D. You likely will not notice anything different in your model, except that it will perform better, after these problems are fixed.

SketchUp checks all files for problems when they are opened and saved if the Automatically check models for problems check box is checked in the General Panel of the Application Preferences. SketchUp displays a dialog box allowing you to fix problems if they are found. This dialog box has three options: Always fix my models, Fix it now, and Fix it later. @Last Software recommends you click on the Always fix my models button in this dialog box to fix the problem. This button also checks the Automatically fix problems when they are found checkbox in the General Panel of the Application Preferences. This preferences option causes SketchUp to automatically fix problems without user intervention. Refer to the Automatically fix problems when they are found option for further information.

SketchUp will check your model for unrecoverable errors during the open, preventing the overwriting of a good auto-save file. Sketchup will display a dialog box in the rare case that unrecoverable errors are found in your model. This dialog box contain the option to quit SketchUp and send a report. @Last Software recommends you click on the quit SketchUp and send report button to terminate SketchUp and preserve your previously valid auto-save file. This report contains valuable information regarding the unrecoverable errors.

## Save

The Save menu item is used to save the currently active SketchUp model to your file system. When you close an unsaved document, or attempt to quit SketchUp with unsaved open documents, SketchUp will prompt you to save your work before continuing.

### Keyboard Shortcut: Ctrl-S



**Tip -** If Create Backup is enabled within the General panel of the preferences dialog box, the existing file will be converted to a backup file (.skb), and the new drawing will be saved in place of that currently existing file (.skp). The Create Backup option can help preserve your data in the event of an accidental removal of a .skp file.



**Tip -** It is good to save often. You can have SketchUp automatically save for you at a specific time increment by enabling the Auto-save option within the General panel of the Preferences dialog box.

**Problem Detection:** Minor problems can occur with your model given the infinite flexibility that SketchUp provides when designing in 3D. You likely will not notice anything different in your model, except that it will perform better, after these problems are fixed.

SketchUp checks all files for problems when they are opened and saved if the Automatically check models for problems check box is checked in the General Panel of the Application Preferences. SketchUp displays a dialog box allowing you to fix problems if they are found. This dialog box has three options: Always fix my models, Fix it now, and Fix it later. @Last Software recommends you click on the Always fix my models button in this dialog box to fix the problem. This button also checks the Automatically fix problems when they are found checkbox in the General Panel of the Application Preferences. This preferences option causes SketchUp to automatically fix problems without user intervention. Refer to the Automatically fix problems when they are found option for further information.

SketchUp will check your model for unrecoverable errors during the save, preventing the overwriting of a good auto-save file. Sketchup will display a dialog box in the rare case that unrecoverable errors are found in your model. This dialog box contain the option to quit SketchUp and send a report. @Last Software recommends you click on the quit SketchUp and send report button to terminate SketchUp and preserve your previously valid auto-save file. This report contains valuable information regarding the unrecoverable errors.

## Save As...

The Save As... menu item is used to open the Save As dialog box which defaults to the current document's folder. You can use this dialog box to save the current drawing as a new document. This file can be assigned a new name, a new location, and a previous version of SketchUp. The new file will becomes the current file in the drawing window.

**Problem Detection:** Minor problems can occur with your model given the infinite flexibility that SketchUp provides when designing in 3D. You likely will not notice anything different in your model, except that it will perform better, after these problems are fixed.

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dialog box has three options: Always fix my models, Fix it now, and Fix it later. @Last Software recommends you click on the Always fix my models button in this dialog box to fix the problem. This button also checks the Automatically fix problems when they are found checkbox in the General Panel of the Application Preferences. This preferences option causes SketchUp to automatically fix problems without user intervention. Refer to the Automatically fix problems when they are found option for further information.

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### Save A Copy As...

The Save A Copy As... menu item is used to save a new file based on your current model. This menu item does not overwrite or close the current file and is useful for saving incremental copies or tentative schemes of your work.

**Problem Detection:** Minor problems can occur with your model given the infinite flexibility that SketchUp provides when designing in 3D. You likely will not notice anything different in your model, except that it will perform better, after these problems are fixed.

SketchUp checks all files for problems when they are opened and saved if the Automatically check models for problems check box is checked in the General Panel of the Application Preferences. SketchUp displays a dialog box allowing you to fix problems if they are found. This dialog box has three options: Always fix my models, Fix it now, and Fix it later. @Last Software recommends you click on the Always fix my models button in this dialog box to fix the problem. This button also checks the Automatically fix problems when they are found checkbox in the General Panel of the Application Preferences. This preferences option causes SketchUp to automatically fix problems without user intervention. Refer to the Automatically fix problems when they are found option for further information.

SketchUp will check your model for unrecoverable errors during the save a copy as, preventing the overwriting of a good auto-save file. Sketchup will display a dialog box in the rare case that unrecoverable errors are found in your model. This dialog box contain the option to quit SketchUp and send a report. @Last Software recommends you click on the quit SketchUp and send report button to terminate SketchUp and preserve your previously valid auto-save file. This report contains valuable information regarding the unrecoverable errors.

### Revert

The Revert menu item is used to revert your current document to its last saved state.

### Export

The Export submenu is used to access SketchUp's export functionality, which is useful for sharing your work with other people or exporting your drawings for use in other applications. You can export your SketchUp model as a 3D model, a 2D graphic, section slice, or animation.

**3D Model...:** The 3D Model... menu item is used to export you SketchUp file to 3D formats, including the AutoCAD 2000 DXF/DWG and AutoCAD R14 DXF/DWG CAD formats.

**2D Graphic...:** The 2D Graphic... menu item is used to export 2D bitmap and dimensionally accurate, resolution independent, 2D vector drawings. Pixel-based images can be exported in JPEG, PNG, TIFF, BMP, TGA, and Epix file formats. These formats allow you to capture the image exactly as you see it on your screen, including shadow and texture settings. You can also specify the image size in pixels, allowing you to export images in much higher resolution. This menu item allows you to apply anti-aliasing, which eliminates the jagged look of pixelation. Note that larger images will take longer to export.

Vector images can be exported in PDF, EPS, DWG, and DXF file formats. This option makes it easy to send your SketchUp files to a plotter, quickly integrate them into construction documentation, or further modify your models using vector-based illustration software. Note that vector output formats may not support certain display options, such as shadows, transparency, and textures.

**Section Slice...:** The Export 2D Section Slice menu item allows you to output dimensionally accurate 2D section slices in standard vector formats.

**Animation...:** The Animation... menu item is used to export a pre-rendered animation file containing the page sequence you have created. This option makes it easy archive your TourGuide tours to CD or DVD and to create smooth animations of complex models.

## Import

The Import menu allows you to import information from other files into your SketchUp drawings.

## Print Setup...

The Print Setup... menu item is used to access to the print setup dialog box. This dialog box is used to select and configure printer and page properties for printing.

## Print Preview...

The Print Preview... menu item is used to preview your model as it will appear on paper (using the print setup settings).

## Print...

The Print... menu item opens the standard Print dialog box. This dialog box enables you to print the current model in SketchUp's drawing area to the currently selected printer.

## Keyboard Shortcut: Ctrl-P

### (Recently Opened File List)

The (Recently Opened File List) menu item lists recently opened SketchUp files. Select a file from this list to open the file.

### Exit

The Exit menu item closes the current file and the SketchUp application window. SketchUp will notify you to save your file if it has not been saved since the last change.

## Edit Menu

The Edit menu contains items allowing you to perform editing functions on SketchUp geometry. These menu items include those for creating and editing groups and components, visibility operations, and standard cut, copy, paste commands.

### Undo

The Undo menu item is used to undo the last drawing or editing commands performed. SketchUp allows you to undo all operations you have performed, one at a time, to the state at which you saved your file. The number of possible sequential Undo commands is limited to 100 steps.

#### **Keyboard Shortcut: Ctrl-Z**



**Note** - Undo will undo any operation that creates or modifies geometry, but it does not undo changes to your point of view. Use the Previous menu item under the Camera menu to undo a point of view change.

### Redo

The Redo menu item returns the last undo to its previous state.

#### **Keyboard Shortcut: Ctrl+Y**

### Cut

The Cut menu item removes the selected elements from your model and places them in the clipboard. The contents of the clipboard can then be inserted back into any open SketchUp document by using the Paste menu item.

Contents of the clipboard will remain on the clipboard until replaced with other content using an additional Cut or a Copy command.

#### **Keyboard Shortcut: Ctrl+X**

## Copy

The Copy menu item copies the selected items to the clipboard without deleting the items from the model. The contents of the clipboard can then be inserted back into any open SketchUp document by using the Paste menu item.

Contents of the clipboard will remain on the clipboard until replaced with other content using an additional Cut or a Copy operation.

### **Keyboard Shortcut: Ctrl+C**

## Paste

The Paste menu item copies the contents of the clipboard into the current SketchUp document. The pasted objects will be attached to and placed by the point of the cursor, allowing you to position the new geometry when it is pasted. Click to set the pasted objects in place.

Contents of the clipboard will remain on the clipboard until replaced with other content using an additional Cut or a Copy operation.

### **Keyboard Shortcut: Ctrl+V**

## Erase

The Erase menu item is used to remove the currently selected entities from your model.

## Select All

The Select All menu item is used to select all selectable entities in the model. Hidden entities, any items on a hidden layer, or geometry that is clipped away using a section plane cannot be selected with the Select All menu item.

### **Keyboard Shortcut: Ctrl+A**

## Deselect All

The Deselect All menu item clears the selection set, deselecting any currently selected items in the model.

### **Keyboard Shortcut: Ctrl+T**

## Hide

The Hide menu item is used to hide any selected object. Hiding geometry can help simplify your current view, or enable viewing and working inside tight areas.



## Unhide

**Selected:** The Unhide menu item unhides any selected hidden object. Ensure Show Hidden Geometry is enabled under the View Menu to view and select hidden geometry.

**Last:** The Last menu item unhides the last entities hidden with the Hide command.

**All:** The All menu item unhides all hidden entities in your current document.

## Construction Geometry

**Hide All:** The Hide All menu item hides all construction line entities in the drawing area.

**Unhide All:** The Unhide All menu item unhides all construction line entities in the drawing area.

**Erase All:** The Erase All menu item deletes all construction line entities from the drawing area.

## Lock

The Lock menu item is used to lock any component or group that you do not want to be able to be moved or edited.

## Unlock

The Unlock submenu contains options for unlocking components and groups.

**Selected:** The Selected menu item is used to unlock all components and groups in a selection set.

**All:** The All menu item is used to unlock all components and groups in your drawing area.

## Make Component

The Make Component... menu item is used to create a component entity from the selected entities.

## Make Group

The Make Group menu item creates a group entity from the selected entities.

## Close Group/Component

The Close Group/Component menu item allows you to exit from a group or component's context or editing session.

## Intersect With Model

Complex geometry in SketchUp can be easily created using the Intersect With Model menu item (also called the *Intersector*). This menu item allows you intersect two elements, such as a box and a tube, and automatically create edges where the elements intersect. The face within the new edges can be deleted, pushed, pulled or manipulated in other ways.

## Entity Commands Sub-Menu

The Entity Commands sub-menu contains all of the commands available to manipulate the currently selected entity (which are the same as the commands found in the entity's context menu). The sub-menu's name and contents will change depending on the entity that you have selected.

## View Menu

The View menu contains menu items that alter the display of entities within your model.



**Note -** Items in the View menu alter the display of entities. Items in the Camera menu alter your point of view.



## Toolbars

The Toolbars menu item is used to toggle the display of all of the toolbars. These toolbars are: Camera, Construction, Drawing, Display Style, Modification, Principal, Sections, Shadows, Standard, Views, and Walkthrough. The Large Buttons option toggles large toolbar buttons on and off. This option can enhance usability on tablets or low-resolution displays.

## Hidden Geometry

The Hidden Geometry menu item is used to display hidden geometry or entities that have been hidden using the Hide menu item or context command. The Hidden Geometry menu item displays hidden faces with a light cross-hatch pattern (edges are displayed dashed), enabling you to select the geometry. Once selected, hidden geometry can be made visible with the Unhide and Unhide All menu items.

## Section Planes

The Section Planes menu item toggles the display section plane entities.

## Section Cut

The Section Cut menu item toggles the display of any section cut effects.

## Axes

The Axes menu item toggles the display of the drawing axes.

## X-ray

The X-ray menu item activates X-ray mode.

## Rendering

The Rendering submenu contains options to activate face and edge rendering styles. The face rendering styles are Wireframe, Hidden Line, Shaded, Shaded with Textures, and Monochrome. The edge rendering styles are Profile Edges, Depth Cued Edges, Extended Edges, Endpoints, and Jitter Edges. See the Display Settings topic for further information.

**Component Edit:** The Component Edit submenu contains commands to alter the display of other entities when editing components.

**Hide Rest of Model:** The Hide Rest of Model menu item is used to toggle the display of the model when editing a component.

**Hide Similar Components:** The Hide Similar Components menu item is used to toggle the display of similar components when editing a component.

## Page Tabs

The Page Tabs command toggles the display of page tabs. See the Page Manager topic for further information.

## TourGuide

The Tourguide submenu contains several menu items related to pages and TourGuide. See the Pages and TourGuide topic for further information.

**Add Page:** The Add menu item is used to add a new page to the current file.

**Update Page:** The Update menu item is used to update a page if you have made changes to the page.

**Delete Page:** The Delete menu item is used to delete a page from the current model.

**Previous Page:** The Previous Page menu item is used to transition to the previous page.

**Next Page:** The Next Page menu item is used to transition to the next page.

**Play Slideshow:** The Play Slideshow menu item starts a TourGuide slideshow. See the Pages and TourGuide topic for further information.

## Camera Menu

The Camera menu contains items for altering your point of view of the model.

### Previous

The Previous menu item is used to undo the last performed camera command.

### Standard

The Standard submenu provides access to standard views: Top, Bottom, Front, Right, Back, Left, and Isometric. Selecting any of these model views will immediately set your active drawing window to that view.

### Perspective

The Perspective menu item is used to toggle between perspective and paraline projections. You are in a perspective projection when this item has a checkmark.



**Note -** SketchUp must be in paraline mode to print to scale. Be aware that faces and edges that are parallel to the view plane will be measurable.

### Field of View

The Field of View menu item is used to invoke the Zoom Tool in Field of View mode allowing you to widen or narrow your field of view.

### Orbit

The Orbit menu item invokes the Orbit Tool.



## Pan

The Pan menu item invokes the Pan Tool.

## Zoom

The Zoom menu item launches the Zoom Tool.

## Zoom Window

The Zoom Window menu item launches the Zoom Extents Tool.

## Zoom Extents

The Zoom Extents menu item launches the Zoom Window Tool.

## Position Camera

The Position Camera menu item launches the Position Camera Tool allowing you to investigate fixed views of your model (such as an eye-level view of a house). This option is often used with the Walk Tool to take an eye-level tour of a structure.

## Walk

This command invokes the Walk Tool for maneuvering through your SketchUp model as though you are taking a virtual tour of the model.

## Look Around

The Look Around menu item invokes the Look Around Tool which pivots the camera around a stationary point at the point of view.

## Draw Menu

The Draw menu contains all of the SketchUp drawing tools and provides an alternative to using the Toolbars or keyboard shortcuts.

### Line

The Line menu item invokes a Line Tool used to draw line entities, or edges, within the drawing area.

### Arc

The Arc menu item invokes a Arc Tool used to draw Arc entities, comprised of multiple connected straight line segments (editable as a single curve).

### Freehand

The Freehand menu item is used to invoke a Freehand Tool used to draw irregular, coplanar connected lines in the form of Curve entities.

### Rectangle

The Rectangle menu item invokes a Rectangle Tool used to draw four coplanar intersecting edges and a subsequent Face entity.

### Circle

The Circle menu item invokes a Circle Tool used to draw Circle entities.

### Polygon

The Polygon menu item invokes a Polygon Tool used to draw regular Polygon entities, inscribed within a circle, consisting of 3 to 100 sides.

## Tools Menu

The Tools menu provides access to all of the modification tools in Sketchup. This menu represents one of three mechanisms for accessing these tools (additional mechanisms are the Toolbars and keyboard shortcuts).

### Select

The Select menu item invokes a Select Tool allowing you to select one or more entities to modify.

### Eraser

The Eraser menu item is used to invoke the Eraser Tool allowing you to erase entities from the drawing area. This tool also allows you to hide and soften edges.

### Paint Bucket

The Paint Bucket menu item invokes the Paint Bucket Tool used to assign materials to entities in your model. Use the Paint Bucket tool to paint individual elements, fill a number of connected faces, or replace a material in your model.

### Move

The Move menu item invokes the Move Tool allowing you to move, manipulate, and copy geometry. This tool can also be used to rotate Component entities.

### Rotate

The Rotate menu item is used to invoke the Rotate Tool used to rotate drawing elements and



single or multiple objects within a single rotation plane. The Rotate Tool can also be used to stretch and distort geometry by selecting only a portion of a model.

### Scale

The Scale menu item invokes the Scale Tool allowing you to resize and stretch selected geometry relative to other elements in your SketchUp model.

### Push/Pull

The Push/Pull menu item invokes the Push/Pull Tool used to manipulate faces of your model. This tool can be used to displace, extrude, re-attach, or subtract faces, depending on the context of selected geometry.

### Follow Me

The Follow Me menu item is used to invoke the Follow Me Tool used to extrude faces along a path such as an edge or line drawn with the Freehand Tool. This tool is useful when adding details to a model because it allows you to draw the detail at one end of a path on the model and then repeat that detail along the path.

### Offset

The Offset menu item invokes the Offset Tool used to create copies of co-planar lines and faces that are a uniform distance from original lines and faces. Lines and faces can be offset either inside or outside the original face. Offsetting a face will always create a new face.

### Tape Measure

The Tape Measure menu item is used to invoke the Tape Measure Tool used to perform a number of dimension-related operations. These operations include measuring the distance between two points, creating Construction Line entities, and re-scaling an entire model to an exact dimension.

### Protractor

The Protractor menu item invokes the Protractor Tool allowing you to measure angles and create Construction Line entities (usually at some angle).

### Axes

The Axes menu item invokes the Axes Tool allowing you to move the drawing axes. This tool is often used when constructing rectangular objects that are skewed relative to one another, or to allow for more accurate scaling of entities that are not oriented along the default coordinate planes.

## Dimensions

The Dimensions menu item is used to invoke a Dimension Tool used to place Dimension entities in your model.

## Text

The Text menu item launches a Text Tool used to insert Text entities into your model.

## Section Plane

The Section Plane menu item invokes a Section plane Tool used to make section cuts in your model.

## Terrain

The Terrain submenu contains options for modifying terrain. This menu item only appears after enabling the terrain modeling tools using the Extensions panel of the Preferences dialog box. See the [Terrain Modeling Tools](#) topic for further information.

## Utilities

The Utilities submenu contains various utilities and macros created using the Sketchup Ruby API. This menu item only appears after enabling the utilities tools using the Extensions panel of the Preferences dialog box.



**Note - Ruby scripts are contained in the Plugins directory under the installation directory.**

**Create Face:** The Create Face menu item invokes the Create Face utility allowing you to troubleshoot face creation and, in most cases, create a face for three or more intersecting edges. Specifically, the Create Face utility is useful when:

- A model has two edges that have a common end point geometrically (the coordinates of the end point are the same), but the edges are not connected topologically. In this case, the Create Face utility will properly connect edges and create a face.
- A face cannot be created because edges are not exactly planer, or appear to be connected when there is actually a small gap between the ends of the edges. In these instances, Create Face will not create a face, but displays a message indicating why SketchUp cannot create a face. This message can be used to troubleshoot face creation.

**Query Tool:** The Query Tool menu item invokes a Query Tool utility that displays the current mouse position in the VCB.

**Fix Non-planar Faces:** The Fix Non-planar Faces menu items invokes the Fix Non-planar Faces utility to find and fix non-planar faces. Fix Non-planer Faces looks at the Vertices of every face and checks to see if they are on the plane of the face. If not, Check Validity tries to break the face into triangle so that each triangular face will be planar.

Fix Non-planar Faces can actually change the model, but does not move vertices or edges. Instead, Check Validity will split faces into triangular faces if they were not planar.



**Note -** Non-planar faces can get created by a bug in the DWG importer.

## Window Menu

The Window menu contains model settings and managers for your models. Model settings are dialog boxes with settings that affect the model, such as shadows, display settings, and model information. Managers are dialog boxes that manage certain aspects of your model such as pages, layers, materials, and components.

### Model Info

The Model Info menu item is used to display the Model Info dialog box used to set settings for your entire model.

### Entity Info

The Entity Info menu item displays the Entity Info dialog box used to view and set settings for the currently selected entity (or entities).

### Materials Browser

The Materials Browser menu item invokes Material Browser used for selecting and applying colors and materials to your models.

### Material Editor

The Material Editor menu item displays the Material Editor dialog box used for editing and mixing colors and materials.

### Components

The Components menu item displays used for managing your components.

### Layers

The Layers menu item is used to display the Layer Manager used to manage your SketchUp layers.

## Pages

The Pages menu item invokes the Page Manager used to manage pages for a TourGuide slideshow. The Page Manger will display with the current page highlighted.

## Display Settings

The Display Settings menu item invokes the Display Settings dialog box containing options to activate the face and edge rendering styles. The face rendering styles are wireframe, hidden line, shaded, shaded with textures, and Monochrome. The edge rendering styles are profile lines, jitter lines, and extended edges.

## Shadow Settings

The Shadow Settings menu item invokes Shadow Settings dialog box for manipulating shadows in your model.

## Soften Edges

The Soften Edges menu item invokes the Soften Edges dialog box used to soften and smooth edges in your model.

## Component Outliner

The Component Outliner menu item invokes the Component Outliner used to display components and groups in a hierarchy.

## Preferences

The Preferences menu item invokes the Application Preferences dialog box where you can set various global settings for the program.

## Hide/Show Dialogs

The Hide/Show Dialog menu item is used to hide and show all currently active dialog boxes.

## Ruby Console

The Ruby Console menu item displays the Ruby Console where you can type Ruby script.

## Help Menu

### SketchUp Help

The SketchUp Help menu item opens SketchUp's online user's guide. The user's guide contains reference information for SketchUp.



## Quick Reference

The Quick Reference menu item is used to display SketchUp's Quick reference card in your default PDF viewer.

## SketchUp Web Site

The SketchUp Web Site menu item invokes your default web browser and connects you to the SketchUp web site. The web site contains SketchUp support forums, additional training information, and access to new software releases.

## View Tutorials

The View Tutorials menu item launches the Video Tutorials portion of the SketchUp web site. Select this command to open your default web browser and connect to the SketchUp online video tutorials.

## License

The License submenu includes several options allowing you to view and manage your SketchUp license.

**License Info:** The License Info menu item displays information about your SketchUp license, including the type of license and serial number.

**Authorize...:** The Authorize... menu item is used to authorize your SketchUp licence to extend the use of SketchUp beyond the 8 hour trial period. Click on this menu item to display the SketchUp Authorization Screen. Type in the authorization number, provided by @Last Software when you buy SketchUp, to authorize SketchUp.

**Unauthorize...:** The Unauthorize... menu item is used to unauthorize your SketchUp licence. This menu item is used primarily when switching from a single-user license to a multiuser network license.



**Caution - SketchUp will not run after you unauthorize your license. You must authorize a new single-user or network license to use SketchUp again.**

## Set Network License File...

The Set Network License File menu item is used to relocate the network license file on the network.

## Unset Network License File

The Unset Network License File menu item is used to delete the location of the network license file on the network.

### Lan License Monitor

The Lan License Monitor menu item displays the LAN License Monitor dialog box. This dialog box displays single user licenses on the network.

### Ruby Help

The Ruby Help menu item opens Ruby Help online user's guide. The user's guide contains reference information for the SketchUp Ruby APIs and examples.

### About SketchUp...

The About SketchUp... menu item activates the About SketchUp dialog box. This dialog box provides information about your SketchUp version and license.

### About Plugins

The About Plugins sub-menu contains information about additional SketchUp plugins.

# Toolbars

SketchUp's toolbars are similar to those in other Microsoft Windows applications in that each contains a related set of tools. These toolbars can be detached by clicking and dragging the title bar of the toolbar, resized by dragging the corners, and reattached or docked to the edge of the drawing area to suit your preferences. Control the toolbars that are displayed using the View > Toolbars submenu.

## Standard Toolbar

The Standard toolbar contains a variety of menu items which help with file and drawing management, as well as shortcuts to printing and help operations. These menu items are New, Open, Save, Make Component, Cut, Copy, Paste, Erase, Undo, Redo, Print, Preferences, and Context Sensitive Help.



## Principal Toolbar

The buttons on the Principal toolbar activate the Select Tool, Paint Bucket Tool, and Eraser Tool.



## Drawing Toolbar

The buttons on the Drawing toolbar activate the Rectangle Tool, Line Tool, Circle Tool, Arc Tool, Polygon Tool, and Freehand Tool.



## Modification Toolbar

The Modification toolbar contains geometry modification tools. The tools on this toolbar are the Move Tool, Push/Pull Tool, Rotate Tool, Follow Me Tool, Scale Tool, and Offset Tool.



## Construction Toolbar

The buttons on the Construction toolbar activate the Tape Measure Tool, Dimensions Tool, Protractor Tool, Text Tool, Axes Tool, and Section Tool.



## Camera Toolbar

The buttons on the Camera toolbar activate the Orbit Tool, Pan Tool, Zoom Tool, Zoom Window Tool, Zoom Extents Tool, and the Previous camera tools.



## Walkthrough Toolbar

The buttons on the Walkthrough toolbar activate the Position Camera Tool, Walk Tool, and Look Around Tool.



## Display Modes Toolbar

The buttons on the Display Modes toolbar activate SketchUp's face rendering styles (wireframe, hidden line, shaded with textures, and X-ray transparency).



## Views Toolbar

The buttons on the Views toolbar activate SketchUp's standard views (top, front, left, right, back, and iso). The bottom view is not included, but is available from the Camera menu.





## Shadows Toolbar

The Shadows toolbar is used to control shadows. This toolbar contains buttons to launch the Shadow Settings dialog box (left-most icon) and enable/disable shadows (second icon). This toolbar also contains slider bars to control settings related to the time of year (left-most slider bar) and time of day (right-most slider bar).



## Section Planes Toolbar

The Section toolbar allows you to conveniently execute common section operations. The controls on this toolbar include buttons for adding new section planes to the current editing context, toggling the section cut effect, and toggling the display of section planes.



## Large Toolbar Buttons

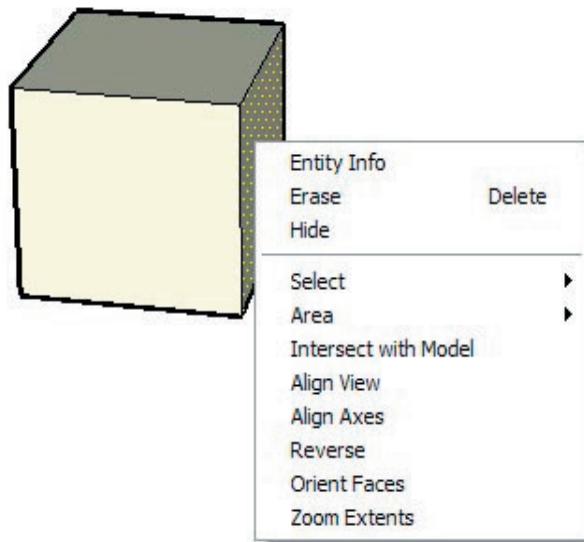
The toolbar can be displayed using large and small tool buttons. Check the Use Large Tool Buttons checkbox in the General panel of the Preferences dialog box to switch between large and small tool buttons.

## Tool Help Tips

Hover over a tool to display a help tag with the name of the tool. A longer help tag will appear in the Status Bar when you activate a tool.

# Context Menus

Menu items also appear in special context menus which are menus whose contents vary depending on the context in which they are invoked (usually on one or more entities in the drawing area or within a user interface component, such as a dialog box). The following image shows a context menu for a Face entity.



## Context Clicking

Context menus are invoked by selecting one or more entities and clicking the right mouse button. This operation is referred to as a *context click*.

Detailed information on the context menu for each entity appears in the Entities section of this user's guide.

Other dialog boxes and user interface components have their own context menus, including drawing axes, page tabs, the Material Browser, and so on.

# Dialog Boxes

Dialog boxes in SketchUp are *modeless* in that they do not limit your interaction with the rest of the application by requiring user input before continuing with your work. Instead, these dialog boxes can be displayed while you work and allow you to make dynamic settings changes to your model while you are sketching. Dialog boxes in SketchUp can be snapped to other user interface components on your screen, resized, and minimized (with some exceptions).

## Snapping Dialog Boxes to Other User Interface Components

Some dialog boxes, such as the Display Settings dialog box, automatically snap to the outside edges of the application window, the outer edge of the screen, with the top and bottom of other dialog boxes to create a *dialog stack*. To snap a dialog box to another user interface component:

1. Display a dialog box, such as the Window > Display Settings dialog box.
2. Click and hold on the dialog box's title bar.
3. Move the dialog box to the outside edge of the application window, outside edge of the screen, or top or bottom edge of another dialog box (such as the Shadow Settings dialog box). The dialog box will snap to and align with the edge.



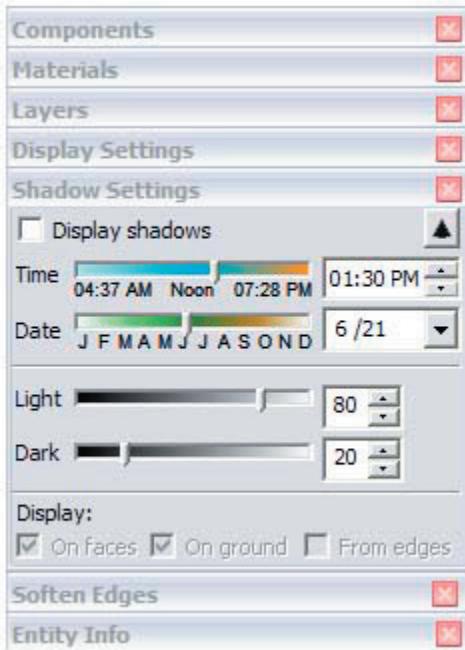
**Note** - Dialog boxes that have been snapped to the application window do not move when the application window is moved (they are not stuck to the window).



**Tip** - Snap several dialog boxes together at their top and bottom edges to form a dialog box stack. Dialog boxes in a dialog box stack move with the stack when the top-most dialog box's title bar is moved.

## Manipulating Dialog Boxes in a Dialog Box Stack

A dialog box stack refers to a series of dialog boxes snapped together at their bottom and top edges such that they form a stack. The following image contains a dialog box stack with the Shadow Settings dialog box opened in the middle.



You can manipulate the dialog boxes in a dialog box stack in the following ways:

- Move the entire dialog box stack by clicking, holding, and moving the top-most dialog box's title bar.
- Click on any dialog box's title bar to minimize or maximize a dialog box within the stack.
- Move a dialog box, residing in the middle of the stack, to the right or left to break it from the stack. Move a dialog box, residing at the bottom of the stack, down to break it from the stack.
- Reinsert the dialog box at the beginning or end of the stack by snapping the dialog box to the bottom or top edge of the stack, respectively.
- Reinsert the dialog box in the middle of the stack by moving sideways into a middle position.
- Snap the entire dialog box stack to the application window or screen by moving it to the outer edge of the application window or screen.



## Resizing Dialog Boxes

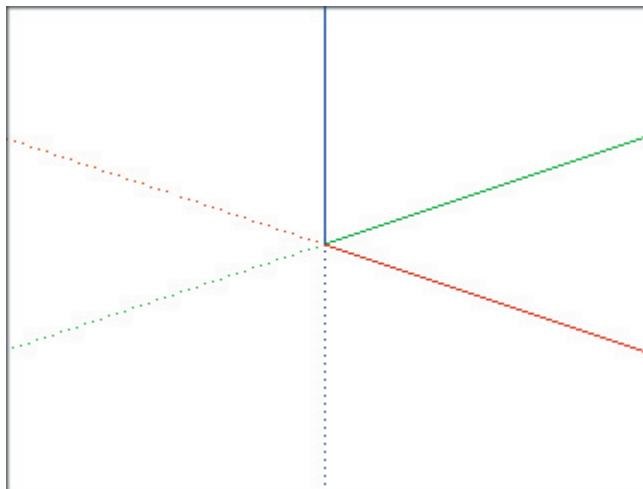
Some SketchUp dialog boxes, such as the Component Browser dialog box, can be resized. Move the cursor around the dialog box's edges to display a resize cursor (two arrows). Click on the edge and move the edge to resize the dialog box.

## Minimizing Dialog Boxes

Click on any dialog box's title bar to minimize or maximize the dialog box (even those that can not be placed in a dialog box stack).

# Drawing Axes

SketchUp's drawing axes consist of three colored lines, perpendicular to each other, displayed in the drawing area. These axes are helpful in providing a sense of direction in 3D space while you work.



**Note** - The Drawing Axes will be automatically hidden in any images exported from SketchUp.

## Moving the Drawing Axes

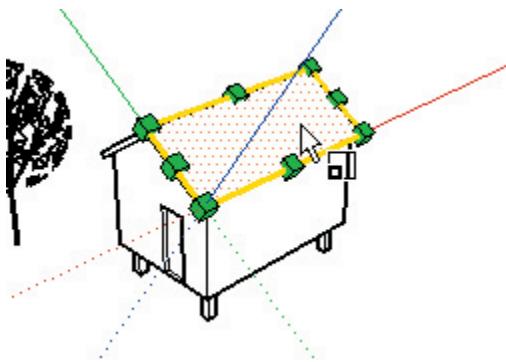
The drawing axes can be manipulated (moved, moved and rotated, and hidden) through their context menu. To move the drawing axes:

1. Context click on the Drawing Axes and select Place from the context menu. The cursor changes to a collection of axes.
2. Move your cursor to the point in your model to be the new coordinate origin. You will see the axes snap to inferred alignments and points as you move around your model.
3. Click to accept the new coordinate origin.
4. Drag your cursor away from the origin to set an alignment for the red axis. Use the inference ToolTips to make sure you are aligned precisely.
5. Click to accept the alignment.



6. Move your cursor away from the origin to set an alignment for the green axis. Use inference ToolTips again to make sure you are aligned precisely.

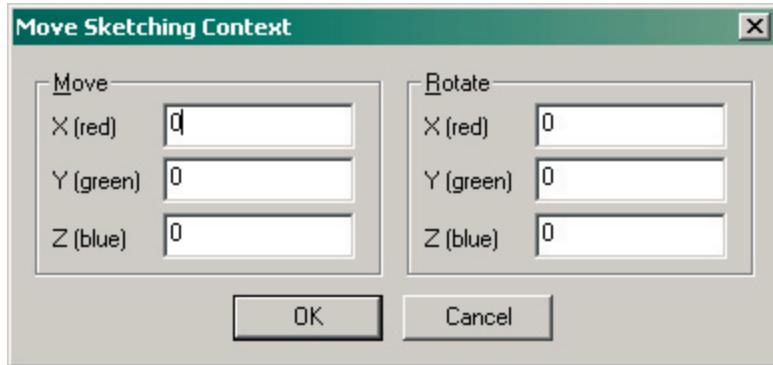
7. Click again to accept the alignment.



## Moving and Rotating the Drawing Axes Relative to Current Position

You can quickly and accurately move and rotate the drawing axes relative to their current position using the Move Sketching Context dialog box. To move and rotate the drawing axes:

1. Right click on the drawing axes. The drawing axes context menu is displayed.
2. Select Move from the context menu. The Move Sketching Context dialog box is displayed.



3. Specify displacement and rotation values in the units specified under the Units panel of the Model Info dialog box.
4. Click OK.

## Aligning the Point of View to the Current View

You can align SketchUp's point of view to be perfectly aligned with the current view. To align the point of view to the current view:

1. Context click on the drawing axes. The drawing axes context menu is displayed.
2. Select Align View from the context menu.

## Hiding the Drawing Axes

You can display or hide the drawing axes from either the View menu or the drawing axes's context menu.

# Inference

SketchUp features a sophisticated geometric analysis engine, called the inference engine, allowing you to work in 3D space using a 2D screen and input device. This engine helps you draw very accurately by inferring points from other points as you draw while also providing you with visual cues.

## Inference Cues

The inference engine uses ToolTip cues, appearing automatically while working on the model, to identify significant points or geometric conditions. These cues make complex inference combinations clear to you as you draw.

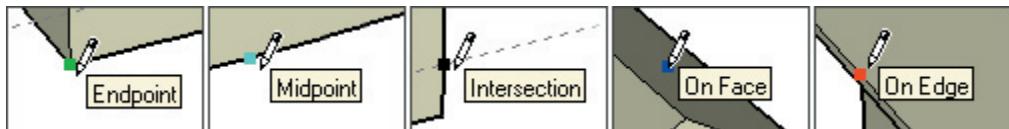
Additionally, the inference engine uses specific colors to indicate its type of inference (covered further in Inference Types).

## Inference Types

There are three main types of inferences: point, linear, and planar. SketchUp often combines inferences together to form a complex inference.

### Point Inference

A point inference is based on an exact point of your cursor in your model.



**Endpoint:** The green Endpoint inference identifies the end of a Line entity or Arc entity.

**Midpoint:** The cyan Midpoint inference indicates the middle point on a line or edge.

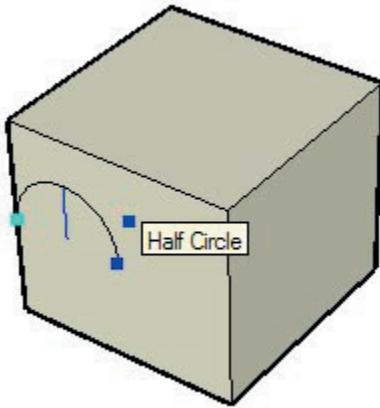
**Intersection:** The black intersection inference indicates an exact point where a line intersects another line or face.

**On Face:** The blue On Face inference identifies a point which lies on a Face Entity.

**On Edge:** The red On Edge inference identifies a point that lies along an edge.

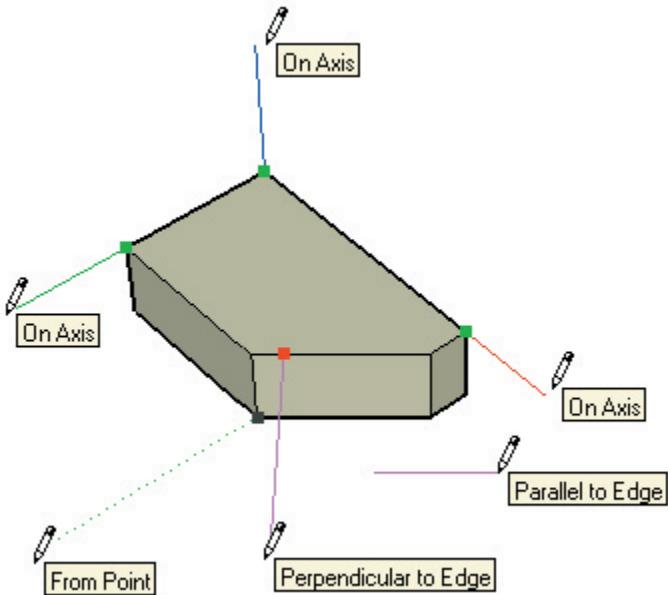
**Equi-Distant On Edge:** The Equi-Distant On Edge inference indicates an equidistant point, or a chamfer, when a magenta line appears between two connected edges.

**Half Circle:** The Half Circle inference appears when drawing an arc to indicate the point that creates an exact half circle.



### Linear Inference

A linear inference snaps along a line or direction in space. In addition to a ToolTip, a linear inference sometimes displays a temporary dotted line while you draw.



**On Axis:** The On Axis inference indicates a linear alignment to one of the drawing axes. The solid line is drawn in the color associated with the corresponding axis (red, green, or blue).

**From Point:** The From Point inference indicates a linear alignment from a point along the Drawing Axes directions. The dotted line is drawn in the color associated with the corresponding axis (red, green, or blue).

**Perpendicular:** Tie Perpendicular magenta line indicates perpendicular alignment to an edge.

**Parallel:** The Parallel magenta line indicates a parallel alignment to an edge.

**Tangent:** When drawing from the endpoint of an Arc entity.

## Planar Inference

A planar inference snaps to a plane in space.

**Drawing Planes:** SketchUp will snap to the planes defined by the Drawing Axes and your view when it cannot snap to geometry in the drawing area. For example, SketchUp will draw on the ground plane when the point of view is that of the ground plane.

**On Face:** A blue On Face inference identifies a point which lies on a face. Although initially a point inference, On Face can also serve as a planar alignment by using inference locking (described below).

## Encouraging an Inference

At times, the inference you need may not come up immediately or SketchUp might choose alignments with the wrong geometry. In these cases, you can increase the chances of a particular alignment by temporarily keeping your mouse cursor stationary over the particular location that you want SketchUp to infer from. When the ToolTips appears, SketchUp will briefly prioritize that alignment as you continue drawing. For example, to encourage SketchUp to create a line to match another, parallel, line (such as when drawing the third line of a rectangle):

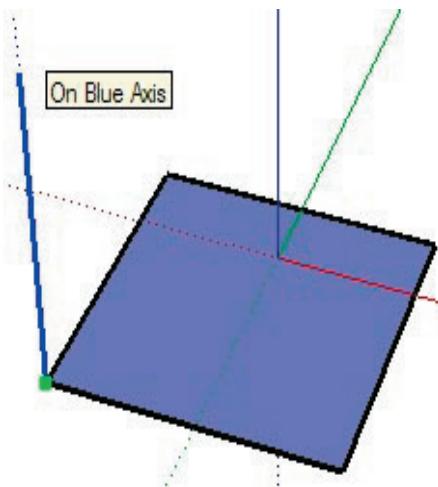
1. Create the first two edges of a rectangle with the first edge being parallel to the red axes.
2. Click on the start point of the third edge and move the line tool as though you were creating the third line parallel to first line. As you begin to move the line tool, the line should appear in red.
3. Mid-line creation, move the line tool over the start point of the start point of the first line you created for the rectangle. Your line will extend diagonally to this point.
4. Leave the tool over this point until the Endpoint ToolTip displays.
5. Move the line tool to the approximate location where the end of the third line should appear. A dotted line, with the color of the corresponding axes (green) will follow the line tool to indicate you are directly inline with the start point of the first line.
6. Click when the line you are drawing is indicated by an axes color (red) and is directly perpendicular to the green dotted line.

## Component Inferences

All normal geometry inferences can be obtained from geometry inside Component entities or Group entities. Group and Component inferences are all indicated by magenta dots.

### Inference Locking

At times, geometry might interfere with your ability to infer points from other points, making it difficult to draw accurately. Use an inference lock, which tells SketchUp not to waver from the direction it is currently inferring from, to solve this problem. To use the inference lock, hold down the Shift key when SketchUp infers the desired alignment (the inference line will bold). The alignment will remain locked, even as you move the mouse and/or pick a secondary inference point. The following image shows the inference locked in the blue direction to ensure that a new line is exactly perpendicular to the face.



Any of the inference conditions may be locked; along an axis direction, along an edge direction, on a face, from a point, parallel or perpendicular to an edge, and so on.

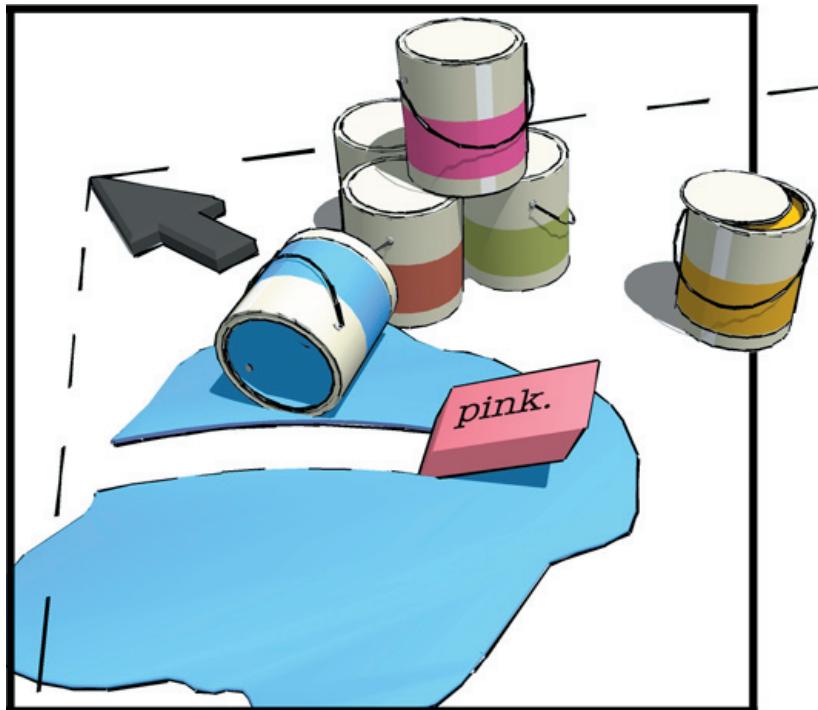


**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

# Principal Tools

This section of the user's guide covers the principal tools within SketchUp. Principal tools are those tools that tend to be used most often in SketchUp.

Tools in this section are covered in the order they appear within the Tools menu.



# Select Tool

The Select Tool is used to specify the entities you will modify when using other tools or commands. The entities that are included in a selection are referred to as the *selection set*. Activate the Select Tool from the Principal toolbar or from the Tools menu.

## Selecting a Single Entity

SketchUp allows you to make both single-entity and multiple-entity selections. To select a single entity:

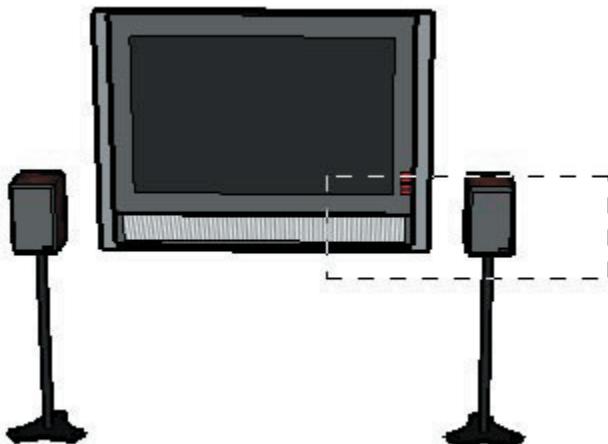
1. Select the Select Tool. The cursor changes to an arrow.
2. Click on an entity. The selected entity is highlighted in yellow.

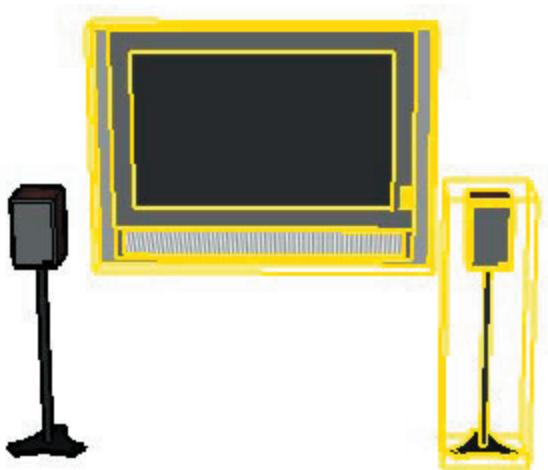
## Selecting Multiple Entities

Multiple-entity selections are useful when you want to perform a single operation on several entities (the selection set). Multiple-entity selections use an expandable *selection box* to surround the entities you want in the selection set. To select multiple entities:

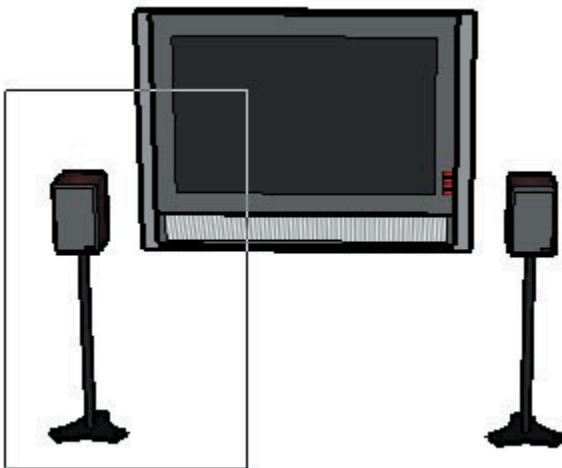
1. Select the Select Tool. The cursor will change to an arrow.
2. Click and hold the mouse button a short distance away from the entities you want to select to start a selection box.

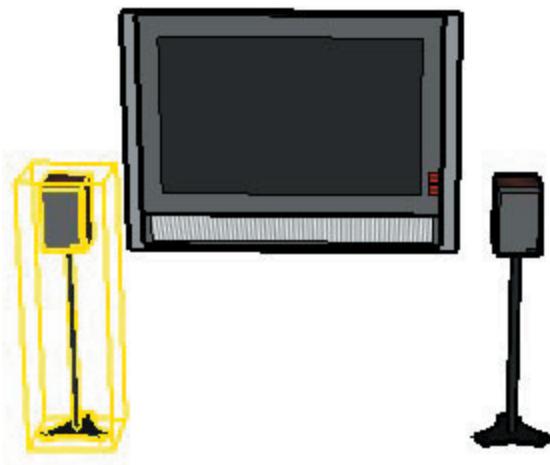
Clicking to the right-side and dragging to the left, called a *crossing selection*, selects any elements within the selection rectangle, including those that are only partially contained in the rectangle. The following images show a right-to-left selection selecting two components, though none are completely within the bounds of the selection box.





Clicking to the left-side of the entities and dragging right, called a *window selection*, selects only those elements completely within the selection rectangle. The following image shows a left-to-right selection selecting one component because only one component (the left speaker) is completely within the bounds of the selection box.





3. Drag the mouse to the opposite corner of the selection starting point.
4. Release the mouse button when all of the elements are either partially included (left-to-right selection) or fully included (right-to-left selection) in the selection box.

## Adding and Removing Entities from the Selection Set

The Select Tool can be used with one or more keyboard modifiers to add or remove entities from a selection set.

### Adding to the Selection Set

Press and hold the Control key (the cursor will change to an arrow with a plus sign) while clicking on additional entities to add entities, one-by-one, to the selection set. Or, press and hold the Shift key (the cursor will change to an plus sign and minus sign) while clicking on additional entities to add entities, one-by-one, to the selection set.

### Changing Selection Status for an Entity

Press and hold the Shift key (The cursor will change to an arrow with plus and minus signs) while clicking on entities to invert the selection status of the entity (currently selected entities will become unselected, unselected entities will be come selected).

## Subtracting from the Selection Set

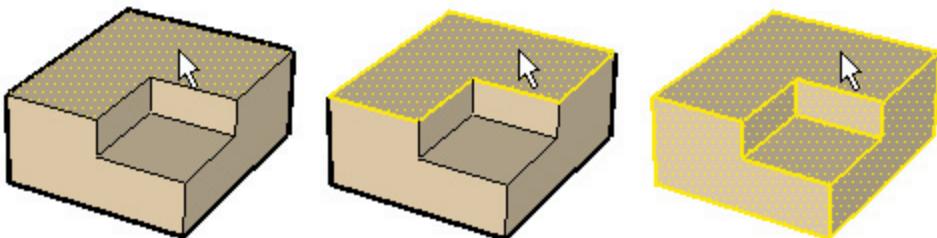
Press and hold the Control and Shift keys simultaneously (the cursor will change to an arrow with a minus sign) while clicking on currently selected entities to remove entities from the selection set. Or, press and hold the Shift key (the cursor will change to an plus sign and minus sign) while clicking on currently selected entities to remove the entities, one-by-one, from the selection set.



**Tip** - Use the Group entity to group items within a selection set as a temporary way of quickly reselecting the same group of items. See the Group entity for additional information.

## Expanding Selection Set

You can automatically add to a selection set clicking the Select Tool multiple times in rapid succession. Click once on an entity to select that entity. Click rapidly twice (double-click) on an entity, namely an edge or face, to select corresponding faces or edges respectively. Click three times (triple-click) on an entity, namely an edge or face, to select the edge or face and all entities physically connected to that edge or face. The following image shows this click/selection sequence.



**Tip** - Use a context-click to invoke the context menu for an entity. Many context menus have a Select submenu allowing you to expand a selection using one of the following commands: Bounding Edges, Connected Faces, All Connected, All on same layer, and All with same material.

## Selecting or De-Selecting All Geometry

Use either the Select All menu item in the Edit menu or press Ctrl+A on the keyboard to select all visible elements in your model.

Use the Deselect All menu item in the Edit model, press Ctrl+T , or click on any empty space in the Drawing Area to de-select all currently selected entities.

# Eraser Tool

The Eraser Tool is used primarily to delete entities, such as Edge entities and Construction Line entities, from the drawing area. The Eraser Tool can also be used to hide and soften edges. Activate the Eraser Tool from the Principal toolbar, or by selecting Eraser from the Tools menu.

## Erasing Entities

As mentioned previously, the Eraser Tool is used to erase entities in the drawing area. Note, however, that the Eraser Tool does not allow you to erase faces. To erase entities:

1. Select the Eraser Tool. The cursor changes to an eraser with a small box.
2. Click on the entity you wish to erase. Alternatively, erase several entities at once by holding down the mouse button and dragging it over several entities to be erased. All selected geometry will be erased once you release the mouse button.

If you accidentally select geometry you do not wish to delete, press the ESC key to cancel the erase operation before it deletes your selection.



**Tip** - Try erasing entities slowly if you continuously skip over entities you want to erase.



**Tip** - It is usually faster to erase a large number of entities by selecting the entities with the Select Tool and pressing the Delete key on your keyboard. You can also delete selected items by selecting Erase from the Edit menu.

## Hiding Edges

Press and hold the Shift key and use the Eraser Tool to hide entities (instead of erasing entities).

## Soften Edges

Press and hold the Ctrl key to soften/smooth edges (instead of erasing entities). Press and hold the Ctrl and Shift keys simultaneously to unsoften/unsmooth edges.

# Paint Bucket Tool

The Paint Bucket Tool is used to assign materials (and colors) to entities in your model. You can use it to paint individual entities, fill a number of connected faces, or replace a material with another throughout your model. Activate the Paint Bucket Tool from the Principal toolbar, or by choosing Paint Bucket from the Tools menu.



**Video** - Various video tutorials on this topic are available at  
<http://www.sketchup.com/training/tutorials.php>

## Applying Materials

Ensure you are using either the Shaded or Shaded with Textures display setting so that you can see materials as they are applied to your model (Window > Display Settings > Shaded or Shaded with Textures). To apply materials:

1. Select the Paint Bucket Tool. The cursor will change to a paint bucket and the Material Browser is activated. The Materials Browser contains libraries of materials you can paint on faces in your model.
2. Select a materials library using the drop down list box. SketchUp contains several default materials libraries, including landscape, roofing, and transparent materials.
3. Select a material from the library of materials.
4. Click on the faces you want to paint. The faces will receive the material.
5. If you select multiple elements using the Select Tool, clicking on the selection with the Paint Tool will paint all of them with a single click.

## Face Painting Rules

There are several face painting rules that apply when painting multiple faces or edges at the same time. These rules follow:

- The side of the faces that will be painted depends on the side initially painted when more than one face is selected. For example, if all faces are selected and you paint the front of one face, the front of all faces will be painted. Conversely, if all faces are selected and you paint the back of one face, all back faces will be painted.
- All selected edges will be painted when you select a face and all edges and paint the front of the face. No edges are painted when you select a face and all edges and paint the back of one face. Ensure you have the Window > Display Settings > Edge color set to By Material to see the painted effect applied to edges.

## Paint Bucket Tool Modifier Keys

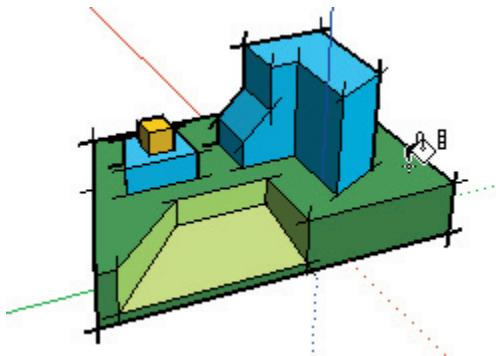
The Paint Bucket Tool can be used to quickly assign materials to many faces at once by using the Ctrl, Shift, and Alt modifier keys.

### Element Fill (No Modifier)

The Paint Bucket Tool normally operates by filling in faces as you click on them. As mentioned previously, entities selected with the Select Tool can be painted with a single click of the Paint Bucket Tool.

### Adjacent Fill (CTRL)

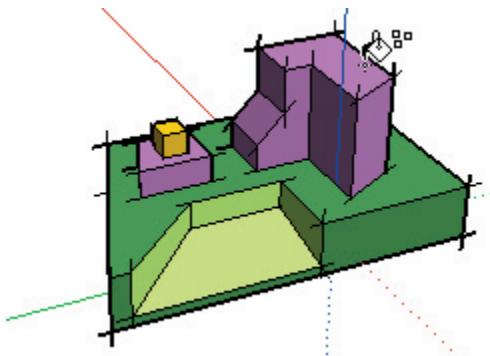
Press and hold the CTRL key while clicking on a face with the Paint Bucket Tool to fill that face and any adjacent (connected) face with the same material. The face you click on and the adjacent face must have the same material prior to performing this operation.



**Note** - Selecting a number of entities with the Select Tool and painting using the CTRL modifier keys causes just the entities within the selection set to be painted.

### Replace (SHIFT)

Press and hold the Shift key prior to clicking on a face with the Paint Bucket Tool to apply a material to every face, with the same material in the current context, with the new material.



**Note -** Selecting a number of entities with the Select Tool and painting using the Shift modifier key causes just the entities within the selection set to be painted.

### Adjacent Replace (CTRL+SHIFT)

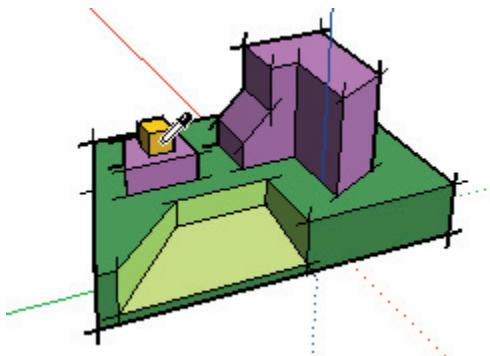
Press and hold both the CTRL and the Shift keys simultaneously while painting to only replace the material on the face within the confines of geometry that is physically connected to that face.



**Note -** Selecting a number of entities with the Select Tool and painting using the CTRL modifier keys causes just the entities within the selection set to be painted.

### Sample Material (ALT)

Press and hold the Alt key to change from the Paint Bucket Tool to a Sample Tool for sampling materials within your model. The cursor will change to an eye dropper. Click on the face whose material you want to sample. Release the Alt key to return to the Paint Bucket Tool. Paint the sampled material on a face.



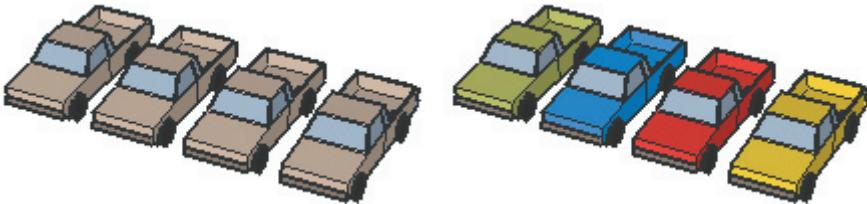
## Painting Groups and Components

Materials can be painted on entire Group entities or Component entities or to the individual entities within the Group or Component. To assign materials to an entire Group or Component:

1. Select the Paint Bucket Tool. The cursor will change to a paint bucket and the Material Browser is activated. The Materials Browser contains libraries of materials you can paint on faces in your model.
2. Select a materials library using the drop down list box. SketchUp contains several default materials libraries, including landscape, roofing, and transparent materials.
3. Select a material from the library of materials.
4. Click on the Group or Component you want to paint. The faces will receive the material.
5. If you select multiple Groups or Components using the Select Tool, clicking on the selection with the Paint Tool will paint all of them with a single click.



**Note -** *If a face within a Group or Component is already painted with a material (other than the default material), before applying a material to the entire Group or Component, the face will not adopt the new material. For example, the windshields, bumpers, and tires in the following image were already painted before a material was applied to these components. Therefore, the windshields, bumpers, and tires maintained their original material.*

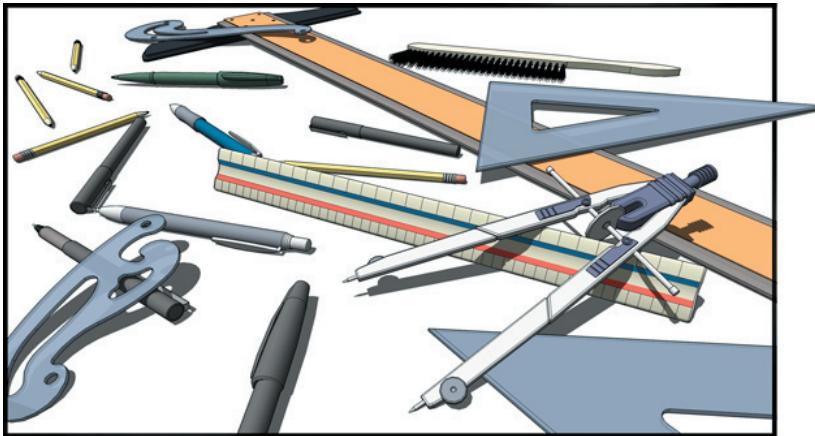


**Note -** *Exploding a Group or Component assigns the object materials to any elements assigned the default material, thus making the material override permanent.*

# Drawing Tools

This section of the user's guide covers the drawing tools within SketchUp. Drawing tools are those tools used to create new geometry. These tools include the Line Tool, Rectangle Tool, and Freehand Tool.

Tools in this section are covered in the order they appear within the Draw menu.



# Line Tool

The Line Tool is used to draw lines or Edge entities, multiple connected edges, or closed shapes. The Line Tool can also be used to divide faces or heal deleted faces. Activate the Line Tool from either the Drawing toolbar or the Draw menu.

## Drawing a Line

Lines can be placed on existing faces or separate from existing geometry (aligned to an axes plane). To draw a Line:

1. Select the Line Tool. The cursor changes to a pencil.
2. Click to set the starting point of your line.



**Note -** Press the *ESC* key at any point during the operation to start over.

3. Move the mouse in the direction you want to make a line. As you draw a line, the length is displayed dynamically in the Value Control Box (VCB).

Length 4' 2 5/8"

4. Click a second time on the endpoint of your line. This ending point can also be the starting point of another line.



The line length can be specified precisely using the VCB either before clicking the second point or immediately after the line has been drawn. See Specifying Precise Line Values for further information on using the VCB with Line entities.

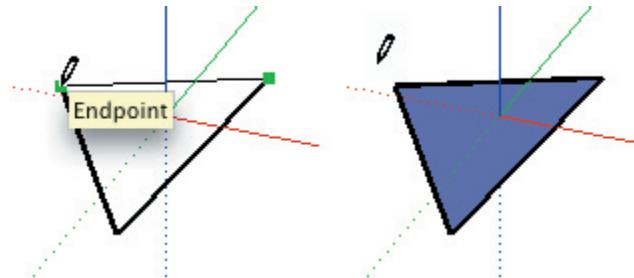


**Tip -** Alternately, you can click and hold the mouse button to set the starting point of the line, and drag outward without releasing the button to set the length. Release the mouse button to complete the line. See the Drawing panel of the Preferences dialog box for further information on setting drawing behavior.

## Creating a Face

The Line Tool will remain at the ending point of every line you create. This end point is automatically treated as a starting point for another line. You can create another line by moving the cursor and clicking again from this starting point. These two lines are said to be coplanar, intersecting lines (intersecting at starting and ending points).

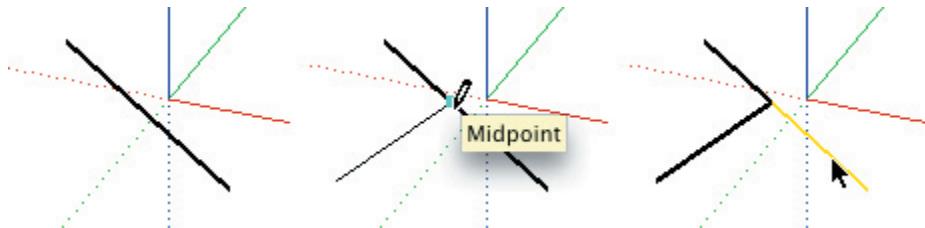
Three or more coplanar lines, intersecting at their ending and starting points (forming a loop), generate a Face entity. Ensure that the Endpoint inference ToolTip is visible whenever you close a Face entity to ensure that any lines you draw are continuous. The Line Tool is released but is still active after a Face has been created.



**Tip** - Set the display settings (Window > Display Settings) to the Shaded rendering style to clearly show new faces as they are created.

## Splitting a Line

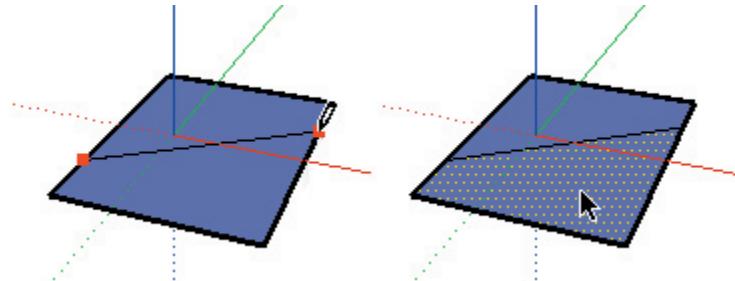
SketchUp automatically splits line segments when new lines are drawn perpendicular to a line. For example, draw a new line to the midpoint (identified by a cyan square) of another line to split a line in half. The following example shows one line being intersected at the midpoint, resulting in two lines.



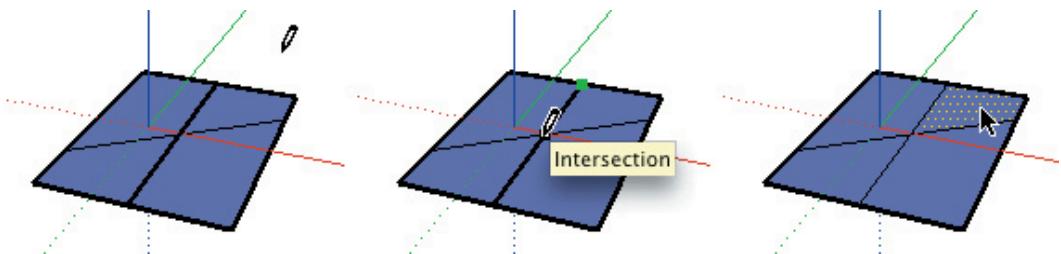
You will see that it has been split into two equal segment when you select the original line again.

## Splitting a Face

Draw a line with starting and ending points on the face's edges to split a face. The following image shows a rectangle being split when a line is drawn from one edge of a face to another opposite edge.



Occasionally, overlapping lines will not be split. Any lines that are not part of a face perimeter will be displayed with a thicker line (when the Profile Edge rendering style is enabled in the Display Settings dialog box). Use the Line Tool to trace along an existing line to attempt to split overlapping lines. SketchUp will re-analyze your geometry and attempt to re-integrate the line. The following images show a bold line that did not effectively split a face.



Tracing over the bolded line yielded an integrated line that effectively split the face.

## Specifying Precision Line Values

The VCB displays the length of your line while you are drawing lines. You can also specify an line length value using the VCB.

### Entering a Length Value

Specify a line length in the VCB by typing in the desired length after placing the starting point of the line and pressing the Enter or Return key. SketchUp will use the current document units setting if you only type in a numerical value. You can also specify either Imperial (**1'6"**) or Metric (**3.652m**) units at any time, regardless the model units setting.

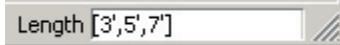


**Note -** The Line Tool will snap to any previously entered length within the VCB.

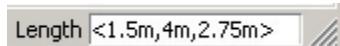
## Entering a 3D Coordinate

The VCB can also be used to place the end of the line at an exact coordinate in space.

**Entering an Absolute Coordinate:** Type in the coordinates of a point in 3D space enclosed by brackets, such as [x, y, z], to get absolute coordinates relative to the current axes.



**Entering a Relative Coordinate:** Specify coordinate points relative to the start point of your line by using the less-than and greater-than symbols, such as <x, y, z>, where x, y, and z values are relative distances from the start point of your line.



**Note -** The exact format for VCB entries will vary depending on your computer's Regional Settings. For European users, the list separator symbol may be a semi-colon instead of a comma, so the format would be [x; y; z].

## Drawing Lines by Inference

The Line Tool uses SketchUp's sophisticated geometric inference engine to help you place your lines in 3D space. The inference decisions, made by the inference engine, are displayed in the drawing area as inference lines and inference points. These lines and points show precise alignment between the line you are drawing and the geometry of your model. Refer to the inference engine topic for additional information.

## Dividing a Line into Equal Segments

Line segments can be divided into any number of equal line segments. To divide a line into equal segments:

1. Right click on an edge
2. Select Divide from the context menu. SketchUp will place points on the line to show where the line will be divided.
3. Move your mouse toward the center of the line to reduce the number of divisions. Move your mouse toward either end of the line to increase the number of divisions.
4. Click on the line when the number of divisions you would like is shown. The line will be divided into an equal number of joined line segments.

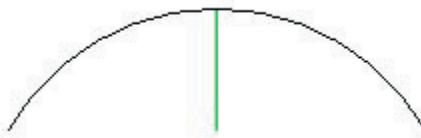
# Arc Tool

The Arc Tool is used to draw Arc entities, comprised of multiple straight line segments (which can be edited as a single arc). Activate the Arc Tool from the Drawing toolbar or from the Draw menu.

## Drawing an Arc

Arc entities consist of three parts: the starting point, the ending point and the bulge. The distance between the starting point and the ending point is also known as the *chord length*. To draw an arc:

1. Select the Arc Tool. The cursor changes to a pencil with an arc.
2. Click to place the starting point of your arc.
3. Move the mouse.
4. Click again to place the ending point of your arc. A straight line is created.
5. Move your mouse perpendicular to the straight line to adjust the bulge distance. Or, optionally type in values for the chord length, bulge distance, radius, and number of segments in the VCB. A straight line will extend perpendicular from the straight line. See Specifying Precise Arc Values for more information on manipulating arcs using the VCB.

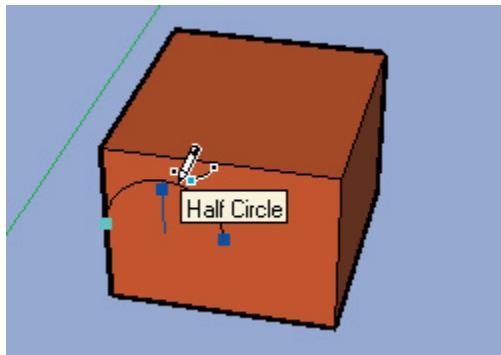


**Note** - Press the *ESC* key at any point during the operation to start over.

6. Click again to set the bulge distance.

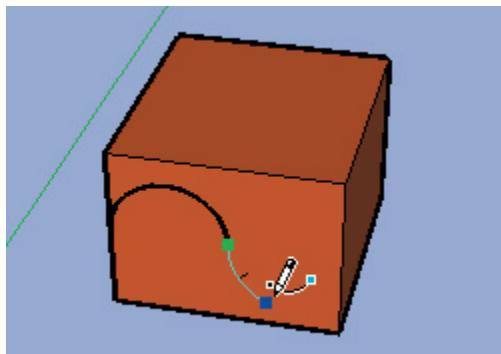
## Drawing a Half-Circle

As you pull out a bulge distance, the arc will temporarily snap to a half-circle. Watch for the Half-Circle inference ToolTip indicating when your arc is a half-circle.



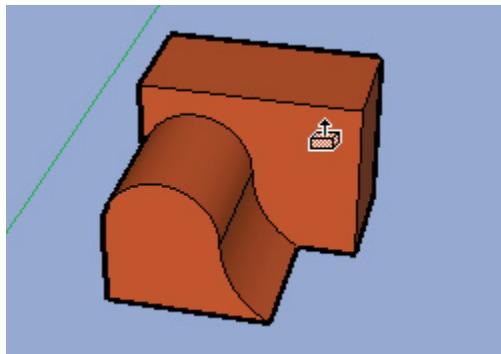
## Drawing Tangentially

The Arc Tool will display a cyan tangent arc while you are drawing from an unconnected end or start point of an existing arc. Keep the mouse still after setting the second point and then click a third time if you want to create a tangent arc. Or, move the mouse to break the tangent inference and set your own bulge distance after clicking the second point.



## Pushing and Pulling a Curved Face

You can use the Push/Pull Tool on faces that have an arc as an edge similarly to using the Push/Pull Tool on regular faces. The curved face that results from the push/pull operation is called a Surface entity. Surfaces can be adjusted as a whole, but are comprised of a number of faces or a *curved face set*.



**Tip** - Select View > Hidden Geometry to view and manipulate the individual faces in the surface.

## Specifying Precise Arc Values

The VCB displays the chord length of the arc (after setting the starting point), then the bulge distance (after setting the end point). The VCB can be used to enter exact lengths for the base chord, bulge distance, radius value, and number of segments.



**Note** - SketchUp will use the current file units setting if you only type in a numerical value. You can also specify either Imperial (1'6") or Metric (3.652m) units at any time, regardless the file's units setting. Units are set within the Units panel of the Model Info dialog box.

## Entering a Chord Length

To specify a value for the chord length, after placing the starting point of the arc, type the desired length in the VCB. To indicate that you want the length to apply in an opposite direction to the current drawing direction, specify a negative value, such as -6.5".



**Note** - The Arc Tool will snap to any previously entered chord length within the VCB.



## Specifying a Bulge Distance

You can also specify an exact bulge distance or the radius for your arc after entering the chord length. The VCB's label will change from a Length to Bulge.

Type the bulge length in the VCB, after setting an ending point, and press the Enter or Return Key. You can enter the bulge distance either during the creation of the arc or after, as long as the bulge length is displayed in the VCB. Negative bulge values can also be used to create an arc in a direction opposite the one indicated while drawing.

## Specifying a Radius Value

You can specify an arc radius instead of a bulge distance. Type the desired radius followed by the letter 'r', (For example: 24r or 3'6"r or 5mr) and press the Enter or Return Key. You can perform this action either during or immediately following the creation of the arc.

## Specifying The Number of Segments

You can also specify the number of arc segments in the Arc entity by typing in the desired number followed by the letter 's' in the VCB. Press the Enter or Return key after you have entered a arc segment number. You can perform this action either during or immediately following the creation of the arc.

# Freehand Tool

The Freehand Tool allows you to draw irregular, coplanar connected lines in the form of Curve entities and 3D Polyline entities. Curve entities are comprised of multiple line segments that are connected together. These curves behave as a single line in that they can define and divide faces. They are also connected such that selecting one segment selects the entire entity. Curve entities can be useful for representing contours in a *contour map* or other *organic shapes*. Activate the Freehand Tool from the Draw menu or from the Drawing toolbar.

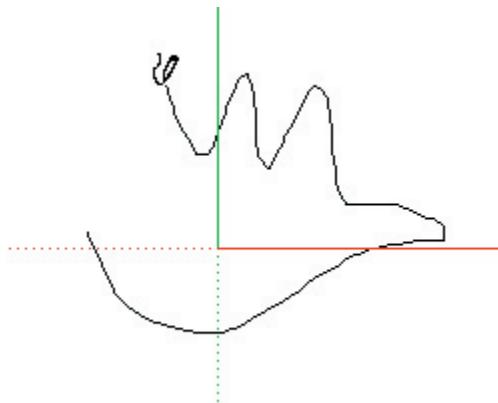


**Note -** 3D Polyline entities do not generate inference snaps or affect geometry in any way. These entities are primarily to add non-geometrical detail to your model.

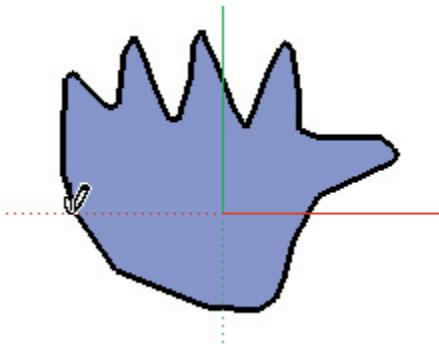
## Drawing Curves

Curves can be placed on existing faces or separate from existing geometry (aligned to an axes plane). To draw a Curve:

1. Select the Freehand Tool. The cursor will change to a pencil with a curve.
2. Click and hold to place the starting point of your curve.
3. Drag the cursor to draw.
4. Release the mouse button to stop drawing.



5. End your line at point where you started drawing to draw a closed shape with the Freehand Tool.



## Drawing a 3D Polyline

3D polylines do not generate inference snaps or affect geometry in any way. 3D polylines are usually used for tracing imported drawings, 2D sketching, or for decorating your model.

3D polylines can be placed on existing faces or alone out in drawing space. To draw a 3D polyline entity:

1. Select the Freehand Tool. The cursor will change to a pencil with a curve.
2. Hold down the Shift key.
3. Click and hold to place the starting point of your freehand sketch.
4. Drag the cursor to draw.
5. Release the mouse button to stop drawing.



**Note** - Select *Explode* from the 3D polyline's context menu to convert a Freehand Sketch into regular edge geometry.

# Rectangle Tool

The Rectangle Tool is used to draw rectangular Face entities, specified by clicking at two opposite corners of the desired shape. Activate the Rectangle Tool from either the Drawing toolbar or the Draw menu.

## Drawing a Rectangle

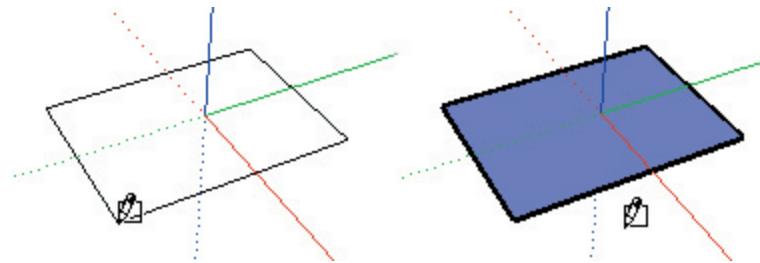
Rectangles can be placed on existing faces or separate from existing geometry (aligned to an axes plane). To draw a rectangle:

1. Select the Rectangle Tool. The cursor changes to a pencil with a rectangle.
2. Click to set the first corner point of the rectangle.
3. Move your mouse to the opposite corner.



**Note -** Press the *ESC* key at any point during the operation to start over.

4. Click again to set the second corner point of the rectangle..



## Drawing a Square

Squares are created with the rectangle tool in conjunction with the Square ToolTip To draw a Square:

1. Select the Rectangle Tool and Click once on the first corner point.
2. Move your mouse to the opposite corner. A diagonal dotted line will appear, along with a Square ToolTip when you are in a position that will create a square.



**Note -** Press the *ESC* key at any point during the operation to start over.

3. Click again to finish.



**Tip** - A dotted line and Golden Section ToolTip appears when you are in a position to create a Golden Section.

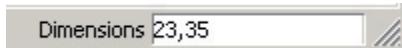
Alternately, you can press your mouse button on the first corner of your rectangle, drag to the opposite corner, and release the mouse button.



**Tip** - Use the Axes tool to re-align the axes, prior to drawing your rectangle, if you want to draw a rectangle that is not aligned with the default drawing axes orientation.

## Entering Precise Dimensions

A rectangle's dimensions dynamically appear in the VCB as you draw. You can specify exact dimensions by typing them in the VCB either after the first corner is clicked, or immediately after the rectangle is drawn.



SketchUp will use the current document units setting if you only type in a numerical value. You can also specify either Imperial (such as 1'6") or Metric (such as 3.652m) units at any time, regardless the document units setting.

You can also type one dimension in the VCB at a time. If you enter a value and a comma, such as 3', the new value will be applied to the first dimension, and the second dimension will be retained from before. Similarly, if you type a comma and then a dimension, such as 3', only the second dimension will be changed.

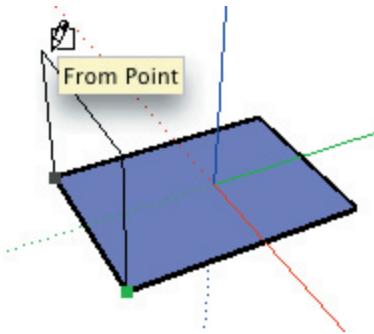


**Tip** - If you enter a negative value (-24, -24), SketchUp will apply that value in a direction opposite to the one you indicated while drawing and accept any new values in the new direction.

## Drawing Rectangles by Inference

The Rectangle Tool uses SketchUp's sophisticated geometric inference engine to help you place your rectangles in 3D space. The inference decisions, made by the inference engine, are displayed in the drawing area as inference lines and inference points. These lines and points show precise alignment between the rectangle you are drawing and the geometry of your model.

For example, if you move your mouse over an endpoint of an existing edge and then move away in an axial direction, a dotted Inference Line with a From Point ToolTip will appear.



This ToolTip indicates that you are aligned to that end point. You can also use a From Point inference to draw rectangles vertically or at non-orthogonal planes.

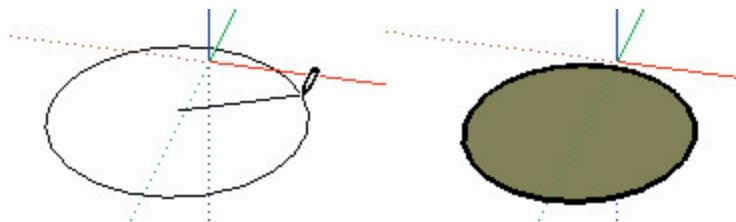
# Circle Tool

The Circle Tool is used to draw Circle entities. Activate the Circle Tool from the Drawing toolbar or the Draw menu.

## Drawing a Circle

Circles can be placed on existing faces or separate from existing geometry (aligned to an axes plane). To draw a circle:

1. Select the Circle Tool. The cursor changes to a pencil with a circle.
2. Click to place the center point of the circle.
3. Move the mouse out from the center point to define the radius of your circle. As you move the mouse, the radius value is displayed dynamically in the VCB and can be specified by typing in a length value followed by the Enter or Return key. You can also specify the segmentation for the circle in the VCB.



**Note** - Press the *ESC* key at any point during the operation to start over.

4. Click a second time to finish the circle.



**Tip** - You can also click and hold the mouse button to set the center of the circle, and drag outward without releasing the button to set the radius. Release the mouse button to complete the circle.

Radius and segment values can be specified using the VCB immediately after a circle is drawn. See Specifying Precise Circle Values for more information on setting the radius and segment values with the VCB.

## Inference Locking

Press and hold the Shift key while in the Circle Tool to lock the circle's orientation.

## Specifying Precise Circle Values

The VCB displays the radius while creating a circle. You can also specify an exact radius or number of line segments in the circle using the VCB.

### Specifying an Exact Radius Value

Specify the radius in the VCB by typing in the desired length after placing the circle's center point. Press Enter or Return to accept the radius value and create the circle.

You can also re-specify a radius immediately (before performing any additional tasks) after the circle has been created simply by typing a radius value in the VCB and pressing Enter or Return.



**Note -** *The Circle Tool will snap to any previously entered radius within the VCB.*

### Specifying The Number of Segments

The VCB displays the number of segments as Sides when the Circle Tool is initially activated. Specify the number of sides in the VCB before clicking to set the center point of the circle by typing a value in the VCB. For example:



You can also re-specify a segment value immediately (before performing any additional tasks) after the circle has been created simply by typing a segment value, followed by an 's' in the VCB and pressing Enter or Return. Segmentation values will now apply retroactively to the last drawn circle until you begin drawing a new circle or change tools. Any values you specify after a circle is drawn will apply to the last circle drawn, and will set the tool to that value as well for future circles.

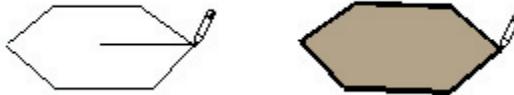
# Polygon Tool

The Polygon Tool is used to draw regular Polygon entities, inscribed within a circle, with 3 to 100 sides. Activate the Polygon Tool from either the Drawing Toolbar, or by selecting Polygon from the Draw menu.

## Drawing a Polygon

Polygons can be placed on existing faces or separate from existing geometry (aligned to an axes plane). To draw a polygon:

1. Select the Polygon Tool. The cursor changes to a pencil with a polygon.
2. Click to place the center point of the polygon. You can also specify the number of sides for the polygon in the VCB.
3. Move the mouse out from the center point to define the radius of your polygon. The radius value is displayed dynamically in the VCB as you move the mouse. You can specify a new radius by typing in a length value followed by the Enter or Return key.



**Note - Press the ESC key at any point during the operation to start over.**

4. Click the left mouse button a second time to finish the polygon. (Alternately, you can click once to set the center of the polygon, and drag outward without releasing the button to set the radius. Release the mouse button to complete the polygon.)

The radius and segment values can still be specified using the VCB immediately after a polygon is drawn.

### Inference Locking

Press and hold the Shift key while in the Polygon Tool to lock the polygon's orientation.

## Entering Exact Radius and Segment Values

The VCB, at the bottom right corner of the SketchUp window, displays the number of sides and the radius of polygon in the document Units panel of the Model Info dialog box while you are drawing polygons. Specify different values simply by typing them in the VCB and pressing Enter or Return.

## Entering the Number of Sides

The VCB input is designed to accept a number polygon sides when you first activate the Polygon Tool. The VCB will accept radius input, as described below, after you place your first shape. However, you can still enter the number of sides by typing in the desired number followed by the letter 's'. For example, '8s' will yield an octagon. Any values you specify after a polygon is drawn will apply to the last polygon drawn, and will also set the tool to that value.

## Entering a Radius Value

You can use the VCB to enter an exact radius for your polygon after you have set the center point of your polygon. Type the radius length into the VCB, followed by the Enter or Return key, to enter an exact radius. You can change this setting either during or following the creation of a polygon.

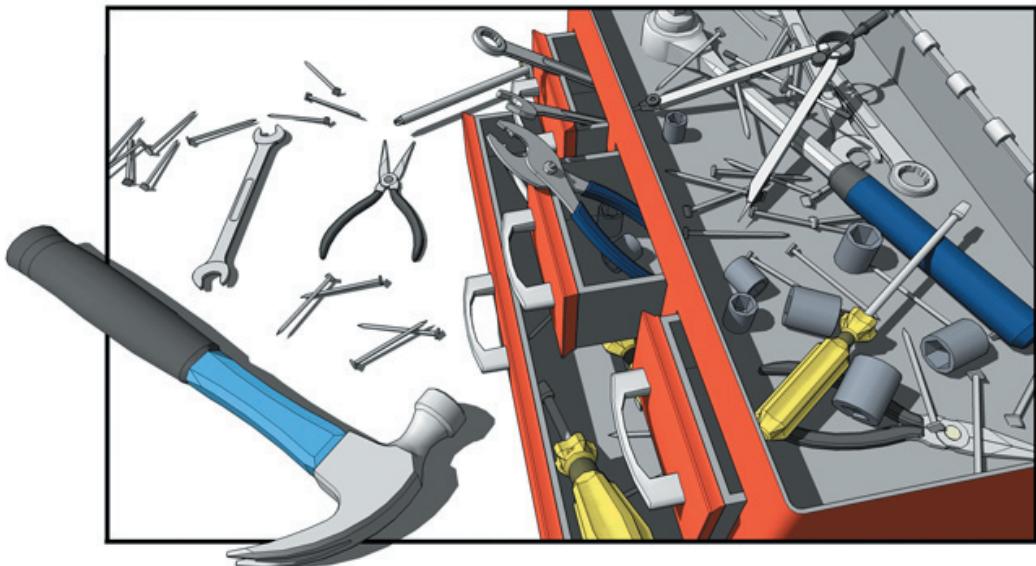


**Note -** *The Polygon Tool will snap to any previously entered radius within the VCB.*

# Modification Tools

This section of the user's guide covers the modification tools within SketchUp. Modification tools are those tools used to modify existing geometry. These tools include the Push/Pull Tool, Eraser Tool, and Select Tool.

Tools in this section are covered in the order they appear within the Tools menu.



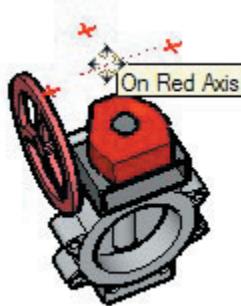
# Move Tool

The Move Tool allows you to move, stretch and copy geometry. This tool can also be used to rotate components and groups. Activate the Move Tool using the Modification toolbar or the Tools menu.

## Moving Geometry

You do not need to click on the entities to move the entities. As long as the entities you want to move are selected, the entities will follow the cursor (even if the cursor isn't over the entities). To move entities using the Move Tool:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be moved.
3. Select the Move Tool. The cursor will change to an four-way arrow.
4. Click once on an entity to begin the move operation. The point where you click on the entity is called the *move point*.
5. Move your mouse to move the entities. The selected entities will follow as you move your mouse. Also, an inference line will appear between the start and ending points of the move, and the distance of the move is displayed dynamically in the VCB. You can also type in a specific distance as described below. The following image shows a component being moved:



**Tip** - Follow inference lines to easily align items in 3D space. Select corners of components as your move point and align to corners of other components when aligning components side-by-side (such as when aligning kitchen cabinet components).

6. Click at the destination point to finish your move operation.



**Note** - Press the *ESC* key at any point during the operation to start over.

## Select and Move

You can activate the Move Tool when nothing is selected to select a single entity to move. The selection click point becomes the base point for the move operation. To select and move a single entity:

1. Select the Move Tool. The cursor will change to a four-way arrow.
2. Click on the entity to move.
3. Move your mouse to move the entity. The selected entity will follow as you move your mouse.
4. Click at the destination point to finish your move operation.



**Note -** If the entity you are moving is connected to other entities, the other entities will be moved or modified accordingly. See the Stretching Geometry section for further information on this behavior.

### Inference Locking During Move

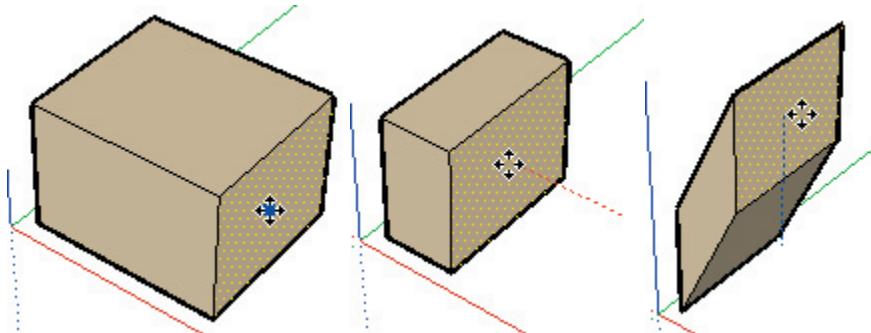
You can perform an Inference Lock by holding down the Shift key before or during a Move Tool operation. Inference locking keeps the inference engine from being confused by other geometry in your model.

### Moving Groups and Components

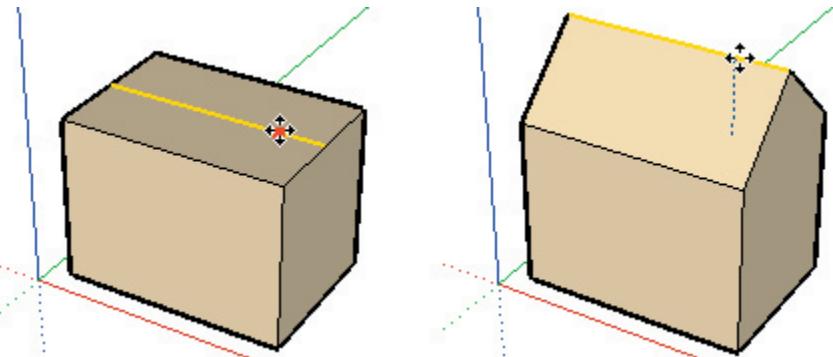
If a component is glued to a face, the component will stay in the plane of that face when moved unless it is unglued. Copies of a glued component will also be glued to the originating plane.

## Stretching Geometry

When you move an element that is interconnected with others, SketchUp will stretch geometry as necessary. You can move points, edges, and faces in this manner. For example, the following Face entity can be moved back in the negative red direction or up in the positive blue direction:

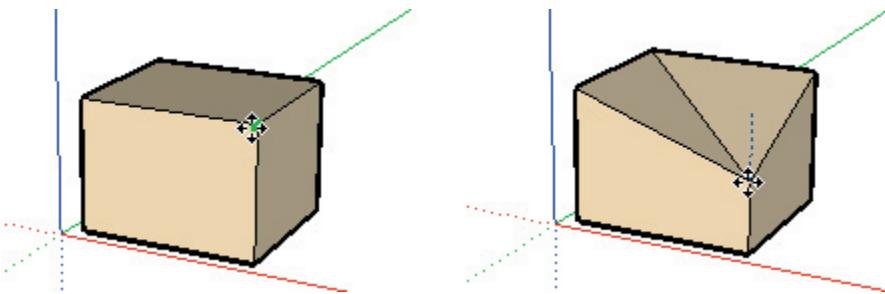


You can also move single line segments to stretch an object. In the following example, a line is selected and moved up in the blue direction to form a sloped roof.



## Moving/Stretching With Autofold

SketchUp will Autofold faces automatically when a move or stretch operation will create non-planar faces. For example, clicking on the corner of a box with the move tool and move down in the blue direction causes SketchUp to create a fold line along the box's top face..

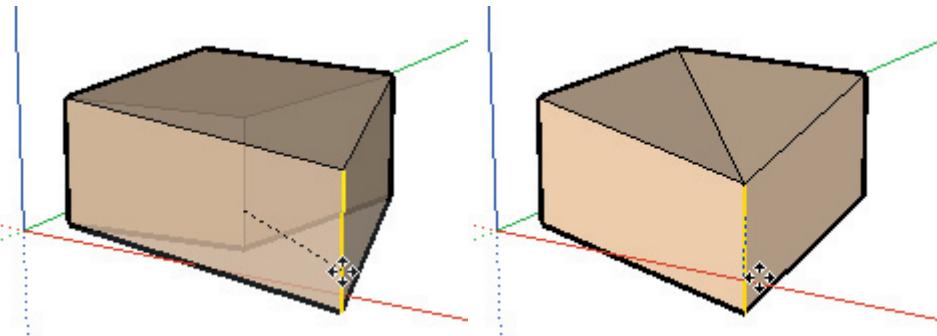


**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Forcing Autofold Behavior

There are times when SketchUp constrains an operation in favor of keeping all faces planar and not creating additional fold lines. For example, clicking on the edge of a box with the move tool only allows you to move the edge in a horizontal direction (red and green), but not vertically (blue).

You can override this behavior by holding down the Alt key before performing the move operation. This key sequence enables Autofold allowing geometry to move freely in any direction.



## Making Copies

As mentioned previously, the Move Tool can be used to make copies of entities within your model. To make copies of an entity using the Move Tool:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be copied.
3. Select the Move Tool.
4. Press and hold the Ctrl key on your keyboard. This action informs Sketchup that you want to duplicate the selected entities.
5. Click on the selected entities to copy.
6. Move your mouse to copy the entities. A copy of the selected entities will follow as you move your mouse.
7. Click at the destination point to finish your copy operation. The copied entities are now selected and the original entities are deselected.

## Creating Multiple Copies (Linear Arrays)

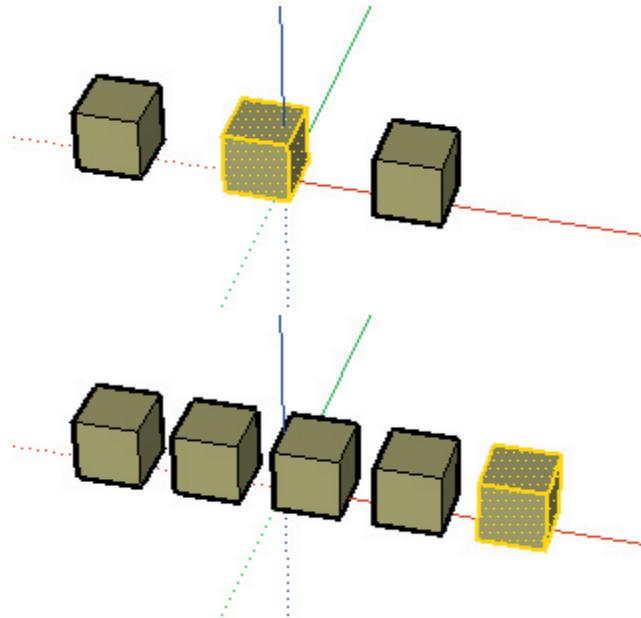
The Move Tool can also be used to create arrays, or a series of copies of geometry. To create multiple copies of one or more entities:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be copied.
3. Select the Move Tool.
4. Press and hold the Ctrl key on your keyboard. This action informs SketchUp that you want to duplicate the selected entities.
5. Click on the selected entities to copy.

6. Move your mouse to copy the entities. A copy of the selected entities will follow as you move your mouse.
7. Click at the destination point to finish your copy operation. The copied entities are now selected and the original entities are deselected.
8. Type a multiplier value to create additional multiple copies. For example, typing in 2x (or \*2) will create one additional copy (or 2 copies total, the one you manually copied plus one you automatically copied using this step) instead of just one.
9. See the Multi-Copy topic for additional information on this feature.

### Creating Copies at an Equal Distance Apart

You can divide the distance between the copy and the original by typing in a divisor value. For example, typing 5/ (or /5) will create five copies evenly distributed between the original and the first copy. You can keep typing in distances and multipliers until you perform another operation.



**Tip** - This feature is particularly useful in creating models of items such as fences, bridges, and decks, where you might want several posts or beams an equal distance apart.

## Entering Exact Move Values

The VCB at the bottom right corner of the SketchUp window displays the length of the move operation (displacement) in the default units, as specified under the Units panel of the Model Info dialog box, while moving, copying, or stretching entities. In addition to creating arrays, you can also specify an exact displacement or a relative or absolute 3D coordinate for the finishing point during, or immediately after, a move operation.

### Entering a Displacement Value

You can specify a new displacement length during or directly following a move operation. To enter a displacement value during a move operation:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be moved.
3. Select the Move Tool. The cursor will change to a four-way arrow.
4. Click once to select the start point of the move operation.
5. Move your mouse to begin moving the entities in the correct direction. The selected entities will follow as you move your mouse. Also, an inference line will appear between the start and ending points of the move, and the distance of the move is displayed dynamically in the VCB.
6. Type the positive or negative displacement value (such as 20' or -35mm) in the VCB and press Enter or Return.



**Note** - You can type values in the VCB using an alternate measuring system than the default system. SketchUp will convert the value to the default system. For example, you can type in 3' 6" even if you are using metric system as your default.

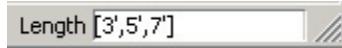
### Entering a 3D Coordinate

SketchUp can move your entities to exact (using []) or relative (using <>) coordinates in 3D space. To enter a 3D coordinate during a move operation:

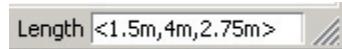
1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be moved.
3. Select the Move Tool. The cursor will change to a four-way arrow.
4. Click once to select the start point of the move operation.
5. Move your mouse to begin moving the entities in the correct direction. The selected entities will follow as you move your mouse. Also, an inference line will appear between the start and ending points of the move, and the distance of the move is displayed dynamically in the Value Control Box (VCB).

6. Type the exact or relative coordinate.

Global Coordinates: [x, y, z] of the current Sketch Axes:



Relative Coordinates: <x, y, z> relative to the start point:



**Note** - You can define only one or two values as part of your 3D coordinate. For example, to move geometry to 2 feet in the z or blue direction enter the following in the VCB: [,2']



**Note** - The exact format for values typed in the VCB will vary depending on your computer's regional settings. For some European users, the list separator symbol is a semi-colon instead of a comma. For example, [x; y; z]

# Rotate Tool

The Rotate Tool is used to rotate drawing elements and single or multiple objects within a single rotation plane. The Rotate Tool can also be used to stretch and distort geometry by selecting only a portion of an object. Activate the Rotate Tool from either the Modification toolbar or the Tools menu.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Rotating Geometry

You can rotate geometry in three different planes in a 3D environment. To rotate geometry using the Rotate Tool:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the geometry to rotate.
3. Select the Rotate Tool. The cursor will change to a protractor with a circular arrow.
4. Move the protractor around to align to the proper orientation in your model. The Protractor Tool will shift its orientation as it aligns with nearby axes and geometry. You can lock in a particular orientation by holding down the Shift key when the protractor is in your desired orientation.

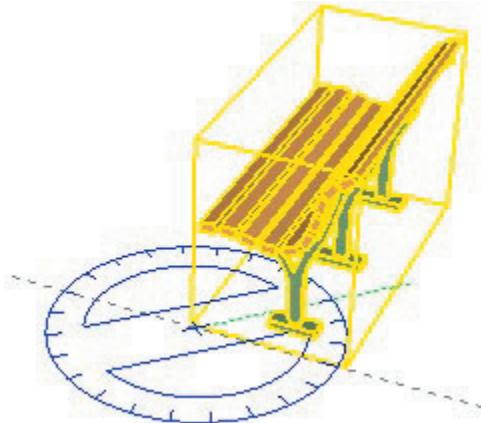


5. Click once to set the rotation point or the point upon which the geometry will rotate. Use the inference ToolTips help you find the center of the rotation.

6. Move the cursor from the starting point in the direction that represents the start of the rotation.



**Note** - Press the *ESC* key at any point during the operation to start over.



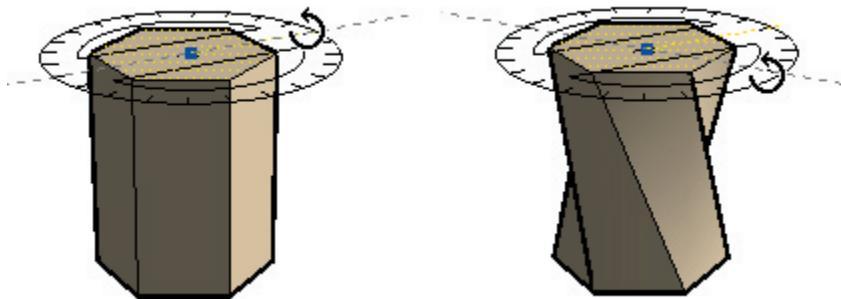
7. Click the mouse again to set the starting point of the rotation.
8. Move the mouse to rotate. If angle snaps are active under preferences, you'll notice that as you move the mouse, movements close to the protractor will result in angle snaps, while those further away from the protractor will allow free rotation.
9. Click a third time at the ending point of the rotation (to complete the rotation).



**Tip** - You can activate the Rotate Tool while nothing is selected by clicking on the Rotate Tool (the cursor turns to a Select Tool cursor), selecting an item with the temporary Select Tool (Rotate Tool remains active), and pressing the *ESC* key to reactive the Rotate Tool.

## Rotational Stretching with Auto-Fold

The Rotate Tool can also be used to stretch geometry by selecting and rotating a portion of the geometry. Any rotational movement that would cause a face to twist in on itself or otherwise become non-planar will activate SketchUp's Auto-Fold feature.



## Copying Objects Using Rotate

Press and hold the Ctrl key before initiating a rotation operation to make SketchUp create and rotate a copy of your selection instead of the original.

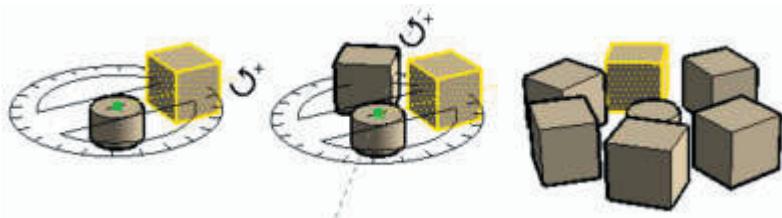
## Creating Multiple Copies (Radial Arrays)

The Rotate Tool can also be used to create radial arrays, or a series of copies around a rotate point. To create a radial array.

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the entities to be copied.
3. Select the Rotate Tool.
4. Press and hold the Ctrl key on your keyboard. This action informs SketchUp that you want to duplicate the selected entities.
5. Move the cursor from the starting point in the direction that represents the start of the rotation.
6. Click the mouse again to set the starting point of the rotation.
7. Rotate to copy the entities. A copy of the selected entities will follow as you move your mouse.
8. Click at the destination point to finish your copy operation. The copied entities are now selected and the original entities are deselected.

9. Type a multiplier value to create additional multiple copies. For example, typing in 2x (or \*2) will create one additional copy (or 2 copies total, the one you manually copied plus one you automatically copied using this step) instead of just one.

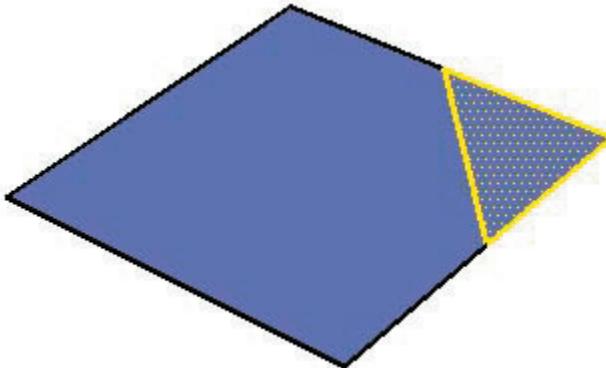
10. See the Multi-Copy topic for additional information on this feature



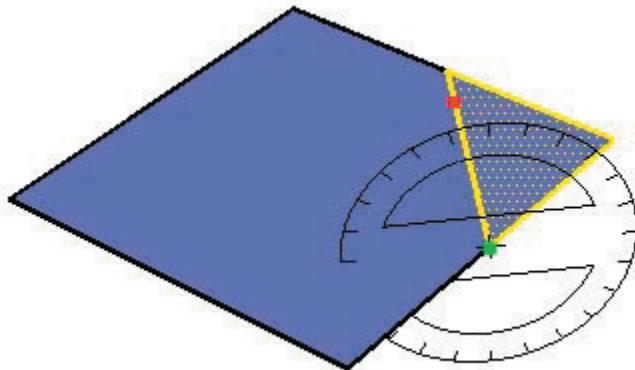
## Folding Along an Axis of Rotation

You can fold geometry by setting the protractor along an edge that will act like a fold line and then folding geometry at that line. To fold geometry along an axis of rotation:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the geometry to rotate. The bottom of the triangle will act as a fold line.

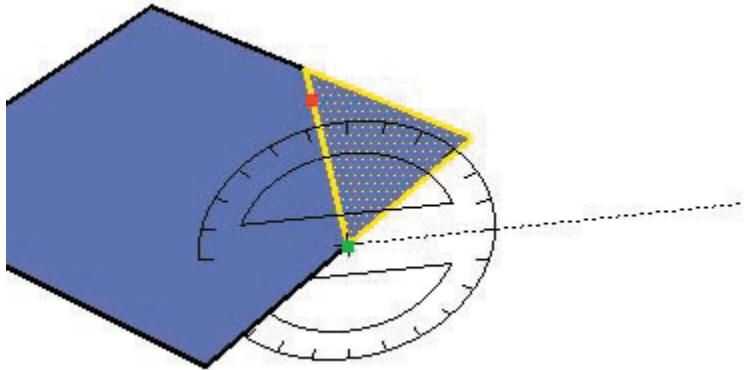


3. Select the Rotate Tool. The cursor will change to a protractor with a circular arrow.
4. Click and hold on one end of the fold line or edge where the fold will appear in your geometry.
5. Drag the cursor along the fold line to align the protractor to the fold line (the bottom of the triangle).

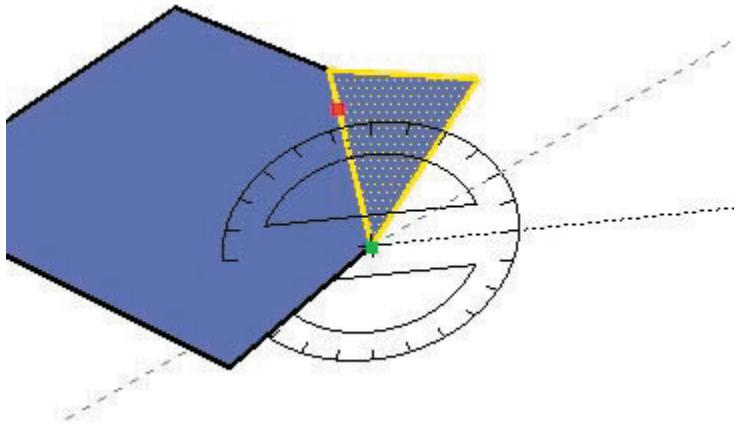


6. Release the mouse button to set the rotation point or the point upon which the geometry will rotate.

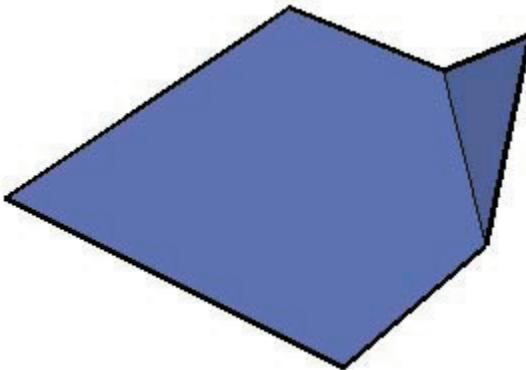
7. Click the mouse again to set the starting point of the rotation.



8. Move the mouse to rotate. If angle snaps are active under preferences, you'll notice that as you move the mouse, movements close to the protractor will result in angle snaps, while those further away from the protractor will allow free rotation.



9. Click a third time at the ending point of the rotation (to complete the rotation).



### Creating Copies at an Equal Distance Apart

You can divide the distance between the copy and the original by typing in a divisor value. For example, typing 5/ (or /5) will create five copies evenly distributed between the original and the first copy. You can keep typing in distances and multipliers until you perform another operation.

### Entering Exact Angle Values

The degree of rotation you have indicated appears in angular degrees in the VCB while rotating. You can also manually enter in angular rotation or slope values directly into the VCB while rotating geometry.

### Entering an Angular Rotation Value

To specify an exact angle in degrees, type a decimal value into the VCB while rotating the cursor around the protractor. For example, typing in **34.1** will give you an exact 34.1 degree angle. Negative values will move angle in a counter-clockwise direction. You can specify an exact angular value either during or immediately after your rotation operation.

### Entering a Slope Value

To specify a new angle as a slope, type in the two values separated by a colon in the VCB, such as **8:12**. Negative values will move angle in a counter-clockwise direction. You can specify an exact angular value either during or immediately after your rotation operation.

# Scale Tool

The Scale Tool is used to resize and stretch portions of geometry relative to other elements in your SketchUp model. Activate the Scale Tool from the Modification toolbar or the Tools menu.

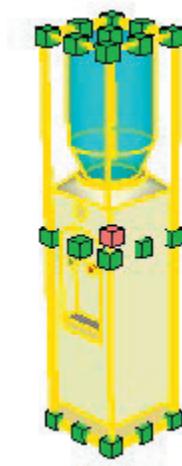


**Note - A Global Scale is an operation whereby the entire model is scaled simultaneously by applying a desired dimension to the distance between two points. The Scale Tool is only intended to perform scaling operations on portions of your model (not the entire model). Use the Tape Measure Tool's Global Re-Scale functionality to perform global scaling operations.**

## Scaling Geometry

To scale geometry:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the geometry to scale.
3. Select the Scale Tool. The cursor will change to a box within another box. Scaling grips will appear around the selected geometry.



4. Click on a scaling grip to select the grip. The selected grip and the opposite scaling grip will highlight in red. Each scaling grip provides a different scaling operation. See Scaling Options section for further information.

5. Move the mouse to scale the geometry. The Value Control Box (VCB) displays relative size of the item as you scale the item. You can enter the desired scale dimensions after the scale operation is complete.



**Note - Press the ESC key at any point during the operation to start over.**

6. Click again when you are done scaling the geometry.

### Scaling Auto-Folding Geometry

SketchUp's Auto-fold feature works automatically with all Scale operations. SketchUp will create folding lines as necessary to maintain planar faces.

## Scaling a 2D Surface or Image Object

Two-dimensional surfaces and Image entities can be scaled just as easily as three-dimensional geometry. The Scale Tool's bounding box contains nine scaling grips when scaling a 2D face. These operate in a similar manner to the grips in a 3D bounding box, and also work with the Ctrl and Shift modifiers.

The bounding box is a 2D rectangle when scaling a single 2D surface that lies in the red-green plane. The bounding box will be a 3D volume if the surface to be scaled is out of plane with the current red-green plane. You can ensure a 2D scale by aligning the Drawing Axes to a surface prior to scaling.

## Scaling Components

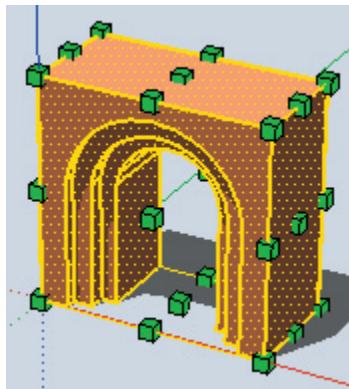
Scaling a Component entity scales the individual instance. All other instances of the component will retain their individual scales. This feature allows you to have many differently scaled versions of the same component in your model.

Scale operations within a component's context (such as scaling a Line entity within a component) affects the component definition and, therefore, all instances of the component will be scaled to match (all instances of the same Line entity in all component instances).

## Scaling Options

Upon activation, the Scale Tool displays all the grips you may use. Any grips hidden behind geometry will become visible whenever touched by the mouse cursor, and remain fully operable. Turn on X-ray Transparency mode to reveal any hidden grips.

The Scale Tool allows you to perform both uniform scaling and nonuniform scaling (stretching operations). The scaling grip that is used dictates the type of scaling you perform.



### Corner Grips

Corner grips scale the selected geometry from the opposite corner. The default behavior is a uniform scale such that the proportions remain intact and a single scale factor or dimension is displayed in the VCB.

### Edge Grips

Edge grips scale the selected geometry from the opposite edge by two dimensions simultaneously. The default behavior is a nonuniform scale, meaning that the proportions of the object will change. The VCB displays two values separated by a comma.

### Face Grips

Face grips scale the selected geometry from the opposite face in only one dimension. The default behavior is a nonuniform scale, meaning that the proportions of the object will change. The VCB displays and accepts a single value.

## Scaling About the Geometry Center

The Scale Tool allows you to scale outward from geometry's center point. Hold down the Ctrl key at any time during a scale operation to display the geometry's center point, click on any of the other scaling grips, and drag outward or inward to scale accordingly.

## Toggling Between Uniform and Non-uniform Scaling

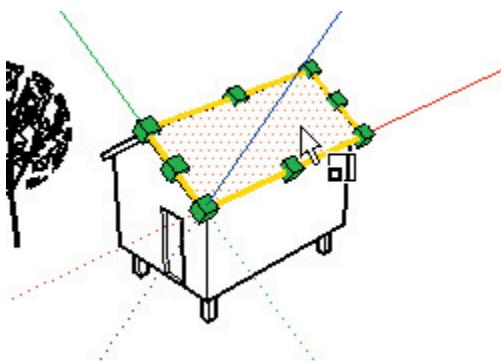
You might need to maintain the uniformity of geometry as it is being scaled, despite performing nonuniform scaling. The Shift key toggles to uniform scaling operation (from a nonuniform scaling operation) and to nonuniform scaling operation (from a uniform scaling operation).



**Note** - The *Ctrl* and *Shift* keys can be used in conjunction to allow uniform and non-uniform scaling from the center of the selected geometry.

## Controlling Scaling Direction With The Axis Tool

You can precisely control the direction of scaling by first repositioning the drawing axes with the Axes Tool. The Scale Tool will use the new red, green, and blue directions to orient itself, and control grip direction, after the axes are repositioned.



## Entering Exact Scale Values

The VCB at the bottom right corner of the SketchUp window displays the axis dimensions that are being scaled, and the value of the scale itself, in the default units (as specified under the Units panel of the Model Info dialog box) during a scaling operation. Type a scale value into the VCB to directly scale geometry during or immediately after a scaling operation.

### Entering a Scale Multiplier Value

You can specify a new dimensional length value during or directly following an scaling operation. To enter a dimensional length value during a scaling operation:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the geometry to scale.
3. Select the Scale Tool. The cursor will change to a box within another box. Scaling grips will appear around the selected geometry.

4. Click on a scaling grip to select the grip. The selected grip and the opposite scaling grip will highlight in red. Each scaling grip provides a different scaling operation. See Scaling Options section for further information.
5. Move the mouse to scale the geometry. The Value Control Box (VCB) displays relative size of the item as you scale the item. You can enter the desired scale dimensions after the scale operation is complete.
6. Type the dimensional length value (such as 2' 6" for two feet and six inches or 2m for two meters) in the VCB and press Enter or Return.

### Mirroring Geometry using the Scale Tool

The Scale Tool can also be used to mirror geometry by pulling a grip towards and then beyond the point about which you are scaling. This operation allows you to pull geometry inside out. Note that the grips snap to certain negative values (such as -1, -1.5, and -2) just as they do in the positive direction. You can force a mirror by typing in a negative value or dimension.

### Entering Multiple Scale Values

The VCB always indicates the scaling factors associated with a particular operation. A 1D scaling operation requires one value. A 2D scaling operation requires two values, separated by a comma. A Uniform 3D scaling operation requires only one value whereas a Non Uniform 3D scaling operation requires three values, each separated by a comma.

You'll notice that during the scale operation, a dashed line appears between the scaling point and the grip you've selected. Entering a single value or distance in the VCB tells SketchUp adjust the anchor to grip distance to be that scale value or distance, regardless of which mode (1D, 2D, 3D) is active.

When scaling in multiple directions, typing in multiple values separated by commas will resize the object(s) based on the entire bounding box dimension(s), not the objects individually. (To scale objects based on a particular edge or known distance, you can use the Tape Measure Tool.)

# Push/Pull Tool

The Push/Pull Tool is used to push and pull Face entities to add volume to or subtract volume from your 3D models. You can use push/pull to create volume out of any face type, including circular, rectangular, and abstract faces. Activate the Push/Pull Tool from either the Modification toolbar or the Tools menu.

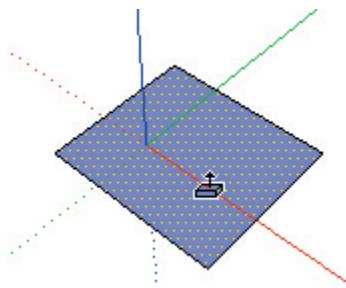


**Note -** Push/Pull works only on faces, and therefore does not work when SketchUp is set to a Wireframe rendering style.

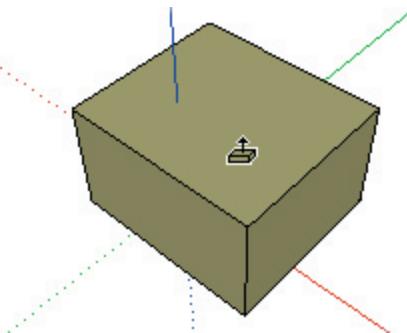
## Creating a Volume

Push/Pull Tool is used to expand or decrease the volume of geometry in your models. To push or pull faces:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the face to push or pull.
3. Select the Push/Pull Tool. The cursor will change to a 3D rectangle with an up arrow.
4. Click on the face



5. Move the cursor up to create volume in the positive direction or move the cursor down to expand the volume in the negative direction.



**Note** - Press the *ESC* key at any point during the operation to start over.

6. Click again when the volume has reached the desired size.



**Note** - You can also press and hold the mouse button, drag the mouse, and release the mouse button to create a volume.



**Note** - When you create a single face, as shown in the previous screenshot, on the ground plane (the red/green plane), SketchUp assumes you are going to be using that face as the floor of a building. The front of the face (green) points down and the back of the face (purple) points up. Therefore, when you pull a single face up (in the blue direction), you are really pulling up from the back of the face and the blue plane temporarily acts as the "below ground" direction. Therefore a double-click, after performing this action, applies the same positive amount or returns the entity to the single face you started with.

## Entering Precise Push/Pull Values

The displacement of a push/pull operation is displayed in the Value Control Box. You can specify an exact push/pull value either during or immediately after your push/pull operation. Negative values will perform the push/pull in the opposite direction.

## Repeating a Push/Pull Operation

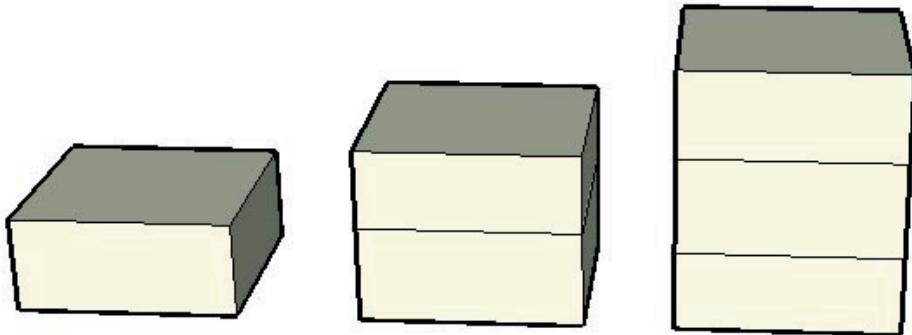
Double-clicking on another face immediately after a push/pull operation will automatically apply another push/pull operation, of the same amount, to the other face.



**Note** - The side of the face that you double-click on affects the direction of the repeated push/pull operation. If your last push/pull was on a front face, and you double-click on the back side face, the push/pull will occur in the opposite direction.

## Create New Push/Pull Starting Face

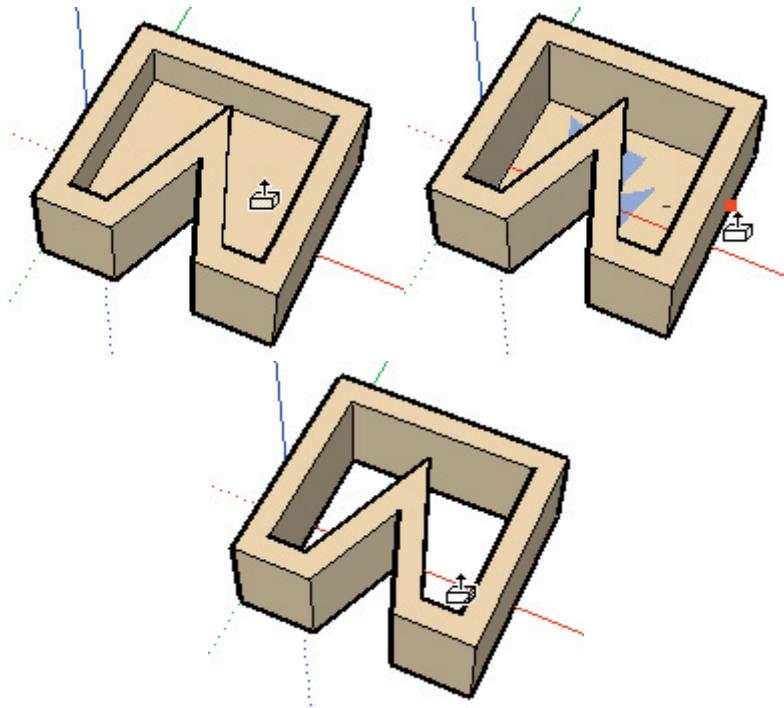
Push/Pull a face (click on the face, move, and then click again) and then hold Ctrl and push/pull again. The lines that represent the edges of the top-most face will remain as the starting point for a new push/pull operation. This mechanism is useful for creating quick multilevel buildings. The following image shows a face that was pulled up (left), then the user pressed Ctrl and pulled again (middle) and then the user pressed Ctrl and pulled again (right).



This operation is particularly useful for creating quick space planning diagrams (such as for an office building). Simply use a combination of push/pull and push/pull with Ctrl to create offices, halls, break rooms, meeting rooms and so on (with walls created when Ctrl is pressed).

## Creating Voids

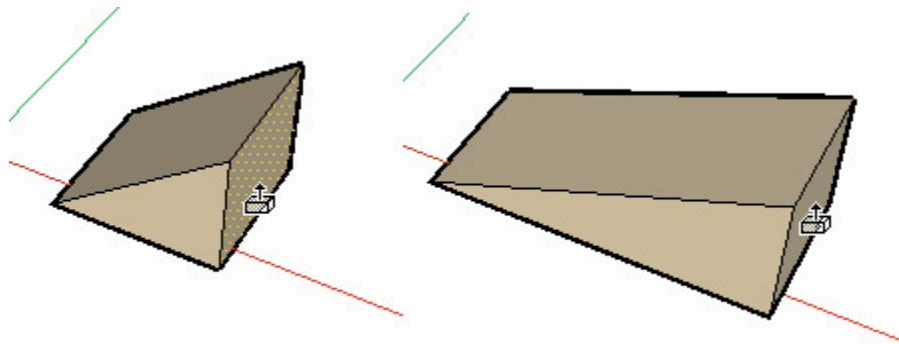
Push/pull will implode the shape into the volume and toward the back face of the volume when you use push/pull on a shape that is part of another volume. SketchUp will subtract the shape and create a 3D void if the shape is pushed completely out of the back of the volume as in the following example.



**Note** - This operation only works when the front and back faces are parallel.

## Moving Faces Perpendicular

You can force push/pull to move faces in a perpendicular direction by holding down the Alt key while using the Push/Pull Tool. This operation is useful for deforming objects. Auto-fold is not available during this operation.



# Follow Me Tool

The Follow Me Tool is used to extrude faces along a path such as an edge or line drawn with the freehand pencil. This tool is especially useful when trying to add details to a model by allowing you to draw the detail at one end of a path on the model and using the Follow Me Tool continue that detail along the path. You can manually and automatically extrude a face along a path using the Follow Me Tool. Activate the Follow Me tool from the Tools menu or the Modification toolbar.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

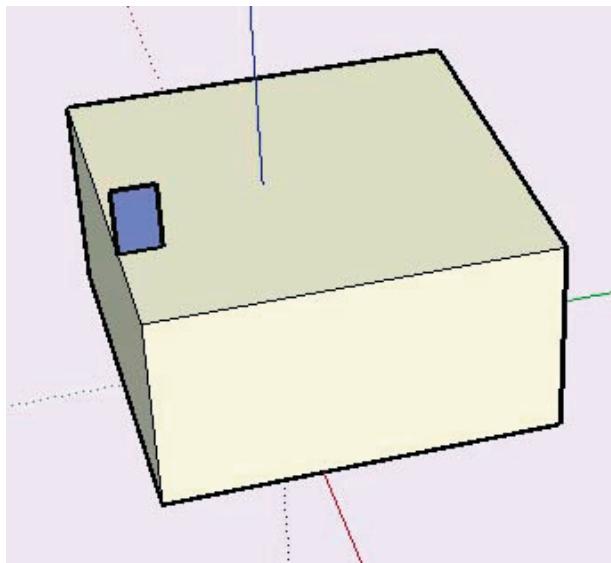


**Note** - The path and the face must be in the same context.

## Manually Extruding a Face Along a Path

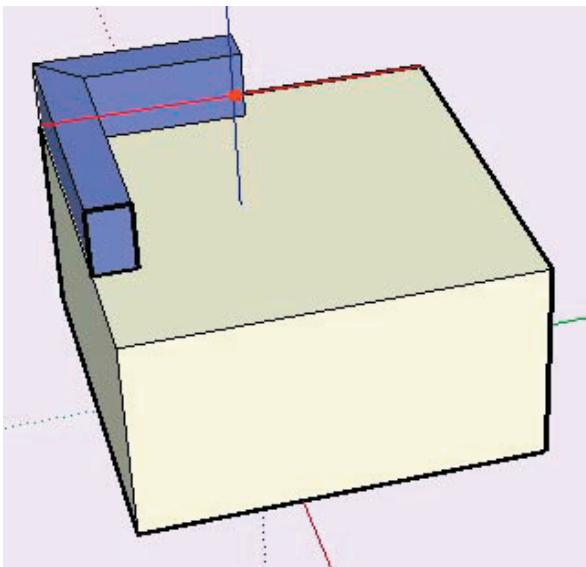
The manual method for extruding a face along a path allows you to control the direction the face will travel while performing the extrude. To manually extrude a face along a path using the Follow Me tool:

1. Identify the edge of the geometry you want to modify. This edge will be your path.
2. Draw a profile of the face that you want to follow the path. Make sure that this profile is approximately perpendicular to the path.



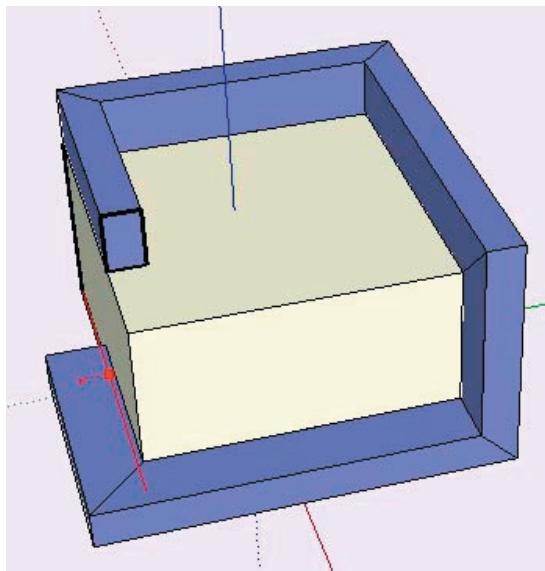
3. Select the Follow Me tool from the Tools menu. The cursor will change to a slanted box with an arrow.
4. Click on the profile that you created.

5. Drag the mouse along the path. SketchUp will highlight the path, in red, you are following as you drag your cursor around the model. You must touch the segment of the path immediately adjacent to the profile for the Follow Me to start in the correct location. If you select an edge, as your starting edge, that is not touching the profile, Follow Me will start extruding at that edge, not from the profile to that edge.



**Note** - Press the *ESC* key at any point during the operation to start over.

6. Click again to execute the Follow Me command when you reach the end of the path.



### Preselecting the Path

You can preselect the path using the select tool to help the Follow Me tool follow the correct path. To extrude a face along a pre-selected path:

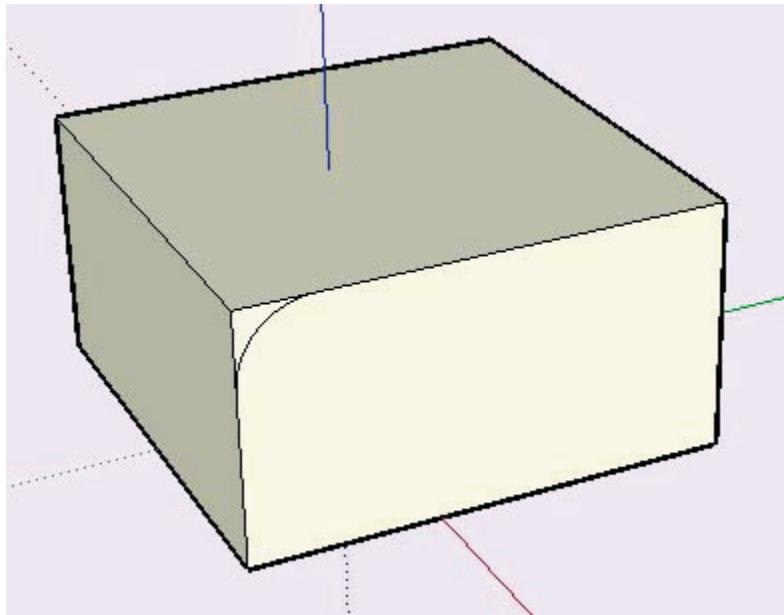
1. Draw a profile of the face that you want to follow the path. Make sure that this profile is approximately perpendicular to the path.
2. Select the continuous set of edges that represent the path.
3. Select the Follow Me tool from the Tools menu (the edges should still be selected). The cursor will change to a slanted box with an arrow.
4. Click on the profile that you created. The surface will be extruded continuously along your pre-selected path.

### Automatically Extruding a Face Along a Single Surface Path

The simplest and most accurate way to extrude a face along a path is to have the Follow Me Tool automatically select and follow a path on a single coplanar surface. To automatically extrude a face along a path on a single surface using the Follow Me tool:

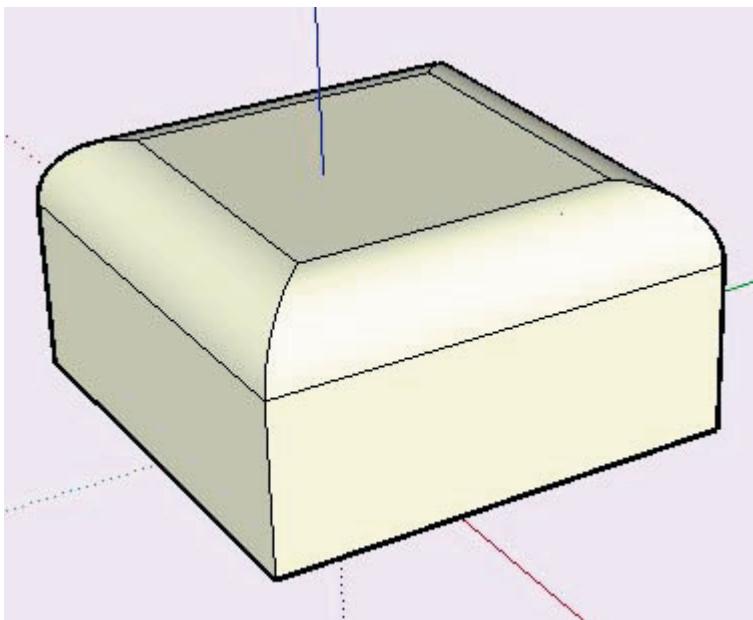
1. Identify the edge of the geometry you want to modify. This edge will be your path.

2. Draw a profile of the face that you want to follow the path. Make sure that this profile is approximately perpendicular to the path.



3. Select the Follow Me tool from the Tools menu.  
4. Press and hold the Alt Key while clicking on the profile that you created.

5. Move the cursor off the profile surface onto the surface around which you wish to sweep. The path will automatically close.

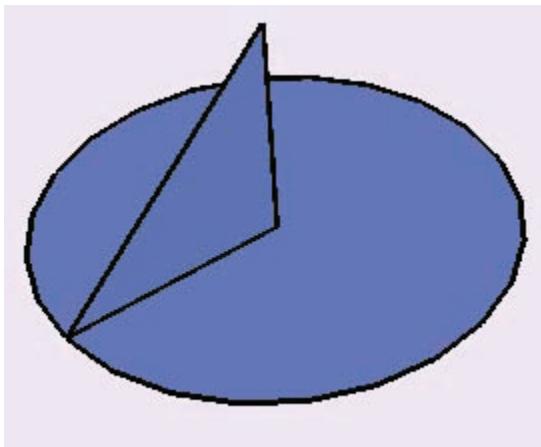


**Note** - If your path consists of the edges around a single surface, you can select the surface and then the Follow Me tool to automatically follow the edges around the surface.

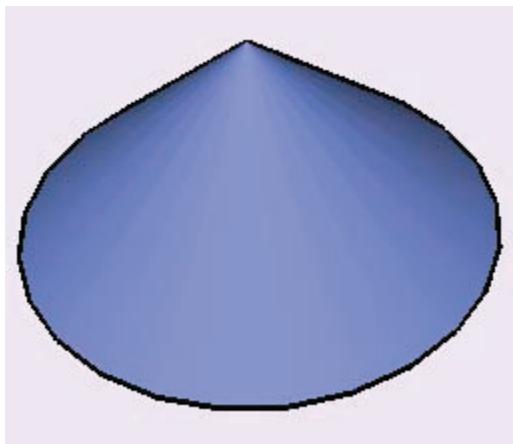
## Creating a Lathed Shape

You can use the Follow Me Tool to create full lathed shapes using circular paths. To create a lathed shape:

1. Draw a circle whose edge will represent the path.
2. Draw a face perpendicular to the circle. The face does not have to be on or even touch the circle's path.



3. Select the Follow Me Tool from the Tools menu. The cursor will change to a slanted box with an arrow.
4. Follow the edge of the circle with the face using one of the methods above.



# Offset Tool

The Offset Tool creates copies of coplanar lines and faces that are a uniform distance from the originals. You can offset edges of faces either inside or outside of the original face. Offsetting a face will always create a new face. Activate the Offset Tool from the Modification toolbar or from the Tools menu.

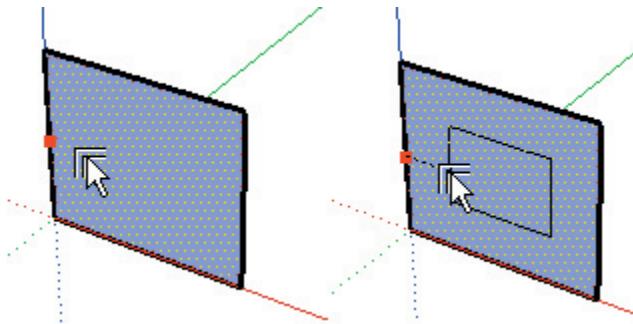


**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Offsetting a Face

The most common use of the Offset Tool is to offset the edges that bound a face. To perform a face offset:

1. Select the Offset Tool. The cursor will change to two offset corners.
2. Click on the face to be offset.
3. Move the mouse cursor and to define the offset dimension. The offset distance is displayed in the VCB. You can offset either within or outside of the face; on rectangular face or circular face.



**Note** - Press the **ESC** key at any point during the operation to start over.

4. Click to accept the offset and create the offset polygon.

## Offsetting Lines

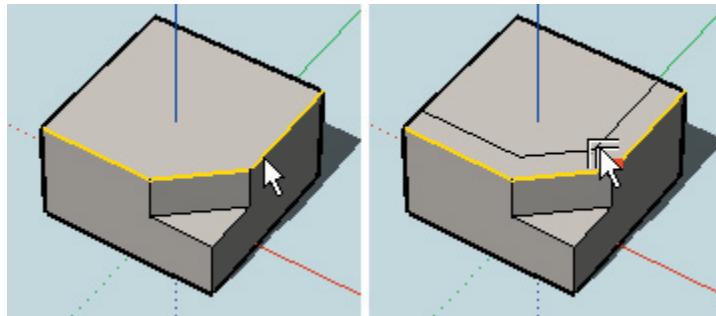
You can also select and offset connected, co-planar, lines (and arcs) for an offset. To offset lines:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the lines to be offset. You must select two or more connected lines, and all your lines must be coplanar. You can use the CTRL and/or Shift keys to change your selection.
3. Select the Offset Tool.

4. Click on one of your selected line segments. The cursor will automatically snap to the nearest line segment.

5. Move the mouse to define the offset dimension.

6. Click your mouse to accept the offset lines.



**Tip** - You can click once on the selected line segments, drag to set the offset while holding down the mouse button, and release the button to accept.



**Note** - Offsetting an Arc entity will create a Curve entity that cannot be edited. The original Arc, however, can still be edited after this operation.

## Repeating an Offset Operation

Double-clicking on another face immediately after a offset operation will automatically apply another offset operation, of the same amount, to the face.

### Entering Precise Offset Values

The VCB at the bottom right corner of the SketchUp window displays the length of the offset in the units as specified under the Units panel of the Model Info dialog box, while offsetting entities. You can also specify an exact offset during, or immediately after, an offset operation.



**Note** - You can type values in the VCB using an alternate measuring system than the default system. SketchUp will convert the value to the default system. For example, you can type in 3' 6" even if you are using metric system as your default.

## Entering an Offset Value

You can specify a new offset length during or directly following an offset operation. To enter a offset value during a offset operation:

1. Select the Select Tool. The cursor will change to an arrow.
2. Select the lines to be offset. You must select two or more connected lines, and all your lines must be coplanar. You can use the CTRL and/or Shift keys to change your selection.
3. Select the Offset Tool.
4. Click on one of your selected line segments. The cursor will automatically snap to the nearest line segment.
5. Move the mouse to define the offset dimension.
6. Click your mouse to accept the offset lines.
7. Type the positive or negative offset value (such as 20' or -35mm) in the VCB and press Enter or Return.

# Intersect With Model Tool

The Intersect With Model Tool is used to create complex geometry. This tool allows you to intersect two elements, such as a box and a tube, to automatically create new edges and faces where the elements intersect. These faces can then be pushed, pulled or deleted to create new geometry. Activate the Intersect With Model Tool from either context menu or the Edit menu.

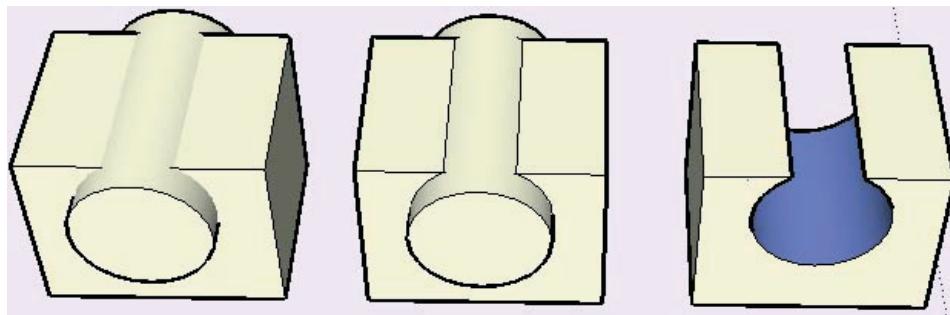


**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Creating Complex Geometry

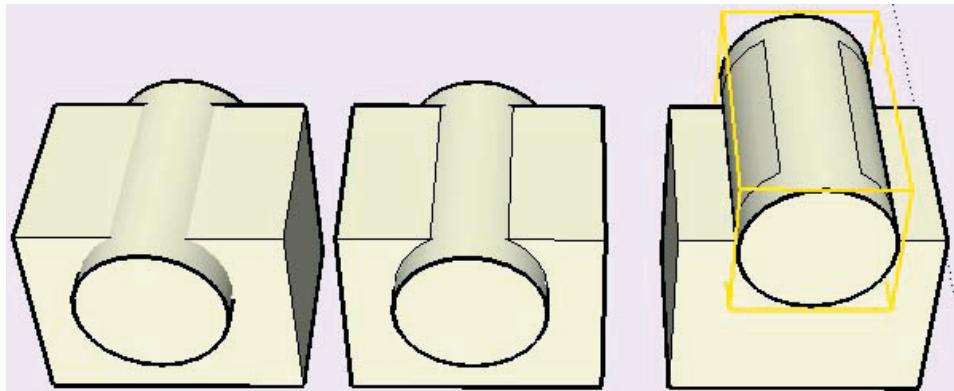
To create complex geometry using the Intersect With Model Tool:

1. Create two distinct geometries, such as a box and a tube.
2. Select the Select Tool. The changes to an arrow.
3. Triple-click on the tube to select all of the entities of the tube
4. Select the Move Tool. The cursor will change to a four-way arrow.
5. Move the tube such that it intersects the box completely in any way you like (left-most image below). Notice that no edges exist where the tube meets the faces of the box. The tube should remain selected although it is within the box.
6. Context-click on the selected tube.
7. Select Intersect With Model from the context commands menu. The tool creates edges where the tube intersects the box (middle image below).
8. Delete or move the portions of the tube that you do not want to keep (right-most image below). Notice that SketchUp will have created new subdivided faces where the tube intersected the box.



## Intersect With Model, Groups, and Components

Edges created by the Intersect With Model Tool are drawn in the current context. For example, if one of your intersecting entities is a group, and you perform the Intersect with Model while editing that group, the intersection lines will be applied within the group (right-most image below).



# Position Texture Tool

Materials within SketchUp are applied as tiled images meaning that the pattern or image will repeat both vertically and horizontally across any entities you paint. SketchUp's Position Texture Tool is used to adjust a material on a surface in a number of ways, including repositioning, resizing, and distorting. Additionally, this tool allows you to perform unique actions on images such as painting a picture around a corner or projecting it on a model. Activate the Position Texture Tool from the context menu for a Face entity.



**Note** - The Position Texture Tool can only be used to modify textures applied to flat surfaces. You cannot edit a texture applied to a curved surface as a whole though you can use the View > Hidden Geometry menu item view and edit the texture on the individual faces that make up the curve's face set.



**Note** - Normally a texture is a subset of a material. However, the terms material and texture are used interchangeably in this topic.



**Video** - A video tutorial on this topic is available at <http://www.sketchup.com/training/tutorials.php>

## Repositioning a Material

Repositioning the material is the easiest of position texture operations. To reposition a material:

1. Context-click on the material to display its context menu.
2. Select the Position option from the Texture sub-menu. A matrix of dotted lines are displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Drag the mouse cursor on the surface to reposition the texture on that surface. If you want to rotate the tiled image, context click on the surface again and select Rotate or Flip.
4. When you are finished modifying the texture, right click and select Done, or just click outside the texture to exit the Position Texture Tool.



**Tip** - Press the Esc key anytime during editing to reset the material's position its previous position. Press Esc twice to cancel the entire texture positioning operation. While positioning a texture, you can back up a step at anytime by right-clicking and selecting Undo from the context menu.

## Material Positioning Pins

The Position Texture Tool uses pins to manipulate a material. Pins can be moved or dragged. A move operation simply moves the pin to another location on the material. A drag operation performs some manipulation of the material, such as a resize or skew.

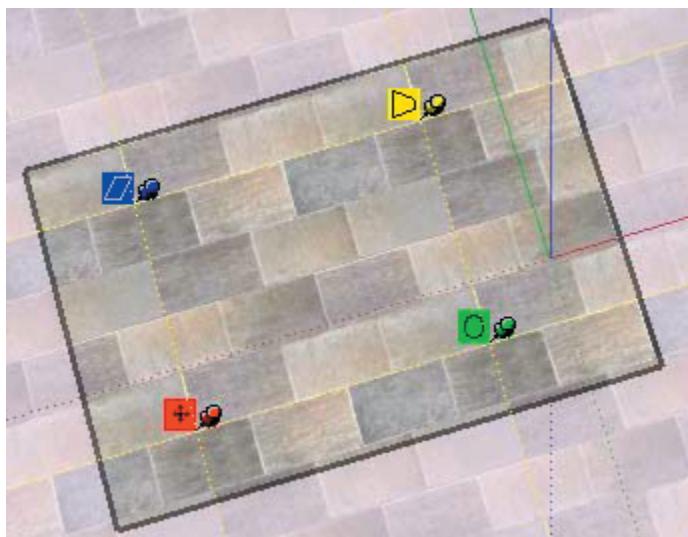
The Position Texture Tool has two modes: Fixed Pin Mode and Free Pin Mode.

SketchUp

## Manipulating a Material Using Fixed Pin Mode

Fixed pin mode allows you to scale, skew, shear and distort a texture, while constraining or "fixing" one or more pins. Fixed pin mode is best for materials that tile like brick or roofing textures. To manipulate a material using Fixed Pin Mode:

1. Context-click on the material to display its context menu.
2. Select the Position option from the Texture submenu. A matrix of dotted lines are displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Right click on the material
4. Select the Fixed Pins menu item if there is not a check mark next to the item. Colored icons appear next to each pin, each icon representing a specific position texture operation.



5. Manipulate the material by clicking, holding, and dragging on one of the pins. See Fixed Pin Mode Options in this topic for additional information.



**Note** - Single-clicking on a pin grabs the pin allowing you to move the pin to a different location on the texture. This new position will be the starting point for any of the fixed pin mode operations. This action works in both Fixed Pin and Free Pin modes.

6. When you are finished modifying the texture, right click and select Done, or just click outside the texture to exit the Position Texture Tool.

## Fixed Pin Mode Options

 **Move Icon and Pin:** Drag (click and hold) the Move icon or pin to reposition the texture. When you are finished modifying the texture, right click and select Done, or just click outside the texture to close it. Or you can simply press Return (Enter) when finished.

 **Scale / Rotate Icon and Pin:** The Scale/ Rotate icon or pin is used to scale and rotate the material to any angle, based on the fixed pin location of the move pin. Dragging the cursor closer to, or farther from, the base pin will scale the material. Dragging the pin icon around the base pin causes SketchUp to rotate the material. A dashed arc is created in the direction that you rotate the material. If you hold the cursor over the dashed arc, the material will rotate, but not scale.

Notice that the dots along the dashed lines and arcs show you the current size and the original size of the texture for reference. You can change back to the original size by moving the cursor to the original arc and line. Or you can select Reset from the Context menu. Be forewarned that selecting Reset also resets the rotation as well as the scale.

 **Scale / Shear Icon and Pin:** The Scale/Shear icon or pin is used to simultaneously slant or shear and resize the material. Notice that the two bottom pins are fixed during this operation.

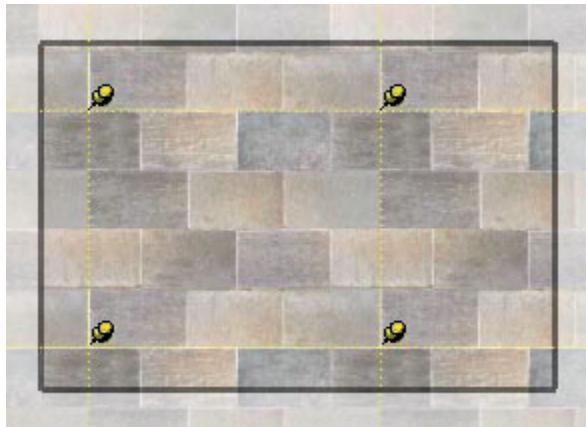
 **Distort Icon and Pin:** The Distort icon or pin is used to perform a perspective correction on the material. This feature is useful for applying image photos to geometry.

## Manipulating a Material Using Free Pin Mode

Pins are not constrained to other pins in Free Pin Mode allowing you to drag pins anywhere to distort the material just as you might distort a material as you stretch it over a drum. Free pin mode is best for positioning and removing the distortion from photographs. To manipulate a material using Free Pin Mode:

1. Context-click on the material to display its context menu.
2. Select the Position option from the Texture submenu. A matrix of dotted lines are displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Right click on the material

4. Select the Fixed Pins menu item if there is a check mark next to the item. Four pins appear.



5. Manipulate the material by clicking, holding, and dragging on one of the pins.



**Note** - Single-clicking on a pin grabs the pin allowing you to move the pin to a different location on the texture. This new position will be the starting point for any of the fixed pin mode operations. This action works in both Fixed Pin and Free Pin modes.

6. When you are finished modifying the texture, right click and select Done, or just click outside the texture to exit the Position Texture Tool.

## Manipulating a Image as Material Using Free Pin Mode

Free pin mode is especially useful when using an image as the foundation for geometry. For example, you can use a picture containing a door as the foundation for a photo-realistic door in a SketchUp model. To manipulate an image using Free Pin Mode:

1. Create a rectangular face.
2. Insert an image, such as the image of a real door, using File > Insert > Image as Texture. The cursor changes to the Paint Bucket Tool with the image.
3. Click on one corner of the face to place the starting point of the texture.
4. Drag the cursor away from the starting point so that the texture is resized over the rectangular face.
5. Click again to place the texture on the rectangular face.
6. Context-click on the material to display its context menu.
7. Select the Position option from the Texture submenu. A matrix of dotted lines are displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
8. Right click on the texture

9. Select the Fixed Pins menu item if there is a check mark next to the item.
10. Manipulate the material by clicking, holding, and dragging on one of the pins.



**Note - Single-clicking on a pin grabs the pin allowing you to move the pin to a different location on the texture. This new position will be the starting point for any of the fixed pin mode operations. This action works in both Fixed Pin and Free Pin modes.**

11. When you are finished modifying the texture, right click and select Done, or just click outside the texture to exit the Position Texture Tool.

## Fixed Pin and Free Pin Mode Context Commands

Context click while using the Position Texture tool to display the position texture context menu.

**Done:** The Done menu item is used to exit the Position Texture Tool and save the current texture position.

**Reset:** The Reset menu item is used to reset the position of the texture to the position prior to using the Position Texture Tool.

**Flip:** The Flip menu item is used to flip the texture horizontally (Left/Right) or vertically (Up/Down).

**Rotate:** The Rotate menu item is used to rotate the texture one of three predefined increments: 90, 180, and 270 degrees.

**Fixed Pins:** The Fixed Pins menu item is used to toggle between Fixed Pin and Free Pin modes.

**Undo:** The Undo menu item will undo the last position texture command. Unlike the Undo command in the Edit menu, this undo command will only keep track of a single operation at a time.

**Redo:** The Redo menu item cancels Undo operations, returning you to the texture positioning state previous to using the Undo command.



**Caution - The Edit > Undo command and Undo button will Undo everything you did during your texture positioning session. The Edit > Redo operation cancels the Undo Edit > Undo operation, returning you to the last texture positioning command that you performed.**

## Wrapping Material Around Corners

The texture can be wrapped around a corner, just as you might wrap a package with wrapping paper. To wrap textures around corners:

1. Create a three-dimensional cube.
2. Insert an image using File > Insert > Image. The cursor changes to the Select Tool with the image.
3. Click in the Drawing Area to place the starting point of the image.

4. Drag the cursor away from the starting point so that the image is resized.
5. Click again to place the image.
6. Context click on the image. The Image entity's context menu appears.
7. Select Use as Material. The image appears in the In Model material library within the Materials Browser.
8. Select the Paint Bucket Tool. The cursor will change to a paint bucket and the Materials Browser is activated.
9. Click and hold the Alt key while using the Paint Bucket Tool to change to the eye dropper.
10. Click on the thumbnail of your image in the In Model material library within the Materials Browser.
11. Release the Alt key
12. Click on a face of your model and paint the material.
13. Context-click on the material to display its context menu.
14. Select Texture > Position. Don't position anything!
15. Context-click again
16. Select Done from the Position Texture Tool context menu.
17. Click and hold the Alt key while using the Paint Bucket Tool to change to the eye dropper.
18. Click on the painted material using the eye dropper to sample the material.
19. Release the Alt key
20. Paint the sampled texture on the remainder of the model. The texture is wrapped around corners.



## Wrapping Material Around a Cylinder

A texture can also be wrapped around a cylinder. To wrap a texture, such as an image texture, around a cylinder:

1. Create a cylinder.
2. Insert an image using File > Insert > Image. The cursor changes to the Select Tool with the image.
3. Click in the Drawing Area to place the starting point of the image.
4. Drag the cursor away from the starting point so that the image is resized.
5. Click again to place the image.
6. Context click on the image. The Image entity's context menu appears.
7. Select Use as Material. The image appears in the In Model material library within the Materials Browser.
8. Click on the material in the Materials Browser. The cursor changes to the Paint Bucket Tool.
9. Paint the Material on the cylinder. The material will automatically wrap around the cylinder, repeating itself as necessary to wrap the entire model.

## Repositioning a Material on Hidden Geometry

You can adjust textures on a face, such as the faces of a cylinder, and then repaint the adjust texture across the entire curved surface of the cylinder. For example, to adjust a texture on a cylinder:

1. Create a cylinder.
2. Insert an image using File > Insert > Image. The cursor changes to the Select Tool with the image.
3. Click in the Drawing Area to place the starting point of the image.
4. Drag the cursor away from the starting point so that the image is resized.
5. Click again to place the image.
6. Context click on the image. The Image entity's context menu appears.
7. Select Use as Material. The image appears in the In Model material library within the Materials Browser.
8. Click on the material in the Materials Browser. The cursor changes to the Paint Bucket Tool.
9. Paint the Material on the cylinder. The material will automatically wrap around the cylinder, repeating itself as necessary to wrap the entire model.
10. Select the Display > Hidden Geometry menu item.
11. Select one of the faces of the cylinder
12. Context click on the selected face. The Face entity's context menu appears.
13. Select the Texture > Position menu item.

14. Reposition the texture on the face.
15. Sample the repositioned texture using the eyedropper button on the material browser, or using the Alt key with the Paint Bucket tool.
16. Click on Display > Hidden Geometry to turn off Hidden Geometry.
17. Paint the sampled, repositioned, texture on the remainder of the cylinder. Your texture now appears as though it has been repositioned on the entire cylinder.

## Projecting a Material

SketchUp's Texturing Positioning feature also lets you project materials or images onto faces as though projected using a slide projector. This feature is particularly useful if you wish to project a topographic image over a site model, or an image of a building onto a model representing the building. To project an image over a model:

1. Create a model, such as a cone, topography, or building front. This model will receive the projected image.
2. Insert an image using File > Insert > Image. The cursor changes to the Select Tool with the image.
3. Place the image in front of the model that will receive the projection.
4. Size the image so it is large enough to cover the entire model.
5. Right click on the image and select Explode to turn the image into a projected texture.



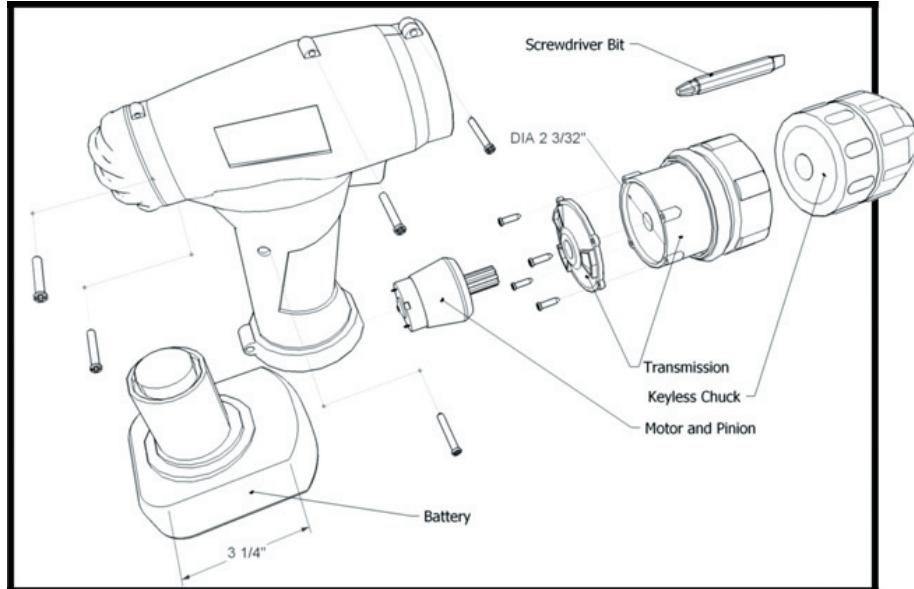
**Note - Turn on x-ray display mode for the image to ensure the image is positioned such that it will cover the entire model.**

6. Select the Sample Paint Tool (eyedropper) from the Materials Browser. Notice, when you drag the Sample Paint Tool over the image, a square appears on the tip of the tool. This square indicates that you are in projected texture mode.
7. Sample the projected texture with the Sample Paint Tool.
8. Paint the texture onto the faces of the model. The image will appear as though it were projected directly on the faces, adjusting to the contours of model.

# Construction Tools

This section of the user's guide covers the construction tools within SketchUp. Construction tools are those tools used to create construction geometry and prepare physical documentation. These tools include the Measure Tool, Text Tool, and Dimension Tool.

Tools in this section are covered in the order they appear within the Tools menu.



# Tape Measure Tool

The Tape Measure Tool is used to perform a number of dimension-related operations. These operations include measuring the distance between two points in 3D space, creating Construction Line entities, creating Construction Point entities, and scaling an entire model to an exact dimension. Activate the Tape Measure Tool from the Construction toolbar or the Tools menu.

## Measuring Distance

The Tape Measure Tool is primarily used to measure distances between two points. To measure a distance between two points:

1. Select the Tape Measure Tool. The cursor changes to a tape measure.
2. Click to set the starting point of your measurement. Use the inference ToolTip to make sure you click on the exact point.
3. Move the mouse in the direction you want to measure. A temporary measuring tape line, with arrows at each end, will stretch out from your starting point as you move the mouse. The Tape Measure Tool's measuring tape line functions like an inference line and will change color to match axes colors when it is parallel to any axes. The VCB dynamically displays the length of your measuring tape as you move the mouse around your model.



**Note** - Press the *ESC* key at any point during the operation to start over.

4. Click again at the endpoint of your measurement. The final distance is displayed in the VCB and, in some cases (such as when measuring from a midpoint of a line to another midpoint), a Construction Line entity is created.



**Tip** - You can also click and hold on the starting point of the distance you wish to measure, drag the mouse to the endpoint of the measurement, and release the mouse to obtain a measurement.

## Creating Construction Lines and Construction Points

Construction Line entities and Construction Point entities are useful for drawing precisely. To create an infinite parallel construction line using the Tape Measure Tool:

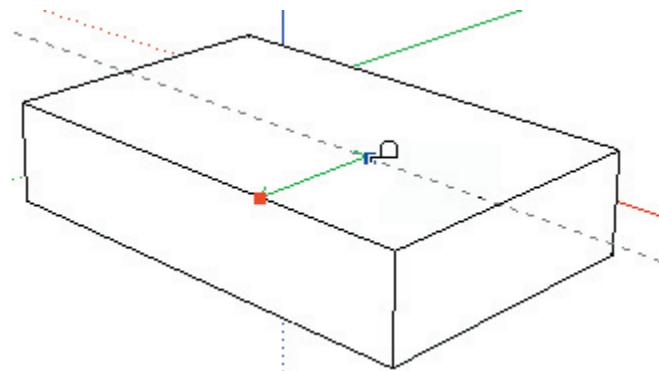
1. Select the Tape Measure Tool. The cursor changes to a tape measure.
2. Click on a line segment, that will be parallel to the construction line, to set the starting point of your measurement. You must click on an "On Edge" point between the start and end points in the line segment.



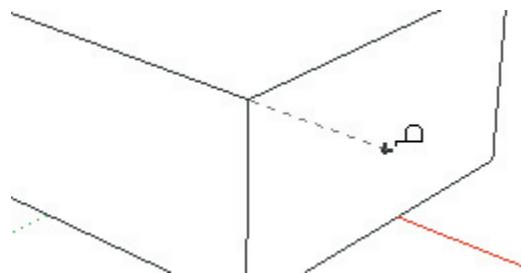
**Note -** Start from an "On Edge" point inference and move across a face to generate an infinite parallel construction line. Start from an "Endpoint" or "Midpoint" to create a finite construction line with a construction point at the end.

3. Move the mouse in the direction you want to measure. A temporary measuring tape line, with arrow heads at each end, will stretch out from your starting point, along with a construction line, as you move the mouse. The Tape Measure Tool's measuring tape line functions like an inference line and will change color to match axes colors when it is parallel to any axes. The Value Control Box dynamically displays the length of your measuring tape as you move the mouse around your model.

4. Click again at the point where you want to set your construction line. The final distance is displayed in the VCB.



**Tip** - Starting from an endpoint or midpoint results in a Construction Point.



### Entering Exact Values

While you are drawing construction lines, the VCB at the bottom right corner of the drawing area displays either the length of the line or its parallel offset in the units specified under preferences. You may specify different values simply by typing them in. You may specify a negative length which draws the line in the direction opposite the one indicated.

## Scaling an Entire Model

You can rescale your model, to a more-precise dimension, during the modeling process by specifying the desired dimension between two points using the Tape Measure Tool. This line is referred to as the *reference line*. To scale an entire model:

1. Select the Tape Measure Tool. The cursor changes to a tape measure.
2. Click one end of a line segment to set the starting point of a measurement. Use the inference ToolTip to make sure you click on the exact point.
3. Move the mouse to the end point of the same line segment. A temporary measuring tape line, with arrows at each end, will stretch out from your starting point as you move the mouse. The Tape Measure Tool's measuring tape line functions like an inference line and will change color to match axes colors when it is parallel to any axes. The VCB dynamically displays the length of your measuring tape as you move the mouse around your model.
4. Click again at the other end of the line segment. The final distance is displayed in the VCB.
5. Enter a new size for the line in the VCB. This size will be used as the basis for a proportional rescale of your model.

A dialog box appears asking if you are sure you want to resize (scale) your entire model. Click Yes to resize the model.



**Note -** Only components that are created within the current model (not dragged from the Component Browser and loaded from an external component file) can be resized.

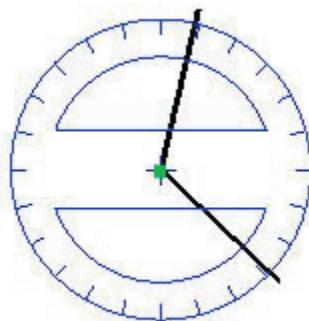
# Protractor Tool

The Protractor Tool is used to measure angles and create Construction Line entities. Activate the Protractor Tool from the Construction Toolbar or the Tools menu.

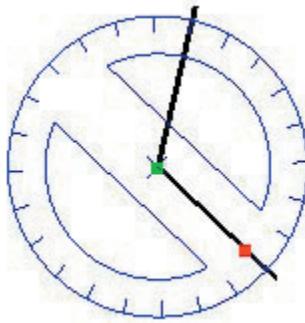
## Measuring an Angle and Placing Angled Construction Lines

It is often necessary to set Construction Line entities at an angle to aid in drawing angles. To measure and set a Construction Line entity at an angle:

1. Select the Protractor Tool. The cursor will change to a protractor, aligned to the red/green plane and with its center point fixed to the cursor.
2. Move the protractor around to align to the proper orientation in your model. The Protractor Tool will shift its orientation as it aligns with nearby axes and geometry. You can lock in a particular orientation by holding down the Shift key when the Protractor is in your desired orientation.
3. Click once to set the center point of the protractor on the vertex point of the angle to be measured. Monitor the inference ToolTips to be sure you have the Protractor exactly where you want it.



4. Align the base of the protractor with the first line in the angle to be measured by dragging away from the vertex along the line while rotating the protractor. Monitor the ToolTips to ensure that you are setting the line appropriately.



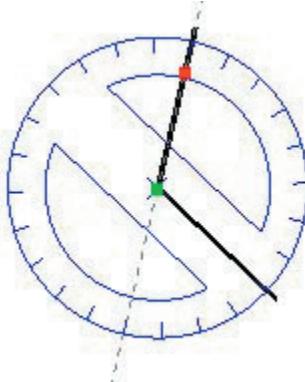
5. Click again to set the start of the angle.

6. Rotate the protractor again such that it is aligned with the second line segment in the angle you are measuring. A dotted construction line will follow your cursor around the Protractor. Notice that the protractor has marks, indicating 15 degree increments, on its edge. The angle will snap to these tick marks when the cursor is close to the protractor while moving around the protractor. Conversely, angle will move in more precise (smaller) increments when your cursor is farther from the center of the protractor while moving around the protractor.



**Note** - Press the *ESC* key at any point during the operation to start over.

7. Click a third time, on the second line in your angle, to display the angle in the VCB. A dotted construction line is also placed at this angle to the starting angle. The value displayed in the VCB is referred to as the *angular rotation value*.



**Note** - Set the angle manually by entering a value in the VCB, and pressing Enter. The value can either be in decimal degrees, such as **34.1**, or slope, such as **1:6**. This value can be changed any number of times before proceeding to the next command.

## Locking the Protractor

Use the Shift key to lock the protractor to its current orientation.

## Entering Exact Angle Values

The degree of rotation you have indicated appears in angular degrees in the VCB while creating construction lines using the Protractor Tool. You can also manually enter in angular rotation or slope values directly into the VCB while measuring an angle and setting a construction line.

### Entering an Angular Rotation Value

To specify an exact angle in degrees, type a decimal value into the VCB while rotating the cursor around the protractor. For example, typing in **34.1** will give you an exact 34.1 degree angle. Negative values will move angle in a counter-clockwise direction. You can specify an exact angular value either during or immediately after your rotation operation.

### Entering a Slope Value

To specify a new angle as a slope, type in the two values separated by a colon in the VCB, such as **8:12**. Negative values will move angle in a counter-clockwise direction. You can specify an exact angular value either during or immediately after your rotation operation.



**Note** - SketchUp can handle up to 0.1 of a degree of angular precision.

# Axes Tool

The Axes Tool is used to move, or reorient, the drawing axes within your model. For example, you might want to move the axes when you are constructing rectangular objects that are skewed relative to one another. Or, you might use this tool to allow for more accurate scaling of objects that are not oriented along the default coordinate planes. Activate the Axes Tool from either the Construction Toolbar or the Tools menu.

## Moving the Drawing Axes

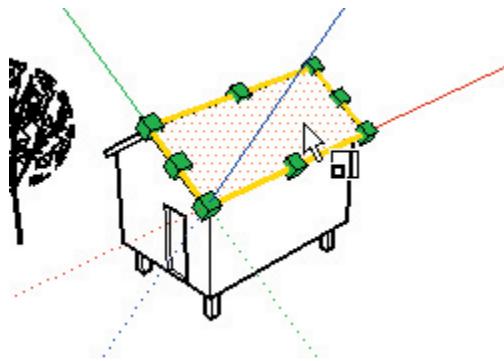
To move the Drawing Axes:

1. Select the Axes Tool. The cursor changes to a collection of axes.
2. Move your cursor to a point in your model that you wish to be the new coordinate origin. You will see the axes snap to inferred alignments and points as you move near them around your model. Use the inference ToolTips to make sure your cursor is located exactly where you want it to be.
3. Click to accept that point.
4. Drag your cursor away from the origin to set an alignment for the red axis. Use the inference ToolTips to make sure you are aligned precisely.
5. Click to accept the alignment.
6. Move your cursor away from the origin to set an alignment for the green axis. Use inference ToolTips again to make sure you are aligned precisely.



**Note -** Press the *ESC* key at any point during the operation to start over.

7. Click again to accept the alignment.



You have moved your axes. The blue axis will appear perpendicular to the new red/green plane.



**Note -** Moving the drawing axes does not change the true ground plane for shadows and ground/sky display.

## Resetting the Drawing Axes

Right click on the drawing axes and select Reset from the context menu to restore the axes to the default position.

# Dimensions Tool

The Dimension Tool is used to place Dimension entities in your model. Activate the Dimensions Tool from either the Construction Toolbar or the Tools menu.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Placing Linear Dimensions

Dimensions in SketchUp are based on a 3D model. Edges and points can be used to take dimensions. Suitable points include: end points, midpoints, on edge points, intersections, and arc and circle centers. Dimension leaders can be adjusted to span between non-linear points in a model so that your dimensions are as useful as possible in 3D. To take a dimension between two points in your model:

1. Select the Dimension Tool. The cursor changes to a arrow.
2. Click to set the starting point of your dimension.
3. Move the mouse in the direction you want to dimension.
4. Click a second time on the endpoint of your dimension.
5. Move the cursor to pull a dimension string out form the model.



**Note** - Press the *ESC* key at any point during the operation to start over.

6. Click the mouse a third time to position the dimension string.



**Tip** - You can take a dimension of a single line simply by clicking on the line and moving the cursor.



**Tip** - You might need to occasionally rotate your model, as you dimension, to position the dimension to the proper plane.

The appearance of all Dimensions are set and controlled from the Dimension panel of the Model Info dialog box. These settings affect all dimensions already in the model.

## Dimension Plane

You can take dimensions in one of several planes. These include the current axial planes (red/green, red/blue, red/green) or the plane aligned to the edge you are measuring. Radius and Diameter dimensions are limited to the plane defined by the arc or circle. Once a dimension is taken and placed within a plane, it can only be moved within that plane.

## Placing Radius Dimensions

To place a Radius Dimension on an Arc entity:

1. Activate the Dimension Tool. The cursor changes to a arrow.
2. Click on an Arc entity.
3. Move the cursor to pull a dimension string out form the model.
4. Click the mouse a third time to position the dimension string.

## Placing Diameter Dimensions

To place a diameter dimension on a Circle entity:

1. Activate the Dimension Tool. The cursor changes to a arrow.
2. Click on an Circle entity.
3. Move the cursor to pull a dimension string out form the model.
4. Click the mouse a third time to position the dimension string.

### Diameters to Radius, Radius to Diameter

Right click on the dimension and choose Type > Radius or Diameter, to change a radius dimension to a diameter dimension (or a diameter dimension to a radius dimension).

# Text Tool

The Text Tool is used to insert Text entities into your model. Activate the Text Tool from either the Construction toolbar or the Draw menu.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

There are two types of text in SketchUp: Leader Text and Screen Text.

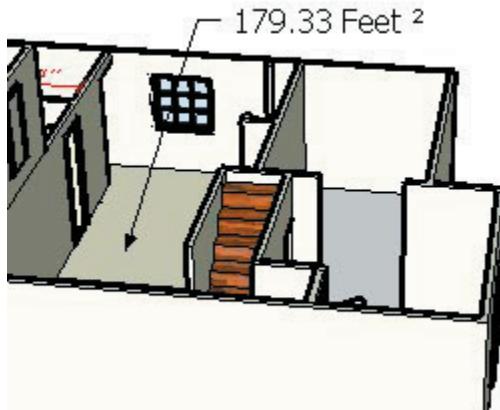
## Creating and Placing Leader Text

Leader Text contains characters and a leader line that points (refers) to an entity. To create and place leader text:

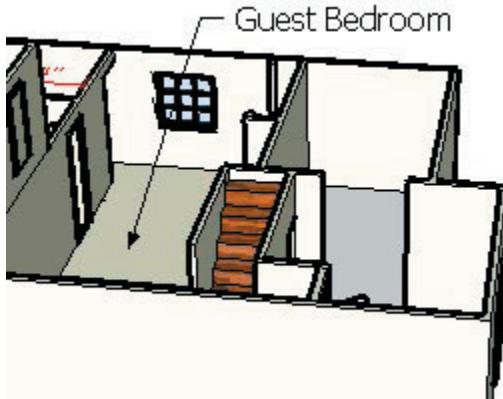
1. Select the Text Tool. The cursor changes to an arrow with a text prompt.
2. Click on any entity to indicate the ending point of the leader line (the location where the leader should point).
3. Move your mouse to a position on the screen where you want the text that refers to the entity to appear. The leader line will grow and shrink as you move your mouse around the screen.
4. Click again to position the text. A text entry box appears with default text, such as the name of a component (if the ending point of the leader line is attached to a component), or the square footage of a square (if the ending point of the leader line is attached to the face of a square).



**Tip** - Double-click on the ending point of the leader line to fix a hidden leader line to the entity.



5. Enter text in the text entry box.



**Note** - Press the *ESC* key at any point during the operation to start over.

6. Click outside of the text box, or press the Enter key twice, to complete text entry.



**Tip** - Press the *ESC* at any time to cancel creation of the Text entity.



**Note** - There are two styles of leaders: View Based and Pushpin. A View Based leader will always retain its 2D screen orientation. A Pushpin leader is aligned in 3D space, and rotates with your model as you change your view. You can specify which type of leader is used from the Text panel of the Model Info dialog box.



**Tip** - Double-click on any face, while in the Text Tool, to display the area of the face as a Text entity.

## Placing Screen Text

Screen Text contains characters and is not associated with an entity. To create and place screen text:

1. Select the Text Tool. The cursor changes to an arrow with a text prompt.
2. Move your mouse to a blank area on the screen where you want the screen text to appear.
3. Click to position the text. A text entry box appears.
4. Enter text in the text entry box.
5. Click outside of the text box, or press the Enter key twice, to complete text entry. Screen text will stay fixed on the screen regardless of how you manipulate and orbit the model.

## Editing Text

Double-click on text , with the Text Tool or Select Tool active, to edit the text. You can also right-click on a Text entity and select the Edit Text menu item from the Text entity's context menu.

## Text Settings

Text entities are created using the settings found in the Text panel of the Model Info dialog box. Refer to the Text entity topic for further information.



**Note** - Text entities can have different attributes (font, size, and so on) while dimension settings are global.

# Section Plane Tool

The Section Plane Tool is used to create Section Cut Effects within your models using Section Plane entities and section slices. Activate the Section Plane Tool from the Construction toolbar or the Tools menu.



**Video** - Several video tutorials on this topic are available at  
<http://www.sketchup.com/training/tutorials.php>

## Creating Section Cut Effects

1. Select the Section Plane Tool. The cursor changes to a pointer with a section plane.
2. Move the section plane around to align to the proper orientation in your model. The Section Plane Tool will shift its orientation as it aligns with nearby axes and geometry. You can lock in a particular orientation by holding down the Shift key when the section plane is in your desired orientation.



**Note** - Press the *ESC* key at any point during the operation to start over.

3. Click to place the section plane entity, create a section slice, and create a resulting section cut effect.



**Note** - A section plane will not work on a selection set (you cannot preselect only those items you want to slice). Instead, the section plane will create a section slice in all entities in the current context and, therefore, the slice will expand to cover all entities in the context.

## Manipulating Section Planes

You can use the Move Tool and Rotate Tool to reposition section planes just as you reposition other entities. Additional methods for manipulating section planes follows.

### Reverse Cutting Direction

The direction of a section plane can be reversed by right clicking on the section plane and selecting reverse from context menu.

### Changing The Active Section Plane

Newly placed section planes are active until another entity, such as another section plane, are selected.

There are two ways to activate a section plane: double-click on the section plane with the Select tool or right click on the section plane and select Activate from the context menu.



**Note** - One section plane can be active for each context in your model. Therefore a section plane within a group or component can be active at the same time, because they are in separate contexts, as a section plane outside of any group or component. A model that has a group that also contains two other groups has four different contexts (one context outside of any group, one context inside the top level group, and one context each for the groups contained within the top-level group), and can have four active sections at once.

## Creating Grouped Section Slices

1. Right click on a Section Plane, then select Create Group from Slice from the context menu.
2. This will generate new edges, encapsulated within a group, wherever the Section Plane intersects with faces.

This Group may be moved off to the side as a section outline, or it may be immediately exploded, making the edges merge with the geometry from which they were generated. This technique allows you to quickly make slices through any complex shape.

## Exporting Sections

Sketchup allows you to export section cut effects (your model with a section slice) and section slices themselves.

### Exporting Models with Section Cut Effects

Exporting models with section cut effects is the equivalent of exporting any model as a bitmap image file. Section planes and section cuts will appear in the exported file depending on the visibility of the section plane and section cut.

### Section Slice...

SketchUp can also export section slices using the Section Slice export feature. 2D vector sections can be accurately scaled and measured similarly to other 2D vector exports.

## Using Sections with Pages

Active Section Plane may be saved to a page. Section cut effects will animate during presentations using SketchUp's TourGuide technology.

## Align View

Use the Align View command from the Section Plane Context menu to re-orient the model view to a view perpendicular to the section plane. Use this command, in conjunction with Paraline mode, to quickly generate sectional elevation or 1-point perspective views of your model.

# Camera Tools

This section of the user's guide covers the camera tools within SketchUp. Camera tools are those tools used to manipulate your point of view of the model. These tools include the Orbit tool, Pan tool, and zoom tool.

Tools in this section are covered in the order they appear within the Camera menu.



## Previous

The Previous option is used to return to the last view of your model. Previous can be used after using the Orbit Tool, Pan Tool, Position Camera Tool, Look Around Tool, or any of the Zoom tools. Activate the Previous option from either the Camera toolbar or the Camera menu.

# Standard Views

SketchUp provides several pre-defined standard point of views for your convenience. These point of views are (from left-to-right in the image below): Top, Front, Right, Left, Back, Isometric, and Bottom. Select a Standard View from Views Toolbar or the Camera menu.



**Tip** - The standard views can be used with a paraline projection to produce elevation, plan, and section drawings.

## Printing Standard Views in Perspective Mode

Traditional rules of perspective apply when you are in Perspective mode and want to print or export your model as a 2D vector drawing, (the output will not be to scale as it is in the SketchUp drawing area). For example, the Top and Isometric views will reorient your view in a similar way, but cannot create a true plan or isometric drawing unless SketchUp is in Paraline mode.



**Note** - You are in a perspective projection when the Camera > Perspective menu item has a checkmark.

## Using the Isometric (Iso) View

SketchUp will move your view to the true Isometric view closest to your current view angle when you activate the Isometric View. Use the Orbit Tool to orient the camera approximately where you want the view to be to change to a different Isometric view. Finally, activate the Isometric view to lock into an actual Isometric view. (Remember that true Isometric views require Paraline mode.)

# Orbit Tool

The Orbit Tool is used to rotate the camera, in three dimensions, about the model. The Orbit Tool is useful when viewing geometry from the outside. Activate the Orbit Tool from either the Camera toolbar or the Camera menu.

## Orbiting the View

The Orbit Tool performs a 3D orbit. To orbit using the Orbit Tool:

1. Select the Orbit Tool. The cursor changes to two inner-connected perpendicular ovals.
2. Click the left mouse button.
3. Move your mouse in any direction to rotate around the center of the drawing area.



**Tip - Double-click the left mouse button on your model to center to the model in the drawing area.**

## Using Mouse Enhancements and Modifier Keys When Orbiting

The Orbit Tool is used heavily when creating and editing models. Therefore, SketchUp contains several mouse enhancements and modifier keys to allow easy access to the Orbit Tool.

### Activating the Orbit Tool While in Another Tool

Click and hold middle mouse button on a three-button mouse to temporarily activate the Orbit Tool while in any other tool (except the Walk Tool).



**Tip - You will also be put in pan temporarily if you click on the middle-mouse wheel and then click and hold the left mouse button or press and hold the Shift key.**

### Suspending the Gravity Setting

The Orbit Tool is designed to maintain a sense of gravity by keeping vertical edges pointed up and down. Press and hold the Ctrl key during orbit to suspend this gravity setting and to roll the camera on its side.



# Pan Tool

The Pan Tool is used to move the camera vertically and horizontally along the picture plane. Activate the Pan Tool from either the Camera toolbar or the Camera menu.

## Panning the View

The Pan Tool performs a pan operation. To pan using the Pan Tool:

1. Select the Pan Tool. The cursor changes to a hand.
2. Click the left mouse button.
3. Move your mouse in any direction to pan.

## Panning While in the Orbit Tool (3 Button Mouse)

Press and hold the Shift key while in the Orbit Tool to temporarily activate the Pan Tool. Or, press and hold the left mouse button while pressing and holding the middle mouse button to activate the Pan Tool.

# Zoom Tool

The Zoom Tool is used to interactively zoom from the current point of view. Activate the Zoom Tool from either the Camera toolbar or the Camera menu.

## Zooming In and Zooming Out of Your Model

To zoom in and out using the Zoom Tool:

1. Select the Zoom Tool. The cursor changes to a magnifying glass with a plus and minus sign.
2. Click and hold the left mouse button anywhere in the drawing area.
3. Drag the mouse up and down to zoom in (closer to the model) and out (farther from the model) respectively.

## Zooming With a Scroll Wheel Mouse

Rolling a wheel forward on a scroll wheel mouse zooms in on your model. Rolling the wheel backward on a scroll wheel mouse zooms out from your model.



**Note -** *The cursor position determines the center of zoom when using the scroll wheel whereas the center of the screen determines the center of zoom when zooming using the left mouse button.*

## Centering the Point of View

Double-click the left mouse button on your model to center to the model in the drawing area.

## Perspective Adjustment (Field of View)

Adjust the screen to an exact perspective or camera lens by typing an exact value in the VCB while the Zoom Tool is active. For example, "45 deg" sets a 45 degree Field of View and "35 mm" sets the equivalent focal length of a 35mm camera. You can also visually adjust the camera lens or Field of View by pressing and holding down the Shift key while using the Zoom Tool. Remember, changing the FOV keeps the camera in the same location in 3D space.



# Zoom Window Tool

The Zoom Window Tool is used to zoom in on a portion of your model. Activate the Zoom Window Tool from either the Camera toolbar or the Camera menu.

## Zooming in on a Portion of Your Model

The Zoom Window Tool allows you to draw a rectangular zoom window around a portion of your model. The Zoom Window Tool will then zoom in on the content within this zoom window. To zoom in on a portion of your model.

1. Select the Zoom Window Tool. The cursor changes to a magnifying glass with small square.
2. Click and hold the left mouse button a short distance away from the entities you want to see in the zoom window. This is the zoom window starting point.
3. Drag the mouse to the opposite corner of the zoom window starting point.
4. Release the mouse button when all of the elements are enclosed within the zoom window. These entities will fill the screen.

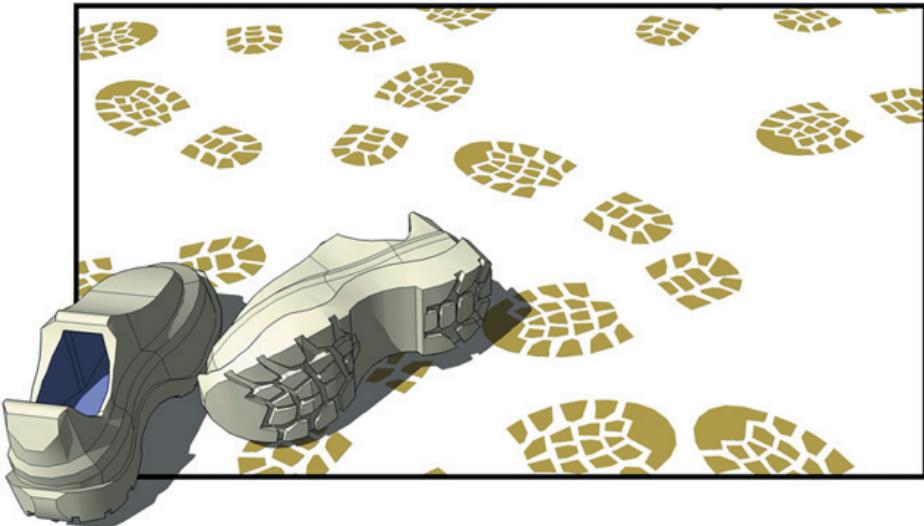
## Zoom Extents Tool

The Zoom Extents Tool is used to zoom your view to a distance which makes the whole model visible and centered in the drawing window. Activate the Zoom Extents Tool from either the Camera toolbar or the Camera menu.

# Walkthrough Tools

This section of the user's guide covers the walkthrough tools within SketchUp. Walkthrough tools are those tools used to view your model as though walking around and in your model. These tools are the Position Camera Tool, Look Around Tool, and Walk Tool.

Tools in this section are covered in the order they appear within the Camera menu.



# Position Camera Tool

The Position Camera Tool is used to position the camera at a certain eye height such that you can check the line of sight of a model or walk through the model. Activate the Position Camera Tool from either the Walkthrough toolbar or the Camera Menu.

## Positioning the Camera

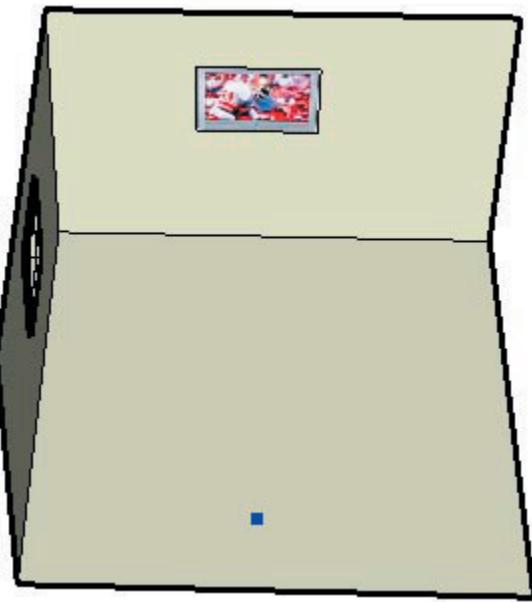
The Position Camera Tool is set using one of two methods. The first method approximates an eye-level view. The second method performs more precise camera positioning.

### Positioning the Camera at an Eye-Level View

The first position camera method allows you to position the camera at a specific eye-height above a specific point that you choose. The camera does not point at anything in particular. However, you are placed in the Look Around Tool allowing you to move the camera around the point to look at items in your model.

1. Select the Position Camera Tool. Notice that the Value Control Box (VCB) indicates that the eye height above the ground plane is set to 5 feet, 6 inches. You can override this height at this time by typing in a desired value.

2. Click the left mouse button on a point in your model. SketchUp places the camera's point of view at an average eye-height over the point you click on. You are also placed in the Look Around Tool. The following image shows a point in the middle of a room. The camera will be positioned directly above this point (5' 6") facing the television set if you click at this point.



**Note** - The viewing direction defaults to the top of screen, which is due north, if you place the camera from a plan view.

3. Use the Look Around Tool to look around the model.

### Positioning the Camera Using Specific Target Points

The second position camera method allows you to position the camera at a specific point that you choose and facing a specific direction in your model.

1. Select the Position Camera Tool. The cursor changes to a camera with a tripod. Notice that the Value Control Box (VCB) indicates that the eye height above the ground plane is set to 5 feet, 6 inches. You can override this height at this time by typing in a desired value.

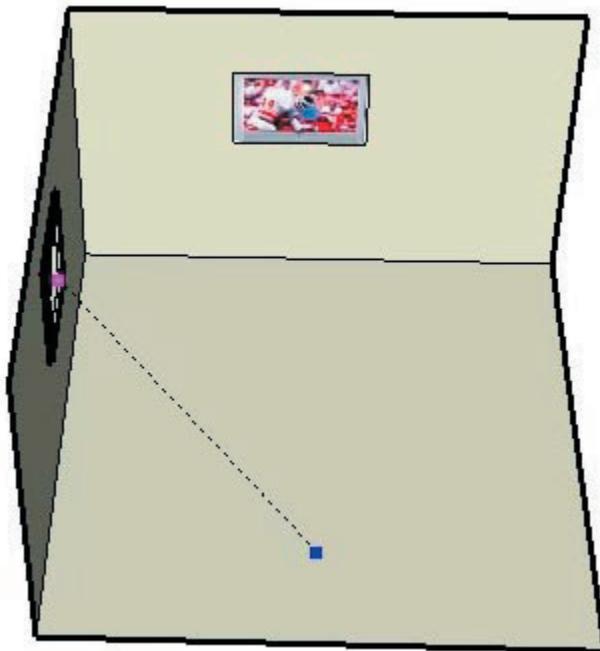


**Tip** - Use the Tape Measure Tool and the VCB to drag parallel construction lines off of edges as a method to provide accurate camera placements.

2. Click and hold left mouse button on a point in your model.
3. Drag the mouse cursor to the portion of the model that you want to look at. A dotted line is extended from the point selected in step 2 to the portion of the model you want to look at.

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4. Release the mouse button. The camera is repositioned at a height of 0 at the point selected in step 2. The camera faces the item you dragged the cursor to in step 3. The following image shows a point in the middle of a room with a dotted line to the window on the left wall of the room. The camera will be positioned directly at the first point (at 0 height) facing up at the window if you release the mouse button on the window.



5. (optional) Type a new eye height into the VCB to reposition the camera at eye height above the point selected in step 2.



**Tip** - Position the camera directly horizontal to the model to achieve a 2 Point Perspective.



**Tip** - Hold the Shift key while clicking on a surface to position the camera directly on the surface.

# Walk Tool

The Walk Tool is used to maneuver through your SketchUp model as though you were walking through your model. Specifically, the Walk Tool fixes the camera to a particular height, and then allows you to maneuver the camera around your model. The Walk Tool is available only in Perspective mode. Activate The Walk Tool from either the Walkthrough toolbar or the Camera menu.

## Using the Walk Tool to Tour Your Model

The Walk Tool is primarily used to take an interactive walk through or around your model. To Use the Walk Tool to tour your model:

1. Select the Walk Tool. The icon changes to a pair of shoes. Notice that the VCB indicates that the eye height above the ground plane is set to 2 feet, 10 1/2 inches. You can override this height at this time by typing in a desired value.
2. Click and hold the mouse button anywhere in the drawing area. A small plus sign (cross hair) is placed at the location where you clicked.
3. Move your mouse up (forward), down (backward), left (to go left) or right (to go right) to walk through or around your model. The further you are from the cross hair, the faster you will go.



**Note** - Press and hold the Shift key while moving the cursor up and down allows you to move up or down instead of forward or backward. Press and hold the Ctrl key to run instead of walk. Press and hold the Alt key to avoid collision detection (walk through walls).



**Tip** - It is often desirable to tour a model with a wide field of view. Activate the Zoom Tool, press and hold the Shift key, and press and drag up and down to widen your field of view prior to walking through a model.

### Looking around While in the Walk Tool

Click and hold the middle mouse button, while using the Walk Tool, to quickly turn your viewpoint using the Look Around Tool.

### Ascending and Descending with the Walk Tool

The Walk Tool is designed to allow you to walk up and down inclines and step up on tables and chairs while maintaining eye height.

### Turning Off Collision Detection

Press the Alt key while walking around to temporarily turn off collision detection. This option is useful when examining models such as mechanical devices or furniture (anything other than the interior of a building).

# Look Around Tool

The Look Around Tool is used to pivot the camera around a stationary point at the point of view. The Look Around Tool behaves similarly to a person standing still while turning their head to up, down (*tilt*), and side to side (*pan*). The Look Around tool is particularly useful for viewing the inside of spaces, or to evaluate visibility after using the Position Camera Tool. Activate the Look Around Tool from either the Walkthrough toolbar or the Camera menu.

## Looking Around

The Look Around Tool performs tilt and pan operations. To tilt and pan using the Look Around Tool:

1. Select the Look Around Tool. The cursor changes to an eye with four arrows.
2. Click the left mouse button.
3. Move your mouse up or down to tilt; right or left to pan.

## Specifying an Eye Height

Type the eye height for the camera in the VCB and press the Enter or Return key while in the Look Around Tool to change the height above the ground plane for the camera.

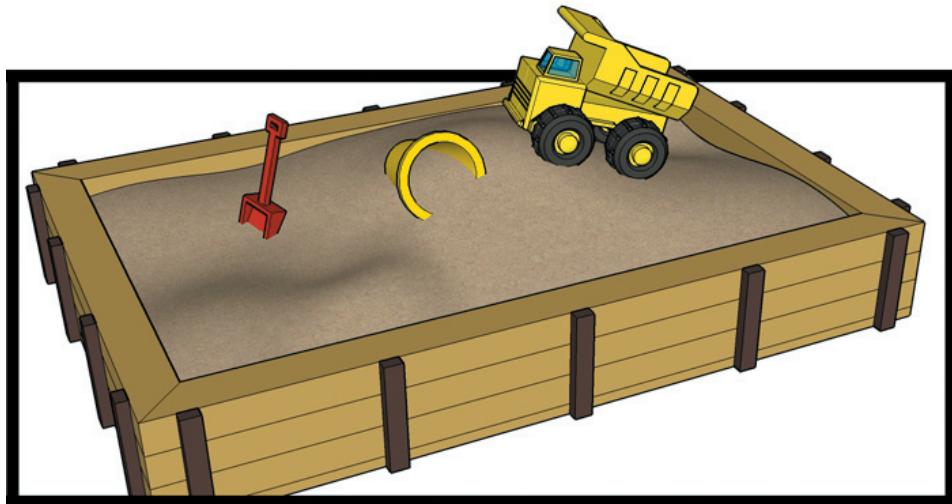
## Activating the Look Around While in the Walk Tool

Click the middle mouse button to activate the Look Around Tool while in the Walk Tool.

# Sandbox Tools

This section of the user's guide covers the sandbox tools within SketchUp. Sandbox tools are those tools used to create and manipulate large surfaces (TINs) in your models. These tools include the Sandbox From Scratch Tool, Sandbox From Contours Tool, Smoove Tool, Stamp Tool, and Drape Tool.

Tools in this section are covered in the order they appear within the Draw > Sandbox and Tools > Sandbox sub-menus.



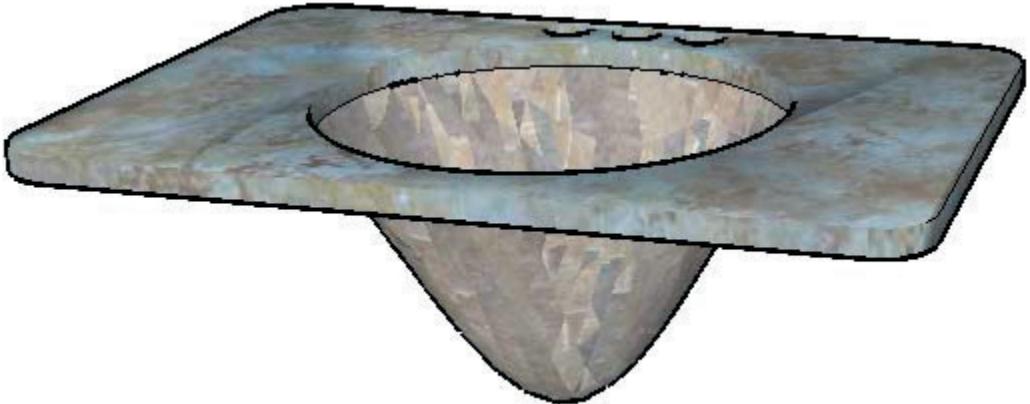
# Enabling the Sandbox Tools

The sandbox tools do not appear in SketchUp by default, but can easily be enabled using the Extensions Manager (the Extensions panel of the Preferences dialog box). To enable the sandbox tools:

1. Open the Extensions Manager (Window > Preferences > Extensions)
2. Click the check box next to the extension that you want to enable.
3. Click Ok.

## The Sandbox Tools and Organic Shapes

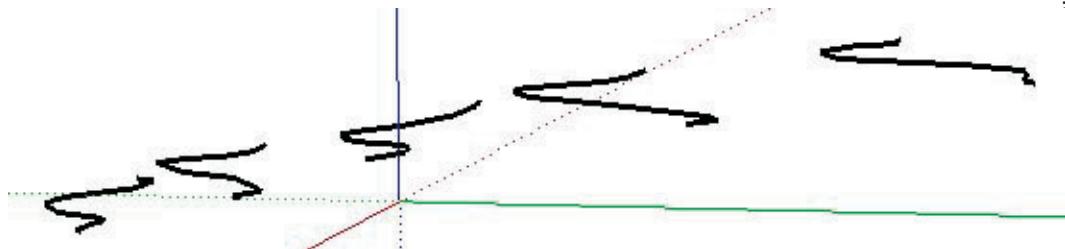
Sandbox tools are not just for modeling terrain, they can also be used to model other organic shapes such as the curved portions of a hand-built wooden chair or a wooden bowl. Organic shapes are those that have a hand-made or custom appearance. The bowl portion the following sink was created with the Smoove Tool.



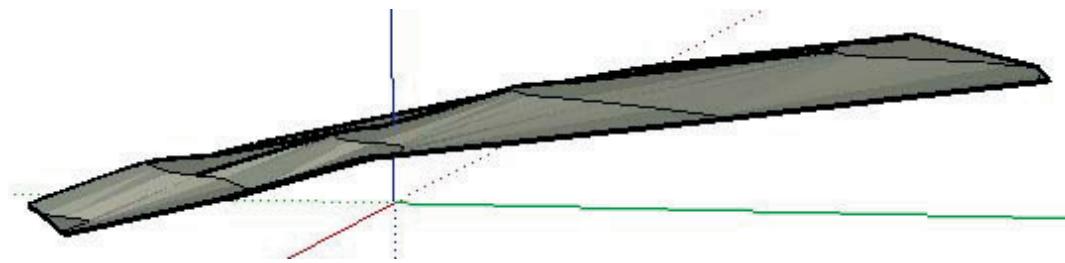
## Sandbox From Contours Tool

The Sandbox from Contours Tool is used to create a TIN from contour lines. You must create or import contour lines that are offset in their height prior to using this tool. Activate the Sandbox from Contours Tool from the Draw menu.

The following image shows several contour lines offset in height from the ground plane.



The following image shows the resulting terrain created when the Sandbox From Contours Tool is used on the selected contours.



**Note -** In some cases the Sandbox From Contour Line Tool might create triangulations in the TIN that result in flat spots or plateaus. These plateaus can be retriangulated (to create a slope) using the Flip Edge Tool.

### Creating Sandbox From Contour Lines

To create a sandbox from contour lines:

1. Import or draw several contour lines. Ensure that the contour lines are offset in their height from the ground plane.
2. Select all of your contour lines.
3. Select the Draw > Sandbox > From Contours. Terrain will be filled in using the contour lines as a guide.

# Sandbox From Scratch Tool

The Sandbox From Scratch Tool is used to generate a flat triangulated TIN that can be sculpted into other forms using other sandbox tools. The Sandbox From Scratch tool generates the TIN in the red/green or ground plane. This tool is useful in creating terrain when no other terrain model or data is available. Activate the Sandbox From Scratch Tool from the Draw menu.

## Creating a New TIN

To create a new TIN:

1. Select the Sandbox From Scratch Tool. The cursor changes to an arrow.

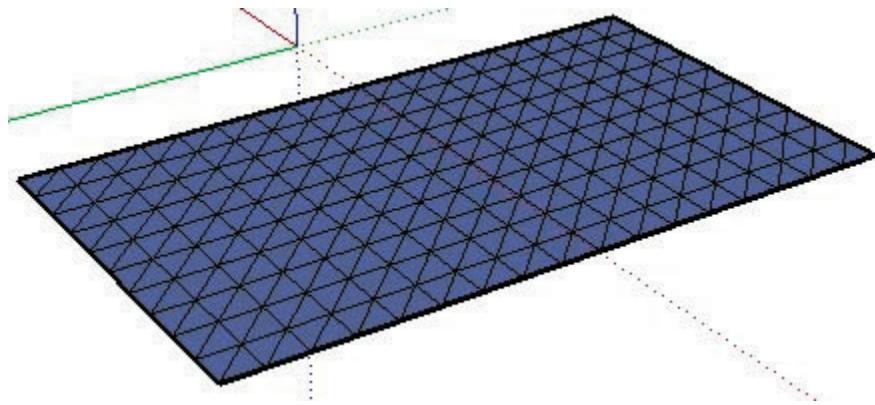


**Note** - The Sandbox from Scratch tool defaults to 10' square grid sizes. You might want to zoom out or change the grid size (Grid Spacing) in the VCB to before creating your sandbox.



**Tip** - You can optionally enter a value in the VCB for the size of each square in your grid at this time (Grid Spacing).

2. Click to set the starting point of your TIN.
3. Move the mouse in the direction you want to be the length of the TIN. As you move your mouse a line appears with tick marks at a specific spacing. The length value is displayed dynamically in the Value Control Box (VCB).
4. Click a second time to establish the length of your TIN.
5. Drag the mouse perpendicular to the length line to establish the width of your TIN.
6. Click a third time to establish the width of your TIN. The following image shows the flat TIN created with the Sandbox From Scratch Tool.



**Note** - The terrain will be placed inside of a Group entity. You must edit or explode the group to perform other operations, such as smoothing with the Smoove Tool, on the TIN.



## Specifying Precision Line Values

The VCB displays the length and width of your TIN while you are drawing a flat TIN. You can also specify a line length and width value using the VCB.

### Entering a Length Value

Specify a length and width in the VCB by typing in the desired length after placing the starting point of the flat TIN. SketchUp will use the current document units setting if you only type in a numerical value. You can also specify either Imperial (**1'6"**) or Metric (**3.652m**) units at any time, regardless the model units setting.

### Entering a Width Value

Specify a width and width in the VCB by typing in the desired length after setting the length of the flat TIN . SketchUp will use the current document units setting if you only type in a numerical value. You can also specify either Imperial (**1'6"**) or Metric (**3.652m**) units at any time, regardless the model units setting.

## Drawing a Flat TIN by Inference

The Sandbox From Scratch Tool uses SketchUp's sophisticated geometric inference engine to help you place your terrain in 3D space. The inference decisions, made by the inference engine, are displayed in the drawing area as inference lines and inference points. These lines and points show precise alignment between the line you are drawing and the geometry of your model. Refer to the inference engine topic for additional information.

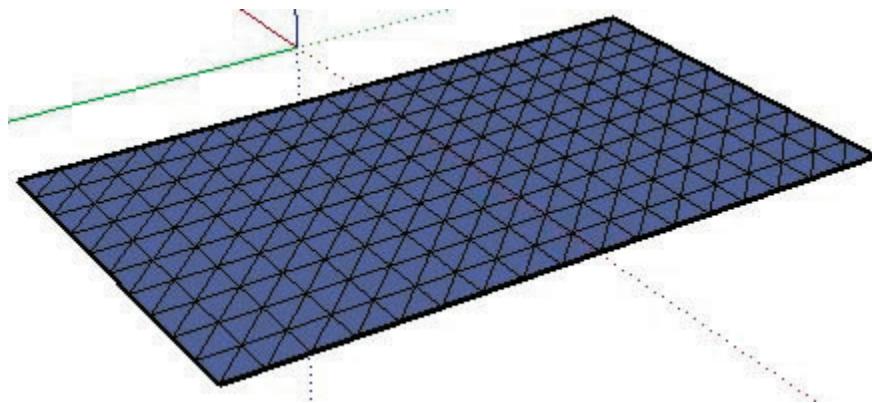
# Smoove Tool

The Smoove Tool is used to sculpt an existing TIN by allowing vertical movement of a selection of points, edges, or faces and their immediately adjacent geometry. The result is a smooth deformation of a triangulated surface. Activate the Smoove Tool from the Tools menu.

## Sculpting a Terrain

To sculpt a terrain:

1. Create, such as with the Sandbox From Scratch Tool, or import a TIN. The following image shows a flat terrain created with the Sandbox From Scratch Tool.

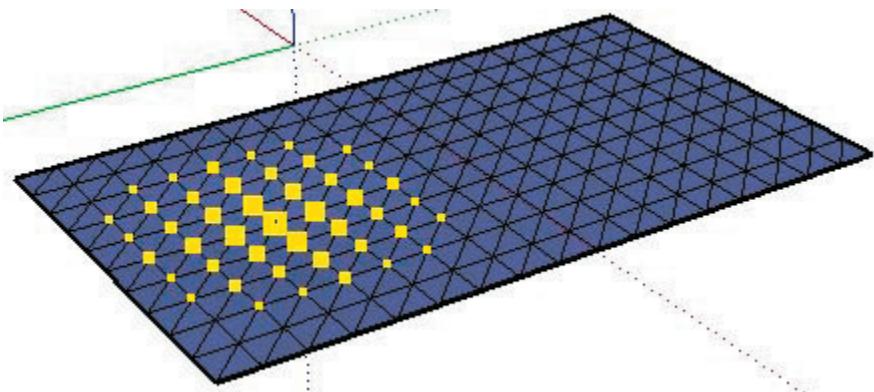


2. Select the Smoove Tool. The cursor changes to an arrow.



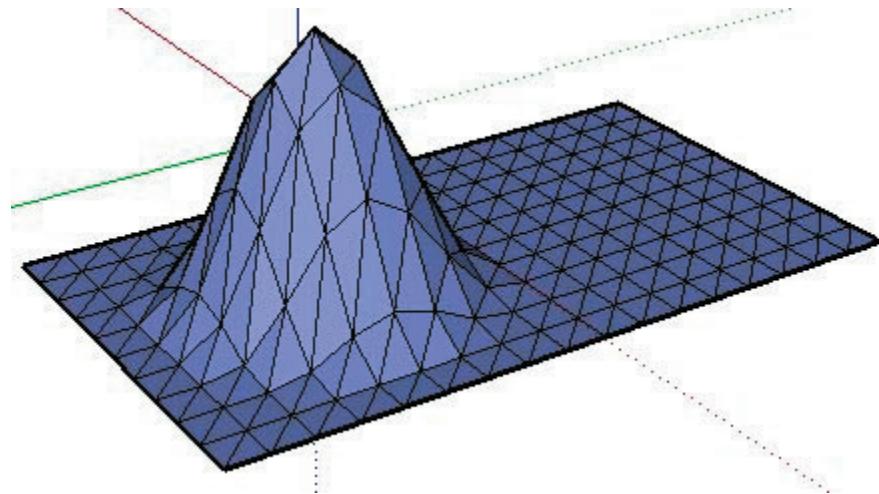
**Tip** - Alternatively, you can preselect a series of points, edges, and faces and then click on the Smoove Tool to begin a smoove operation.

3. Click on a point, edge, or face within the TIN. SketchUp highlights all neighboring vertices to be included in the sculpting operation within the radius specified in the VCB. The following image shows the point that was selected (the largest square in the middle) and neighboring vertices.



**Note** - Remember to ungroup a terrain created with the Sandbox From Scratch Tool prior to using the Smoove Tool.

4. Move the mouse up or down in the vertical (blue) direction to sculpt the TIN. The TIN will expand or contract accordingly. You can also type offset value in the VCB, such as 10'. The following image shows the terrain resulting from moving the point and neighboring vertices up.



**Note** - You can sculpt in a direction other than vertical by holding down the Shift key while moving the mouse. However, this operation can result in a non-functional terrain or terrain that can no longer be manipulated in a known way. This operation can only be performed while in active selection (not after preselection).

5. Smooth and soften the TIN to get your final model.

## Specifying Sculpting Radius Values

SketchUp determines the number of vertices included in the sculpting operation based on a radius value. The larger the radius the more vertices included in the sculpt. The VCB displays this radius while you are sculpting a TIN. You can also specify an radius value while sculpting, such as 10', using the VCB.



**Tip** - Preselect a series of points, faces, and edges within your TIN to sculpt more specific shapes such as ridges and berms.

## Specifying an Offset value

The VCB also specifies an offset while you are sculpting. You can also specify a an offset value, such as 10', using the VCB.

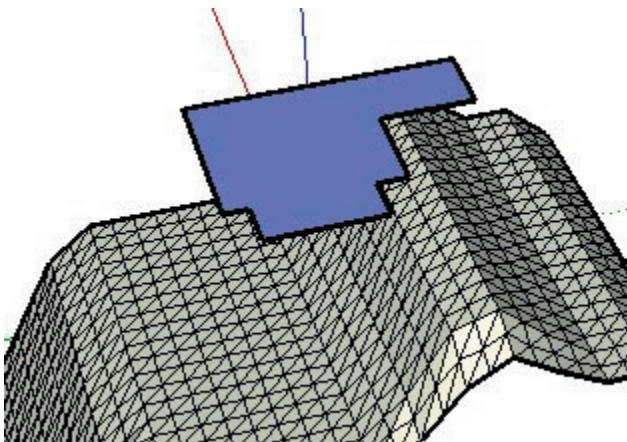
# Stamp Tool

The Stamp Tool is used to create impressions in a TIN by pushing a duplicate of the base of geometry, such as a house, into a TIN. Activate the Stamp Tool from the Tools menu.

## Creating Impressions of Geometry in a TIN

To create an impression of geometry in a TIN:

1. Create your TIN, such as the terrain where a building might exist
2. Create the faces to be used as the stamp, such as the bottom faces of the building.
3. Move the stamp over the location in the TIN where you want to create an impression.



**Note** - The stamp can be an individual face, selection set of faces, a group, or a component.

4. Select the Stamp Tool. The cursor changes to an arrow.
5. Click on the stamp. SketchUp will create an offset around the stamp.
6. (optionally) Adjust the offset size by typing in a new value in the VCB.
7. Click on the TIN.
8. A copy of the stamp will be indented into the TIN. The cursor will be on the copy of the stamp allowing you to move the stamp, and connected TIN, up or down.
9. Move the mouse to adjust the stamp within the TIN.

10. Click when you are done adjusting the stamp.
11. Smooth and soften the TIN to get your final model.



**Note** - The Stamp tool will not work on an group or component that is open for editing.

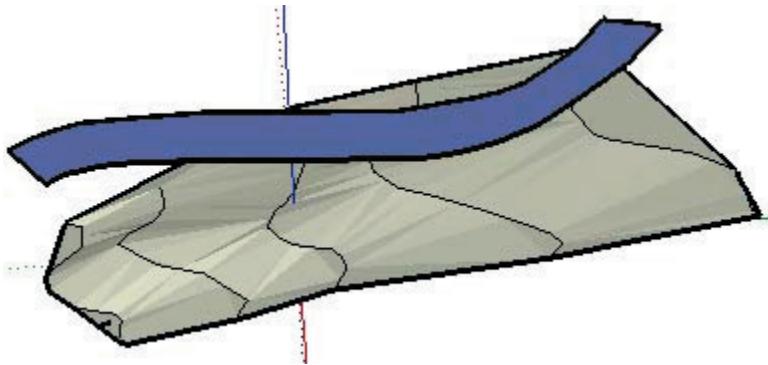
# Drape Tool

The Drape Tool is used to project edges, such as the edges in a road, onto a terrain. Activate the Drape Tool from the Tools menu.

## Draping Edges Onto a TIN

To drape edges onto a TIN:

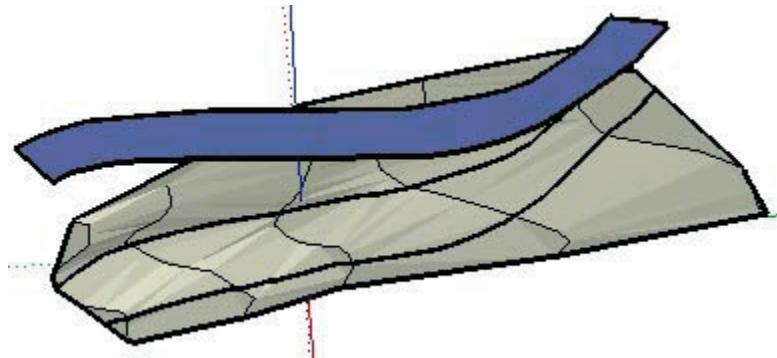
1. Create, such as with the Sandbox From Scratch Tool, or import a terrain.
2. Create the edges you want to drape over the TIN. The edges can be in a single plane (flat) and must be positioned above a TIN. The following image shows a 2D road positioned above a terrain.



**Note** - The Drape Tool will only drape the edges surrounding a face if you attempt to drape a face or series of faces over the TIN.

3. Select the edges that you want to drape over the TIN.

4. Select the Drape Tool. The cursor changes to an arrow.
5. Click on the TIN to drape the selected edges on the TIN. The following image shows the edges draped over the terrain.



**Note** - The draped edges will be outside of a group or component context if the TIN is encapsulated within a group or component.



**Tip** - This tool also works in active selection mode to select the drape edges if nothing is preselected.

# Add Detail Tool

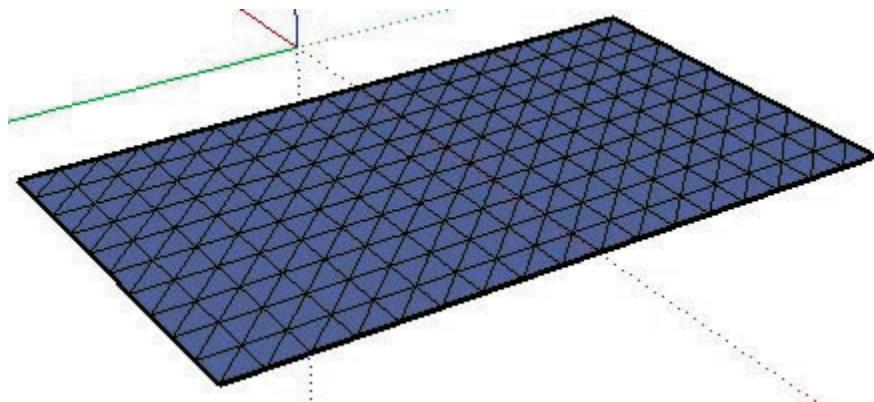
TINs do not have to consist of triangles of the same size. The Add Detail Tool is used to split triangles into additional, smaller triangles, and make small detailed modifications to your TIN. Any click on your TIN creates new triangles centered around a new vertex that you can manipulate. Activate the Add Detail Tool from the Tools menu.

Smaller triangles equate to a smoother final TIN. However, having an abundance of small triangles in your TIN can also decrease performance. The Add Detail Tool can help strike a balance between detail and performance by allowing you to add detail only to areas of the TIN where it is needed.

## Detailing a TIN

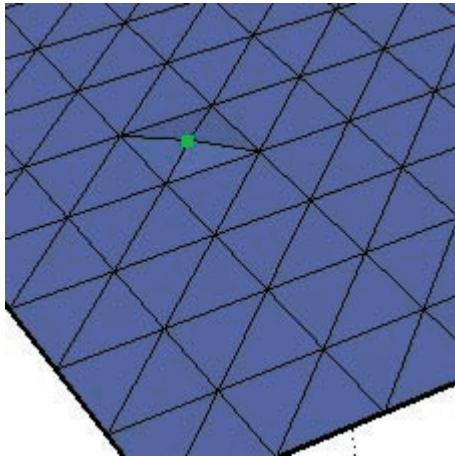
To detail a terrain:

1. Create a TIN. The following image shows a flat TIN or sandbox created with the Sandbox From Scratch Tool.

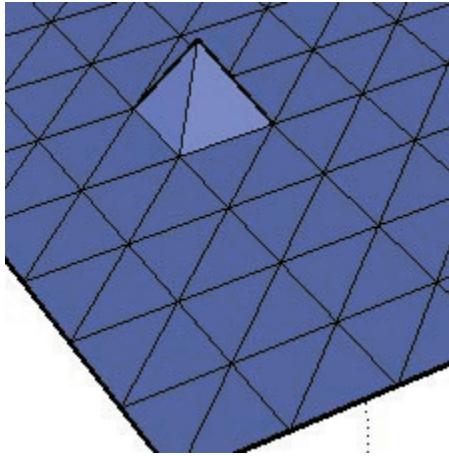


2. Select the Add Detail Tool. The cursor changes to an arrow.

3. Click on a point, edge, or face within the TIN. SketchUp creates a new vertex where you clicked with new triangles spanning out from the new vertex. The following image shows two additional triangles created when the midpoint of a line was selected.



4. Move your mouse up or down to adjust the height of the vertex and surrounding triangles. The following image shows the resulting TIN when the new vertex was lifted.



**Note** - You can also type in an offset distance to adjust the height of the vertex after clicking on a point, edge, or face within the terrain. For example, 10' will raise the new vertex 10' above the starting terrain.



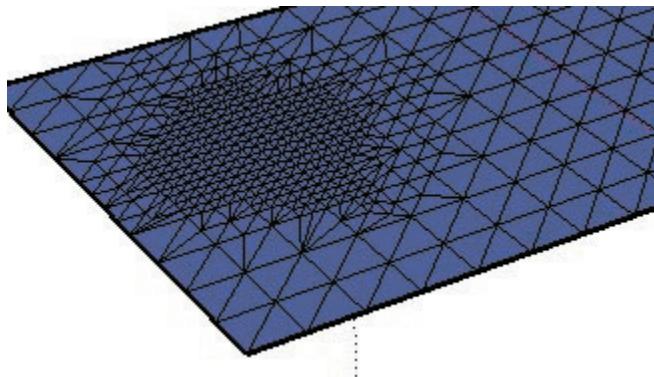
**Note** - You can sculpt in a direction other than vertical by holding down the Shift key while moving the mouse. However, this operation can result in non-functional terrain or

*terrain that can no longer be manipulated in a known way.*

5. Continue clicking on points, edges, or faces (either on the TIN you created in step 3, or on other portions of the TIN) to create additional triangles for manipulation.
6. Smooth and soften the TIN to get your final surface.

## Creating Smooth Areas

You can preselect a series of triangles in your model using the Select Tool and then use the detailer to divide those triangles to create a smoother area. The following image shows a flat TIN with a selection set that was detailed two times (creating several tiny triangles). This area of the model will appear very smooth when sculpted, using a tool such as the Smoove Tool.



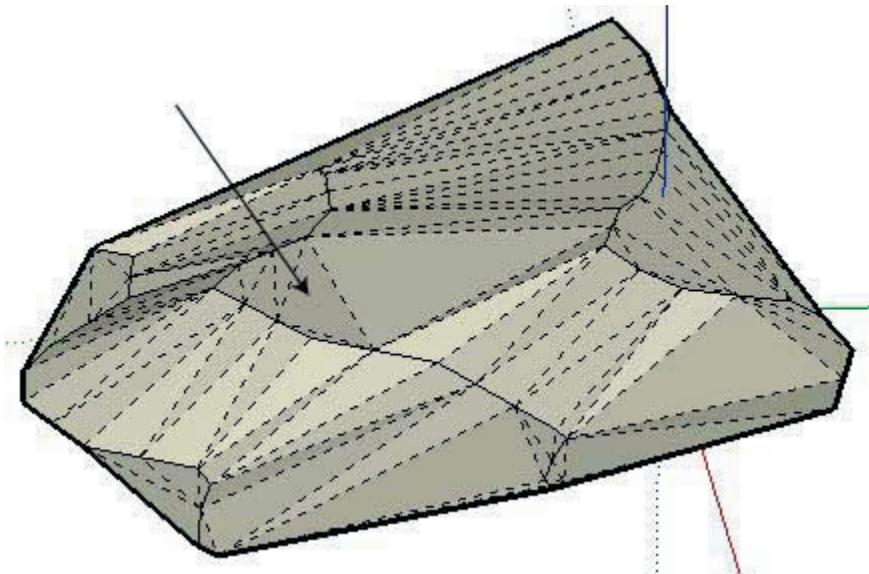
## Specifying an Offset value

The VCB also specifies an offset while you are detailing a TIN. You can also specify a an offset value, such as 10', using the VCB.

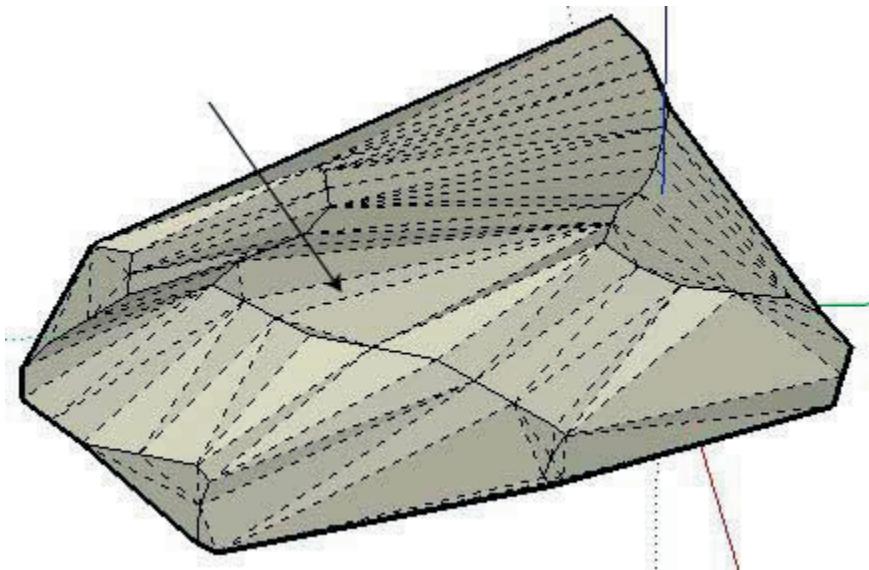
## Flip Edge Tool

The Flip Edge Tool is used to manually adjust the triangulation for any pair of adjacent triangles in a TIN. This tool is useful in removing flat spots or plateaus in a terrain generated from contour lines. Activate the Flip Edge Tool from the Tools menu.

The following image contains a terrain with several opposing triangulations (triangles that point opposite to each other).



Triangulations whose sides run north and south create plateaus in the TIN. The Flip Edge Tool can be used to flip these triangulations and remove these plateaus (creating slopes). The following image shows terrain with the triangulations flipped.



**Tip** - You can easily see triangles with opposing triangulation by displaying hidden geometry. Turn on hidden geometry (View > Hidden Geometry) to view the triangles in the TIN.

## Modifying TIN Triangulations

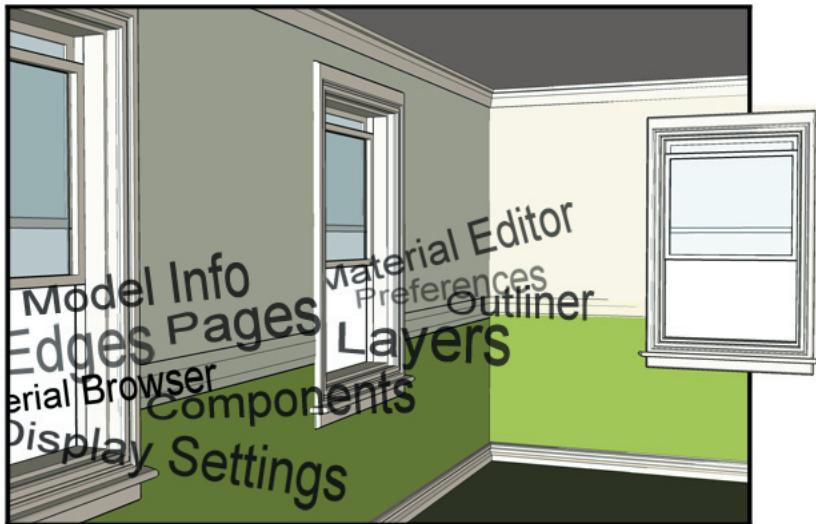
To modify TIN triangulations:

1. Create a TIN using a tool such as the Sandbox From Contours Tool.
2. Select the Flip Edge Tool.
3. Move the mouse over the TIN to highlight edges that can be flipped.
4. Click on any edge to flip the edge to an opposite triangulation. SketchUp analyzes the two adjacent triangles that share the edge, removes the selected edge, and replaces it with a perpendicular edge.

# Model Settings and Managers

This section of the user's guide covers the various model settings and manager dialog boxes within SketchUp. These dialog boxes include the Component Browser, Application Preferences, Material Browser, and Display Settings dialog boxes.

Each dialog box in this section is covered in the order they appear within the Window menu.



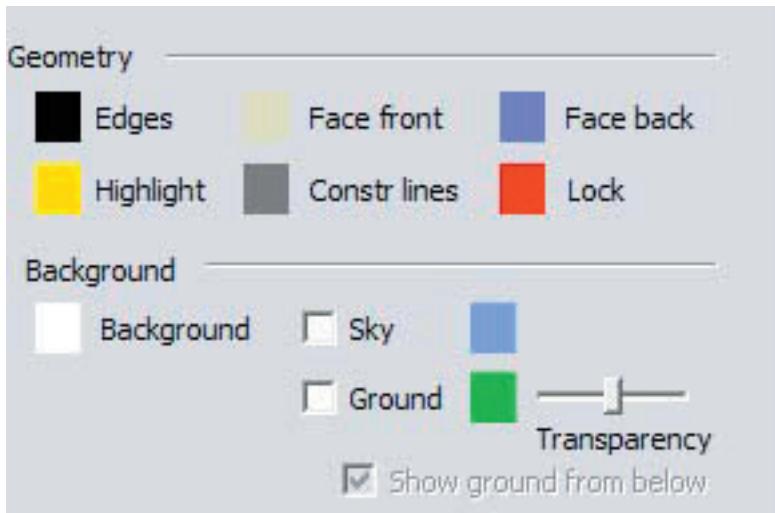
# Model Info Dialog Box

The Model Info dialog box is used to configure a number of different settings specific to your current SketchUp model. Activate the Model Info dialog box from the Window menu.

The Model Info dialog box contains several panels, including the Dimensions panel for setting global dimensions and the Units panel for setting units to be used in your model.

## Colors

The Colors panel is used to set default colors for several entities in the drawing area including default face and edge color. Click on any of the color wells to activate the Color Pickers and select a new default color for an entity.



### Geometry

The Geometry portion of the Colors panel contains default colors for geometrical entities in the Drawing Area.

**Edges:** The Edges option is used to set the default color for edges in SketchUp. You can force SketchUp to use this color for all edges (instead of a color you might have assigned manually) by setting Edge Color to All the Same in the Display Settings dialog box.

**Face Front:** The Face Front option is used to set the default color for all front sides of faces. Materials assigned to faces override this setting.

**Face Back:** The Face Back option is used to set the default color for the back sides of faces. Materials assigned to faces override this setting.

**Highlight:** The Highlight option is used to set the color of the highlight that is used to indicate a selection. Use a color that will contrast well with the other colors in your model when selecting a Highlight color.

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**Construction Lines:** The Construction Lines option is used to set the color of any Construction Line entities in your model.

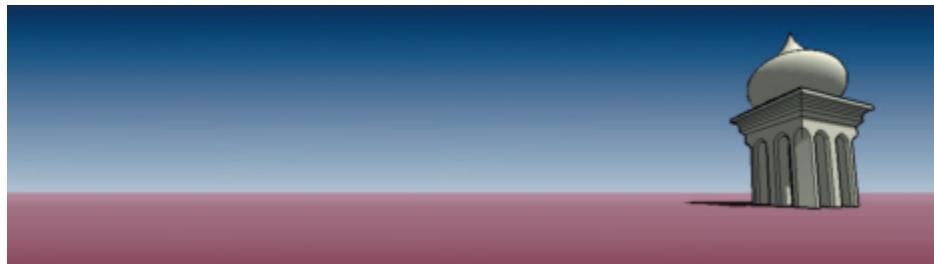
## Background

**Solid Color:** The Solid Color option is used to select a color for the entire background in the drawing area (both sky and ground). This option is commonly used when creating models other than architectural or geographic models..

**Gradient:** The Gradient option is used to select separate colors for both sky and ground planes. SketchUp will show the sky color as a gradient from the horizon up. SketchUp will show the ground color as a gradient from the horizon down.

**Sky:** The Sky option is used to select a separate color for the sky.

**Ground:** The Ground option is used to select a separate color for the ground.

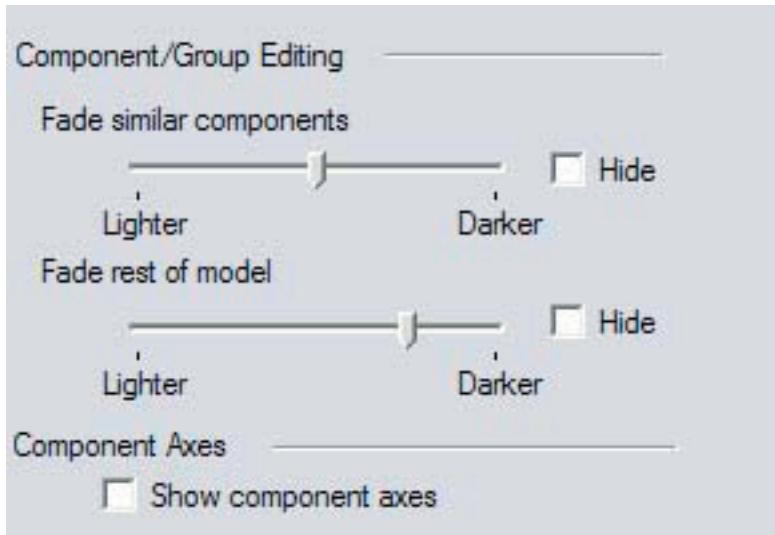


**Transparency:** The Transparency option is used to adjust the level of transparency for the ground plane. Place the slider toward the left-most position to minimize below-ground visibility. Place the slider toward the right-most position to maximize below-ground visibility.

**Show Ground from Below:** The Show Ground from Below option is used to toggle the display of the ground plane from a viewpoints below the horizon. Click the Show Ground from Below box to see the ground plane when your point of view is below the ground.

## Components

The Components panel is used to modify the visual appearance of your model while editing groups and components. SketchUp allows you to fade or hide other similar components or fade or hide the rest of the model to make it easier to concentrate on modifications to the edited component.



### Fade Similar Components

The Fade similar components option is used to modify the appearance of the other visible instances of the component you are editing. Click on the Hide check box to hide components similar to the one you are editing. Unclick the Hide check box to fade components similar to the one you are editing. You can control the degree of fading with the adjacent slider.

### Fade Rest of Model

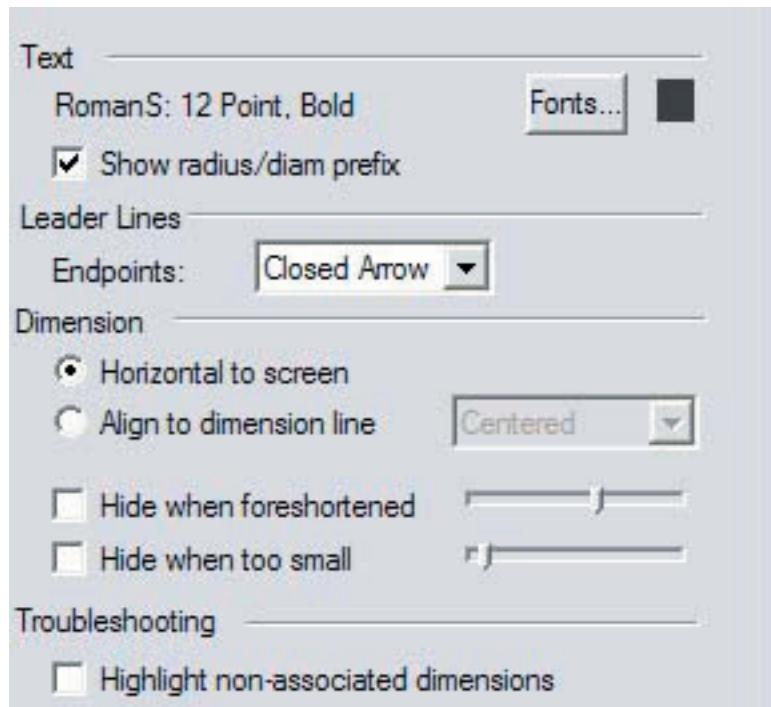
The Fade rest of model option is used to modify the appearance of the parts of your model unrelated to the group or component you are editing. Click on the Hide check box to hide the rest of the model while editing a component. Unclick the Hide check box to fade the rest of the model while you are editing. You can control the degree of fading with the adjacent slider.

### Show Component Axes

Components contain their own axes. These axes can be shown or hidden by clicking or unclicking the Show component axes check box.

## Dimensions

The Dimensions panel is used to change the appearance and behavior of Dimension entities in your model.



### Text

The Text portion of the Dimensions panel is used to select the font used by Dimension entities in the drawing area.

**Fonts...:** The Fonts... option is used to define the font to be used for all Dimension entities in SketchUp. Click on the Fonts... button to invoke the Font dialog box where you can choose the font, font style, and size for your dimensions font.

**Show radius/diam prefix:** The Show radius/diam prefix check box is used to toggle the visibility of a "R" and "DIA" prefixes for all radius and diameter dimensions, respectively.

## Leader Lines

Leader Lines can have different end point styles, such as a closed arrow. Select the end point style for all of your end points from the Endpoints drop-down list. These styles are None, Slash, Dot, Closed and Open.



## Dimension

The Dimension portion of the Dimensions panel contains default display options for Dimension entities.

**Horizontal to screen:** The Horizontal to screen option is used to allow Dimension entities to rotate as you orbit the model (always face the camera).

**Align to dimension line:** The Align to dimension line option is used to allow Dimension entities to align to the dimension line instead of the camera. As you orbit your model, the text will remain aligned with the dimension lines.

**Hide when foreshortened:** The Hide when foreshortened option is used to allow Dimension entities to hide when they become foreshortened. Use the slider bar to set the threshold angle that will cause Dimension entities to become hidden.

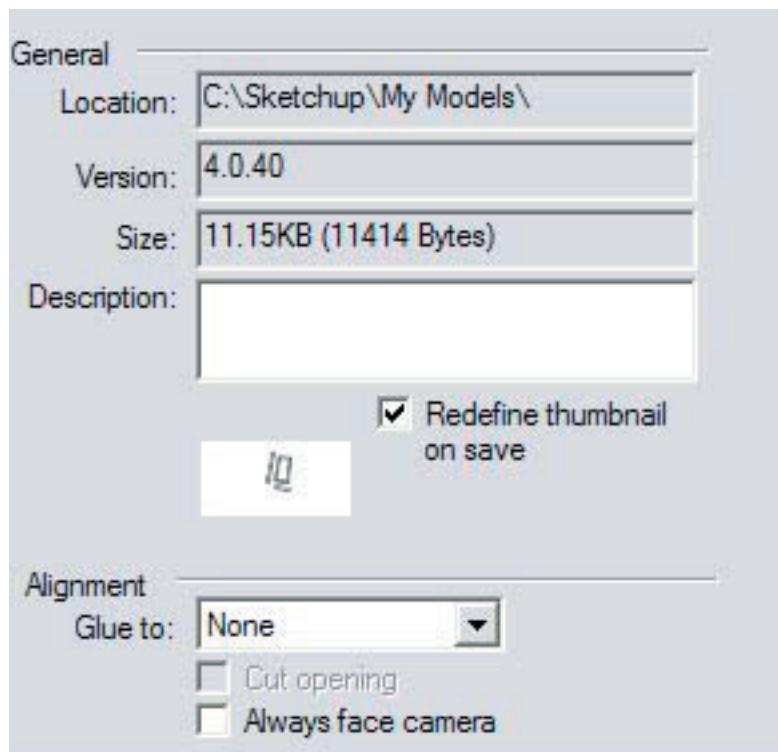
**Hide when too small:** As your view moves further away from your model, dimensions appear smaller and text remains the same size. This behavior can result in unclear Dimension text. The Hide when too small option is used to automatically hide dimensions when they are hard to read. Use the slider bar to set the size that will cause Dimension entities to be hidden.

## Troubleshooting

The Highlight non-associatied dimensions option is used to assign a color to dimensions that are not associated with entities in your model. Non-associated dimensions are usually the result of erasing an entity that was used to calculate a dimension, such as an edge.

## File

The File panel is used to configure settings related to your SketchUp model's file, including the location of the file (on your file system), size, date of last modification, and version of SketchUp used for last modification.



### General

The General portion of the File panel contains default file options for the current model. The Description field is the only field that is editable within the General section.

**Location:** The Location field contains the location of the model's file on the file system (if it has been saved).

**Version:** The Version field contains the version of SketchUp last used to modify the model.

**Size:** The Size field contains the Size, in kilobytes, of the current SketchUp model's file.

**Description:** The Description field contains a description of the model. Click in this field to enter a description for your model.

**Redefine Thumbnail on Save:** The Redefine Thumbnail on Save option is used to save the current view of the model as the thumbnail representation of the model. This thumbnail is shown when browsing through models when locating a file to open.

## Alignment

The Alignment options are used to determine the alignment of the model when loaded and used as a component.



*Note - These options are also displayed when you create a component from a selection set within SketchUp. However, the options presented during component creation pertain to components that are already within a model while the options in the Model Info dialog box pertain to a model file that can be loaded as a component in another model.*

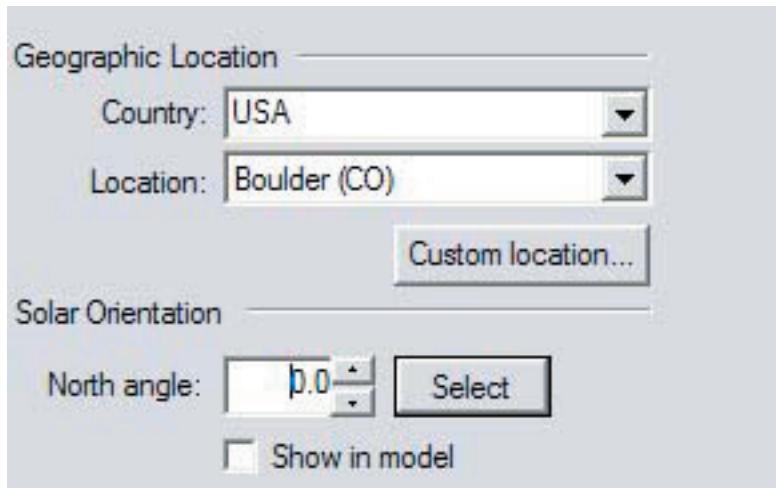
**Glue to:** The Glue to option is used to define the surfaces where your file can be placed. For example, a model of a door would only be glued to surfaces in the horizontal planes.

**Cut Opening:** The Cut Opening option is used to establish whether your model will be able to penetrate a face, thus creating an opening, when used as a component. The model will cut the face to which it is attached at the model's perimeter.

**Always face camera:** The Always face camera option is used to treat the model as though it were a 2D component by forcing the model to always face forward when loaded as a component. This option increases performance by eliminating the need to render the component as a 3D form.

## Location

The Location panel is used to specify a location for your model based on the nearest major city. This option is useful to determine accurate shading of architectural and geographical models.



## Geographic Location

The Geographic Location options are used to assign a geographic location to your model.

**Country:** The Country list box is used to assign a country of origin for your model. Select the country from the drop-down list box.

**Location:** The Location list box is used to assign a city of origin for your model. Select a city from the drop-down list box.

**Custom Location....:** The Custom Location dialog box is used to display the Set Custom Location dialog box. This dialog box allows you to manually enter Country, Location, Latitude, Longitude, and Timezone for any geographic location that is not already in the Country or Location drop-down list boxes.

## Solar Orientation

The Solar Orientation options are used to set the path of the Sun in relation to the axes.

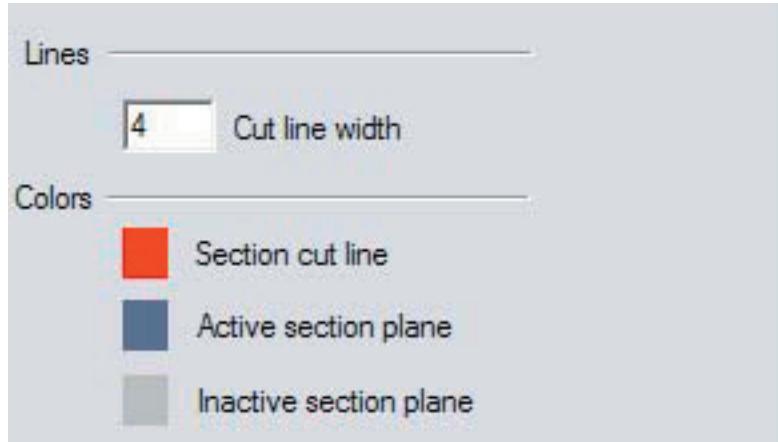
**North Angle:** The North Angle is used to establish the path of the Sun. East, West, and South can be computed by entering a North angle (0.0 by default for a model whose faces are directly aligned to the axes). Enter a new North angle if your north is at an angle to your model. Or, to determine a less precise North angle using the Select button:

1. Click on the Select button. The cursor changes to a cross within a circle.
2. Place the center of the cross at the origin of the North angle (usually the axes origin)
3. Click the left mouse button.
4. Drag the cursor away from the origin in the north direction
5. Click again to establish the new north angle.

**Show in Model:** The Show in Model option highlights the North angle in orange.

## Section Planes

The Section Planes panel is used to change the appearance and behavior of Section Plane entities in your model.



### Lines

The Cut Line Width option is used to define the thickness (in pixels) of all cut lines in the active section plane. Type in a number or click on the up and down arrows to modify the cut line width.

### Colors

The Colors options are used to establish colors for the section cut line, active section plane, and inactive section plane. See the Sections topic for more information.

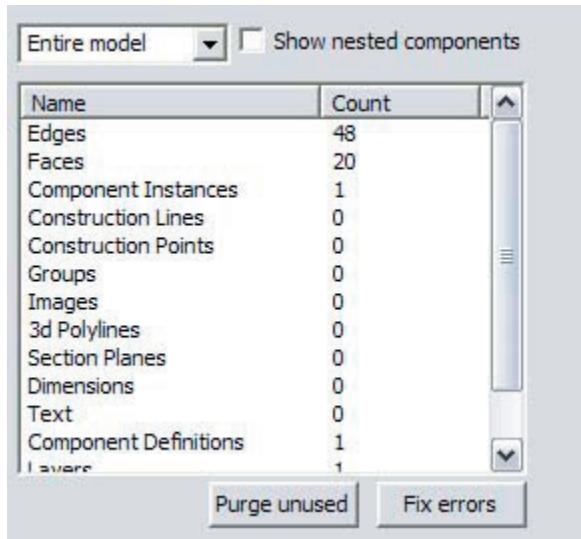
**Section Cut Line:** The Section Cut Line option is used to define the color for the active section plane's section slice line.

**Active Section Plane:** The Active Section Plane option is used to define the color for the currently active (highlighted) Section Plane entity.

**Inactive Section Plane:** The Inactive Section Plane option is used to define the color for the currently inactive (unhighlighted) Section Plane entities in your model.

## Statistics

The Statistics panel is used to troubleshoot performance issues in SketchUp. This panel displays information about the type and number of drawing elements in your model and allows you to perform verifications on your model.



The screenshot shows the SketchUp Statistics panel. At the top left is a dropdown menu set to "Entire model". To its right is a checkbox labeled "Show nested components" with a checked state. Below this is a table with two columns: "Name" and "Count". The table lists various drawing elements and their counts. At the bottom of the panel are two buttons: "Purge unused" and "Fix errors".

Name	Count
Edges	48
Faces	20
Component Instances	1
Construction Lines	0
Construction Points	0
Groups	0
Images	0
3d Polylines	0
Section Planes	0
Dimensions	0
Text	0
Component Definitions	1
Layers	1

### Entire Model and Components

The Entire Model and Components options in the drop-down list box allow you to toggle between displaying statistics for your entire model and displaying statistics related only to components. The Components option is useful for generating a tally of the number of certain components used in your model (such as the number of a particular part in an assembly).

### Show Nested Components

The Show nested components option is used to include individual elements within placed components in the statistics totals.

### Purge Unused

The Purge Unused option is used to remove any unused components, materials, image objects, layers, and other extraneous information from your file.

## Fix Problems

The Fix Problems option is used to find minor issues with your SketchUp model due to the infinite flexibility that SketchUp provides when designing in 3D. Click the Fix Problems button to scan your model, and report and fix any problems. This option checks for several cases, including:

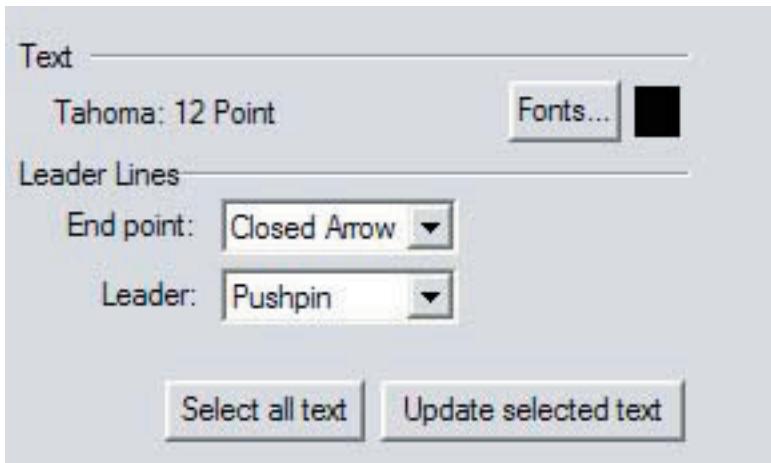
- faces bounded by properly connected edges.
- vertices of faces are on the same plane.
- edges bounding a face are in the same component.
- faces do not have zero area size.
- start and end points of an edge are not the same.



**Note -** Problem checking is also performed automatically (if turned on within the General panel of the Application Preferences dialog box) when a file is loaded, manually saved, or automatically saved (using the SketchUp auto-save feature). Refer to the Save menu item in the File menu or the auto-save feature in the General panel of the Application Preferences dialog box for further information.

## Text

The Text panel is used to change the appearance and behavior of Text entities in your model.



## Text

The Text portion of the Text panel is used to select the font used by Text entities in the drawing area.

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**Fonts...:** The Fonts... option is used to define the font to be used for all Dimension entities in SketchUp. Click on the Fonts... button to invoke the Font dialog box where you can choose the font, font style, and size for your dimensions font.

**Default Text Color:** Selects the display color for any unpainted Text entities. (Those painted with the default material.) If you paint a Text entity, it will take on the material color you specify instead.

## Leader Lines

The Leader Lines options are used to identify the type of leader line used with Text. Text might or might not have leader lines.

**End Point:** The End Point option is used to select the end point type for the leader line. Choose between None, Dot, Closed Arrow, and Open Arrow.



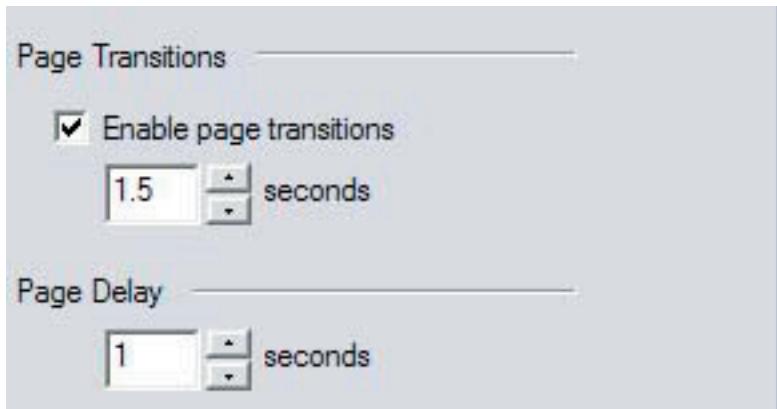
**Leader:** The Leader option is used to select between View Based and Pushpin leader types. A View Based leader will always retain its 2D screen orientation. A Pushpin leader is aligned in 3D space, and rotate with your model as you change your view.



**Update Selected Text:** The Update Selected Text option applies the current settings to any selected text objects.

## Tourguide

The Tourguide panel is used to set properties for Tourguide slideshows.



### Page Transitions

The Page Transitions options is used to determine the amount of time it takes to transition from one page in the Tourguide to another page. Click on the Enable page transitions check box to enable page transitions. Enter the number of seconds that should transpire before the next page appears.

### Page Delay

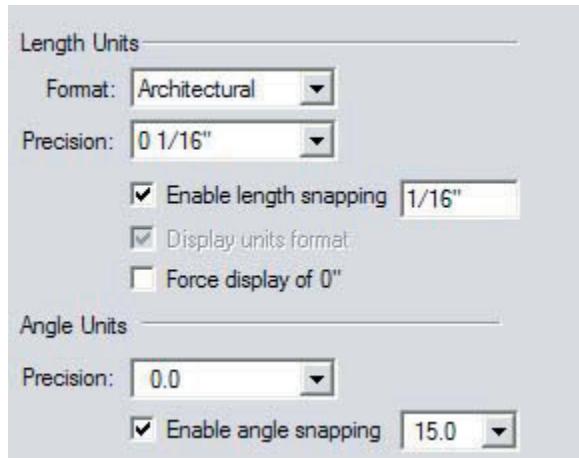
The Page Delay option is used to determine the amount of time Tourguide will spend on each page before transition to another page. Enter the number of seconds that should transpire before the transition begins to another page.

## Units

The Units panel is used to set defaults for linear and angular unit measurements.



**Note** - You can override the default units using the VCB to work in different units simultaneously.



### Length Units

The Length Units options are used to set the default units for your model.

**Format:** The Format option is used to set the type of units displayed for measurements, dimensions, and within the VCB. Architectural displays as feet and inches. Decimal displays decimal units, including inches, feet, millimeters, centimeters, and meters. Engineering displays feet and decimal units of feet, and fractional units display only fractional inches.



**Note** - Fractional units are primarily used by wood workers in the United States.

**Precision:** The Precision option is used to set the precision for displaying units. Architectural unit precision can be set from 1" to 1/64". Decimal unit precision can be set in numbers of decimal places beyond zero for any of these units. SketchUp users working in Metric should use the decimal unit setting. Engineering unit precision can be set in decimal places beyond zero.

**Enable length snapping:** The Enable length snapping option is used to snap lines and other entities to the specified sized increments. Length snaps can be overridden by entering exact values into the VCB, or by using any of SketchUp's intervening functions. Type the snap length in the text field to have SketchUp snap to specific increments when drawing.



**Note** - SketchUp does not have a grid snap option similar to other CAD applications.

**Display units format:** The Display units format is used to suppress the display of unit markers such as the double quotes ("") for inches.

**Force Display of 0":** The Force Display of 0" is used to display 0" in architectural units when there are no inches in the measurement. Ordinarily, inches would be suppressed in this case. For example, a measurement of three feet in length would read as 3'0" with this setting enabled.

## Angle Units

The Angle Units options are used to define how units are displayed for angled measurements

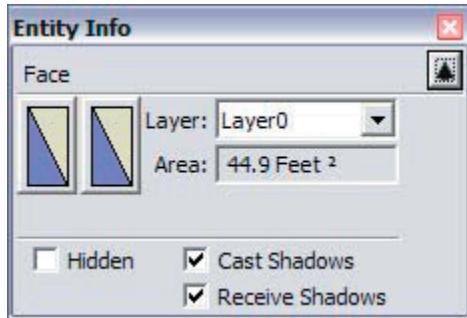
**Precision:** The Precision option is used to set the use of decimal degree units to either a level of precision of 0 (no decimal places) or 0.0 (one decimal place) for greater accuracy.

**Enable angle snapping:** The Enable angle snapping option is used to enable snapping to the specified increment while using the Protractor or Rotate Tool. Move your cursor inside the protractor to snap to the specified increment while using these tools. You will see tick marks corresponding to your snap angle on the Protractor.

**Snap Angle:** The Snap Angle option is used to choose the snap increment for angular measurements and rotations.

# Entity Info Dialog Box

The Entity Info dialog box displays information about selected entities, and allows you to view and change their properties. The Entity Info dialog box is context-sensitive, and will contain different options as you select different entities in your model. Activate the Entity Info dialog box from either the Window menu or through a context menu.



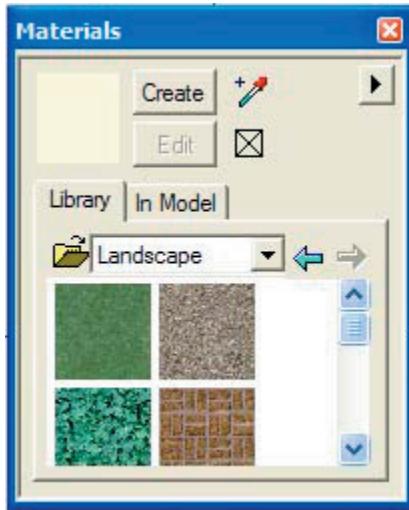
Entity Info Properties for each type of entity are described in detail within each entity in this section.

## Details Arrow

Click on the arrow in the upper-right portion of the Entity Info dialog box to show or hide additional Entity Info details.

# Material Browser

SketchUp contains a library of predefined materials that you can apply to faces in your model. The Material Browser is used to organize materials and colors into libraries and to select and to apply materials to your model.



## Applying Materials and Colors

There are multiple methods to apply a material to entities in your model. To apply materials using the Paint Bucket Tool:

1. Select the Paint Bucket Tool. The cursor will change to a paint bucket. The Materials Browser will open.
2. Click on the Library tab.
3. Locate and click on a material library within the drop-down list.
4. Click on the material or color you want to use.
5. Drag the paint bucket cursor over the selected entities.
6. Click on the selected entities to apply the color or material.



**Note -** Ensure that the Shaded with Textures display style is selected to see the materials that you have applied to entities.

## Materials Browser Options

### Active Material Swatch

The Active Material Swatch contains the material that is currently active and can be painted on a model. Click on this swatch to automatically activate the Paint Bucket Tool to paint the material.

### Controls for Manipulating Materials

The controls for manipulating materials are to the right of the Active Material Swatch. These controls allow you to create new materials, edit materials, and sample materials.

**Create Button:** The Create button is used to mix new materials using the Material Editor.

**Edit Button:** The Edit button is used to edit existing materials using the Material Editor.

**Sample Material Button:** The Sample Material button, appearing as an eye dropper, is used to sample and repaint materials appearing in your model. See the Paint Bucket Tool for additional information.

**Library Tab:** SketchUp contains a library of predefined materials and colors. These materials are grouped into categories in the Library tab below the Active Material Swatch. Select the category of material you want to use from drop-down list of material libraries. Or, use the forward and backward arrows next to the pop-up list to quickly step forward and backward through your previously selected material groups.

**In Model Tab:** The Material Browser contains a special material library called In Model containing all materials currently used in your model. The contents of this library are generated automatically as you add materials to your SketchUp model.

Select the In Model tab from the Materials Browser to view all of the materials in your model.

## Deleting Materials

Materials added to your model are stored within the model's .skp file. A material with only color information is very small, but materials with textures can get fairly large, depending on the file size of your texture. Therefore, it is suggested you delete unused materials from the In Model tab to minimize the size of your model file. To delete a material:

1. Click on the In Model tab in the Materials Browser. Thumbnails of all materials that have been painted within your model are displayed. Materials that are currently used in your model have a small triangle in the bottom right corner of their thumbnail.
2. Right-click on the material you want to delete to display the material's context menu.
3. Select Delete from the context menu. A dialog box will appear if you are deleting a material that is currently used in your model.
4. (optional) Click Yes to delete the material and replace it with the default material.

Alternative:

1. Click on the details menu (the arrow in the upper-right hand corner of the In Model panel).
2. Select Purge Unused to purge all unused materials from the In Model tab

## Calculating Material Surface Area

SketchUp allows you to calculate a total surface area covered by a particular material. This feature is useful in determining the actual amount of material costs for a portion of a model. To calculate a surface area covered by a specific material in your model:

1. Select the In Model tab of the Materials Browser.
2. Right click on the material whose usage you want to calculate. The area used by that material is displayed in the units designated in the Units panel of the Model Info dialog box.

## Selecting All Entities With a Specific Material

This ability to select all entities with a specific material is useful if you want to apply a new material to all items containing that material. To select all entities with a specific material:

1. Select the In Model tab of the Materials Browser.
2. Right-click on the material whose corresponding painted entities you want to select and select Select. All of the entities that have the identified material are selected in the drawing area.
3. (optional) Locate a new material to apply to these entities.
4. (optional) Paint the new material on the selected entities.

## Adding Materials to a Library

To add a material to the current library:

1. Select the In Model tab of the Materials Browser.
2. Right-click on the material you want to add and select Add to Library. The material is added to the currently displayed library in the Library Tab.



**Note -** You will be asked if you would like to save the currently displayed library when you switch to another materials library in the Materials Browser. Click the Yes button to save modifications to the currently displayed materials library.

## Updating Library Copy

Sketchup allows you to easily modify materials in the library and update the library copy with the modified materials. To update a material in the library:

1. Select the In Model tab of the Materials Palette.
2. Double-click on a material that is currently in one of the materials libraries. Or, right-click on the material and select the Edit menu item. The Material Editor is displayed.
3. Modify the material using the Material Editor. See the Material Editor for further information. Any changes you make to the material are reflected throughout your model.
4. Close the Material Editor.

5. Right click on the modified material in the In Model tab and select Update Library Copy to update the corresponding material in the materials library.



**Note** - You will be asked if you would like to save the currently displayed library when you switch to another materials library in the Materials Browser. Click the Yes button to save modifications to the currently displayed materials library.

## Details Menu

Click on the right arrow in the upper right-hand corner of the Material Browser to invoke details pop-up menu. This menu allows you perform additional material-related functions. The following detail menu items might appear in either or both the In Model or materials library details menus.

### Display Sizes (Small Images, Medium Images, Large Images, Library Default)

The thumbnail images of the materials can appear in different sizes. Click on the Details menu and select Small, Medium, or Large to resize the thumbnail images. Library Default sets the thumbnail images to the size saved within the materials library.

### Insert Material

The Insert Material menu item is used to insert the currently active material (in the material swatch) to the currently displayed material library.

### Purge Unused

Allows you to purge unused materials from your In Model pallet. Objects painted with the removed material revert to the default material and the selected material will be purged from the model.

### Clear Library...

To clear all materials out of a library, click on the Details menu and select Clear Library....

### Open Library...

To open a library on the file system, click on the Details menu and select Open Library...

### Merge Library...

To merge in all materials from another library, click on the Details menu and select Merge Library...

## Sort by Name

Sketchup sorts colors in a library by hue. To sort colors in a library by name, click on the Details menu and select Sort by Name...

## Saving Libraries (Save, Save As..., Save as Library)

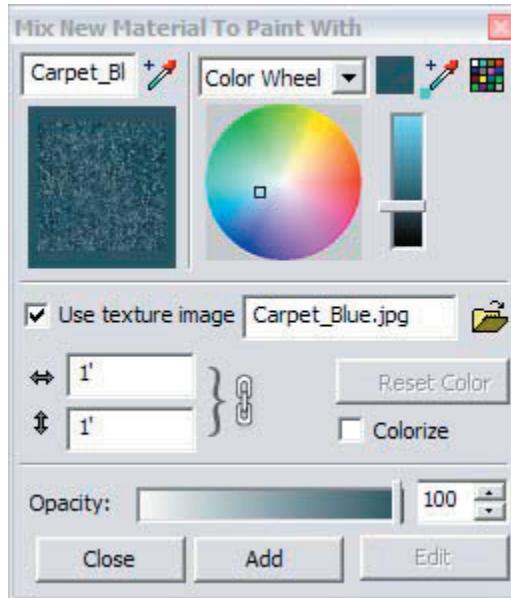
The Save and Save As menu items allow you to update and save out any changes made to your library. The Save as Library menu item allows you to create a new library from the materials currently in the In Model tab.

# Material Editor

The Material Editor is used to create or edit materials. Activate the Materials Editor either by clicking on the Create or Edit menu items and buttons in the Material Browser, or by selecting Material Editor from the Window menu.



**Tip** - Double-clicking on a material thumbnail also opens the Materials Editor.



## Editing Materials

Changes made to a material will automatically apply to the entities in the model painted with that material. This behavior allows you to interactively experiment with color variations in your model. To edit a material:

1. Select the thumbnail of the material to edit from the In Model tab of the Materials Browser.
2. Click the Edit button to open the Material Editor. The Material Editor is displayed with the material swatch.
3. Use the color pickers to change the color of the material. See Color Pickers for further information.
4. Modify the name, texture image, width, and height as desired. Refer to the following Materials Editor Options section for further information.

5. Click the Close button at the bottom of the Material Editor when you are done editing a material.



**Note -** Select the *Edit > Undo* menu item to revert back to a previous material if you make a mistake while you are editing a material.

## Material Editor Options

The top-third of the Material Editor is used to apply a new color to the material. The middle third of the Material Editor is used to apply a new texture to the material. The final third of the Material Editor is used to control the transparency of the material.

### Material Name

The Material Name is the name of the material currently being edited or created. This field appears in the upper left-hand corner of the Material Editor.

### Material

The Material swatch contains a sample of the material. This field appears below the Material Name.

### Sample Material

The Sample Material button, appearing as an eye dropper, is used to sample and repaint materials appearing in your model. See the Paint Bucket Tool for additional information.

### Color Pickers

The Color Pickers drop-down list is used to select between the RGB, (Red-Green-Blue) HSL, (Hue-Saturation-Lightness) HSB, and Color Wheel color pickers. The Color Pickers list is to the right of the Material swatch. See Color Pickers for more information.

### Undo Color Changes

The Undo Color Changes button is used to return the material color to the one used in the beginning of the edit session. This button is to the right of the Color Pickers drop-down list.

### Match Color of Object in Model

The Match Color of Object in Model button, appearing as an eye dropper with a square at its tip, is used to sample and apply a material's color from an entity in the drawing to the currently edited material. Click on a painted entity in your model with on the Match Color of Object in model cursor to sample and apply that color to the currently edited material.

## Match Color From Library Sample

The Match Color From Library Sample is used to select and apply a color from another material in the material library to the currently edited material.

## Use Texture Image

The Use texture image checkbox is used to toggle the use of a texture in the material. Checking the Use texture image box displays a file open dialog for you to choose a texture image such as a .jpg, .gif or other Raster File Format. Unchecking the box specifies that you no longer want to use a texture and erases your texture setting. Note that re-checking the use texture checkbox will not restore a previous image setting, but instead will prompt you to select a new image file.

## Texture File

The Texture File field is used to specify the texture image file. You can drag image files into the box or click on the file open icon to the right to find and open a image file.

## Reset Color

The Reset Color button is used to reset the color of the material to the original color in the image file.

## Colorize

The Colorize check box is used to lock all colors throughout the image to the same hue. This option is useful for files that are not displaying proper colors.

## Dimensions

The Dimensions options are used to specify the dimensions of the texture image as it appears in your SketchUp material. These options do not affect the original image file itself, but scale the texture. The horizontal and vertical arrows are a button that allows you to reset to the previous width and height settings.

## Lock/Unlock Aspect Ratio

The Lock/Unlock Aspect Ratio is used to lock the current aspect such a change to the height is automatically reflected by that change to the width (to maintain aspect ratio). The horizontal and vertical arrows are a button that allows you to reset to the previous width/height settings.

## Opacity

The opacity slider is used to adjust the amount of transparency in the material being edited. Move the slider to the left to make the material more transparent. Move the slider right to make the material more opaque. Alternatively, type a number from 0 to 100 (0 being fully transparent) in the opacity field.

### Add

This button will add the material appearing in the material swatch to the In Model section of the Material Browser.

### Edit

The edit button sets the editor to edit materials interactively. This button is only available when you are in the create mode and you have just selected a material that is already in the model.

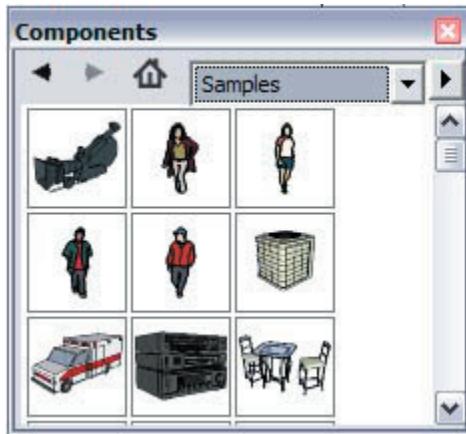
## Bringing in an Image File as a Texture Material

To specify an image file to use as a texture material:

1. Click the open button to the right of the image file text box. The Choose Image dialog box appears.
2. Locate the image on your file system.
3. Click the Open button.

# Component Browser

The Component Browser is used to instance Component entities from component definitions, including those components that you create and a variety of pre-built Components that you can use in your models. SketchUp adds the components definitions for the components you create to the In Model library, which is one of the libraries in the drop-down list of component libraries in SketchUp. Activate the Component Browser from the Window menu.



**Note** - See *component definitions and instances* for further information.

## Finding a Component

There are several ways to find components within the component library. These include the backward and forward buttons, the home button, and the component library list.

### Backward and Forward

Click on the forward and back arrows to navigate forward and back through the previously browsed component libraries.

### Home

Click on the icon shaped like a house to navigate to the In Model component library. This library contains all components within the current model.

### Component Library List

Select one of the component libraries from the component library list box to navigate to the library. The component library list represents a series of directories in the file system where components are stored.

## Inserting a Component

Components can be inserted from any component library into your model within the drawing area. To insert a component in your drawing area:

1. Click on the component in the Component Browser.
2. Move the mouse to the location in the drawing area where you want to place the component. The cursor changes to the Move Tool.



**Note** - The move tool will usually be anchored to the origin of the component's axes (also called the insert point). To see the origin of the component, select Window > Model Info > Components > Show component axes.



**Tip** - The component axes is the default insertion point (the point where the cursor grabs the component) for a component the first time you place the component in the model from the Component Browser. The insertion point will change to the point where the Move Tool selects a component if you place a component and then reposition the component using the Move Tool.

3. Click the mouse button again to release the component.



**Note** - You can also insert components from an external SketchUp (.skp) file or by dragging and dropping a file from the Explorer into your model. See the Component entity for further information.

## Component Browser Context Commands

A number of useful Component Browser-related commands are accessible using context menus. Right-click on any component's icon or name to access the Component Browser context menu. The following options might appear:

### Properties

The Properties menu item is used to display the Properties dialog box. This dialog box contains the location for the component file, when the file was last opened, the name of the component, and the number of component instances in the model.

### Delete

The Delete menu item is used to delete a single unused component from the "In Model" component library.

### Reload

The Reload menu item is used to reload a component from its original file in the event that the component's file has changed since being used in your model.

### Save As...

The Save As... menu item is used to save the selected component to a separate SketchUp file, with a new name and location. You can use this menu item to save components that have been modified or created during a drawing session.

### Reset Insertion Point

The Reset Insertion Point menu item is used to reset any changes made to a component definition's insertion point.

### Select Instances

The Select Instance menu item is used to select all of the instances of a specific component in your model.

### Replace Selected

The Replace Selected menu item is used to replace the currently selected components in the drawing area with the currently selected component in the Component Browser.

## Details Menu

Click on the right arrow next to the component library list to open the details pop-up menu. This menu allows you perform additional component-related functions. The following detail menu items might appear in either or both the In Model or component library details menus.

### Icon View

The Icon View menu item is used to display components in the Component Browser as icons.

### List View

The List View menu item is used to display the components in the Component Browser by name and dimension.

### Expand

The Expand menu item is used to show components nested within other components.

### Purge Unused

The Purge Unused menu item is used to purge all unused components from the "In Model" component library.

## Add Folder

The Add Folder menu item is used to add a new component library to the list of component libraries in the Component Browser. You will be prompted to locate a directory within your file system to create a new component library directory. You can copy new component files (.skp or .skb files) into this directory once this directory is created.

## Remove Folder(s)

The Remove Folder(s) menu item is used to remove a component library from the list of component libraries in the Component Browser. A dialog box with a list of all component libraries will be listed. Click on a library to highlight the library and then click on the Remove button.

## Refresh

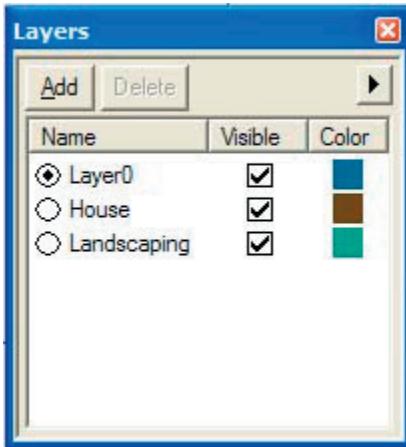
The Refresh menu item is used to update the components in the Component Browser with any updated versions on the file system.

## Get More...

SketchUp contains a sampler of a few of the over 3000 pre-created components available on the SketchUp web site. The Get More menu item is used to retrieve additional components from the SketchUp web site.

# Layer Manager

The Layer Manager is used to apply and manage layers in your model. Activate the Layers Manager from the [Window menu](#).



The Layers Manager displays all the layers and their associated visibility in the model. Every model has one layer, called Layer0, which will be visible the first time you activate the Layers dialog box.



**Note** - You cannot delete Layer0. If you use any other layer as a drawing layer, all the entities that were created on that layer will become invisible when you hide that layer. This is not the case when hiding layer 0. Any entities that are on layer 0, but inside a group or component that is on another layer, will remain visible when you hide layer 0.



**Caution** - Remember, layers in SketchUp do not work exactly like layers in 2 dimensional programs. Namely, entities drawn with one layer appear in all layers in SketchUp.

## Moving Geometry Between Layers

To move geometry from one layer to another:

1. Select the Select Tool. The cursor changes to an arrow.
2. Select one or more entities. The selected entities are highlighted in yellow.
3. Activate the context menu for the selected entities.
4. Select the Entity Info menu item. The Entity Info dialog box appears.
5. Select the layer for the entities from the Layers drop-down list.

## Layer Manager Options

### Add

The Add option is used to create additional layers. Click Add to add a layer, enter the layer name (or press Enter or Return to accept the default name). Each new layer has a different color to help you distinguish layers.

### Delete

The Delete option is used to delete a layer. Select a layer and click Delete to delete a layer. You will be prompted to move entities on the deleted layer to another layer (the default or current layer) if there are entities unique to that layer.

### Name

The Name column lists all the layers, by name, in the drawing. The current layer has a check next to its name. Check the box next to a layer name to make it the current layer. Click on the name of a layer to edit the name of the layer.

Click on the Name column header to sort the layer alphabetically. Click on the Name column again will reverse the order. You can select a multiple layers by dragging. You can also use a Ctrl-click to selectively pick layers or Shift-click to select a series of contiguous layers.

### Visible

Toggle the visibility of a layer by clicking on the Visible checkbox. Click on the Visible column header to sort layers by visibility. Click again on the Visible column header to reverse the order. A hidden layer becomes automatically visible when you select the layer.

### Color

The color column displays a color associated with each layer. You can change the color of a layer by clicking on it and selecting a new color. Click on the Color column header to sort layers by color. Click again on the Color column heading to reverse the order.

## Details Menu

Click on the right arrow next to the Delete button to open the details pop-up menu. This menu allows you perform additional Layer Manager-related functions.

### Select All

The Select All option is used to select all layers in the list of layers.

## Purge

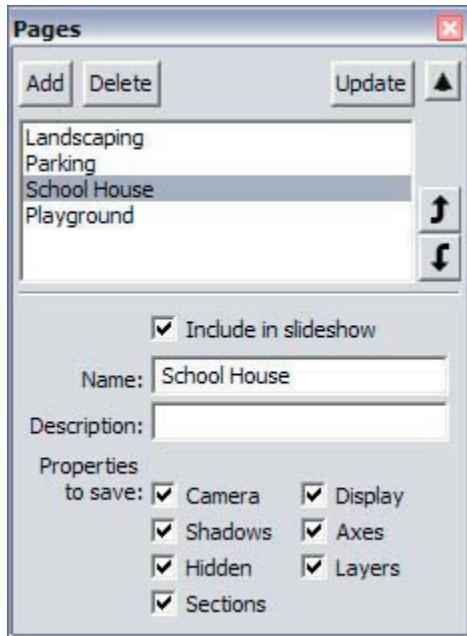
The Purge option is used to delete all unused layers (layers without any geometry).

## Color by Layer

The Color by Layer option is used to apply materials to geometry on a per layer basis using the color associated with the layer. Ensure shaded or shaded with Textures display is turned on to view materials.

## Page Manager

The Page Manager box is used to control the various features of SketchUp Pages. Activate the Pages dialog box from the Window menu or from the page tabs.



The Page Manager contains a list of all of the pages for the model. Pages in this list are displayed in the order in which they will be displayed when running a TourGuide Tour.



**Caution -** Each page has a series of properties that you can store with that page (Properties to save). Additionally, when you update a page, you can choose those properties, of the properties that are stored with the page, to update (the dialog box that appears when the Update button is clicked). If you choose to update a property that you are not storing with a page (using Properties to save), the property will not be updated. Therefore, it is important to ensure you have all of the properties you want to store with a page checked prior to updating a page.

## Pages Manager Options

### Add

The Add button is used to add a new page to the current file. To add a page:

1. Click on the Add button. A new page, with a default page name, is added to the list of pages and a page tab appears above the drawing area. See Page Manager Tabs later in this topic for further information.
2. (optional) Rename the default page name to something that represents the view of the model that will appear in the page. For example, if your page contains a top-down view of the model, call the page "Top."
3. (optional) Provide a description of the page.
4. (optional) Unselect any properties you do not want to save with the page.

### Delete

The Delete button is used to delete a page from the current file. Select the page in the list of pages and click the delete button to delete the page.

### Update

The Update button is used to update a page if you have made changes to the page. To update a page:

1. Select the page you want to update.
2. Click on the properties, within Properties to save, to store with the page.
3. Click on the Update button. A Properties to Update dialog box appears.
4. Click on the properties you want to update. Remember, you can only update those properties that you have previously selected to be stored with the page.
5. Click on the Update button.

### Include in Slideshow

The Include in Slideshow option is used to indicate whether a page should be used in the TourGuide Tour. Select a page and check the Include in Slideshow option to include the page in a slide show. Disable this option for pages you want to work on but do not want in a slideshow.

## Name

The Name field is used to name the currently active page.

## Description

The Description field is used to provide a short description or note for the currently active page.

## Properties to Save

The Properties to Save is used to control the different properties that can be stored with each page. Properties that are not checked in the Properties to save section cannot be updated with the Update button.

**Camera:** The Camera option is used to store the point of view, including the zoom distance and field of view, with the page.

**Shadows:** The Shadows option is used to store all shadow-related information, including type, time, date, and so on, with the page.

**Hidden:** The Hidden option is used to store the hidden or unhidden status of entities with the page.

**Sections:** The Sections option is used to store the active section plane with a page.



**Tip -** Use different section cuts in successive pages to create exciting slide shows with different cross sections of your model.

**Display:** The Display option is used to store display settings, such as edge rendering, with the page.

**Axes:** The Axes option is used to store display and position of SketchUp's drawing axes with the page.

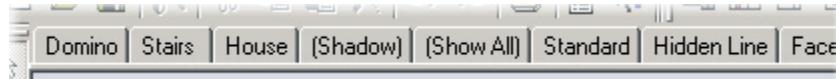
**Layers:** The Layers option is used to store the layers with the page.

## Details Arrow

Click on the arrow in the upper-right portion of the Page Manager dialog box to show or hide additional Entity Info details.

## Page Manager Tabs

Page Tabs are to switch between the pages you have created. Page Tabs are created for each page as each page is created using the Page Manager.



**Note -** Page Tabs can be hidden using Page Tabs menu item in the View Menu.

SketchUp

## Page Tab Context Menu Items

Page Tabs contain their own context menus. Right click on a page tab to access its context menu.

### Move Left/Move Right

The Move Left and Move Right menu items are used to reposition a page in the series of pages.

### Add

The Add menu item is used to add a new page to the current file.

### Update

The Update menu item is used to update a page if you have made changes to the page.

### Delete

The Delete menu item is used to delete the currently selected page.

### Slideshow

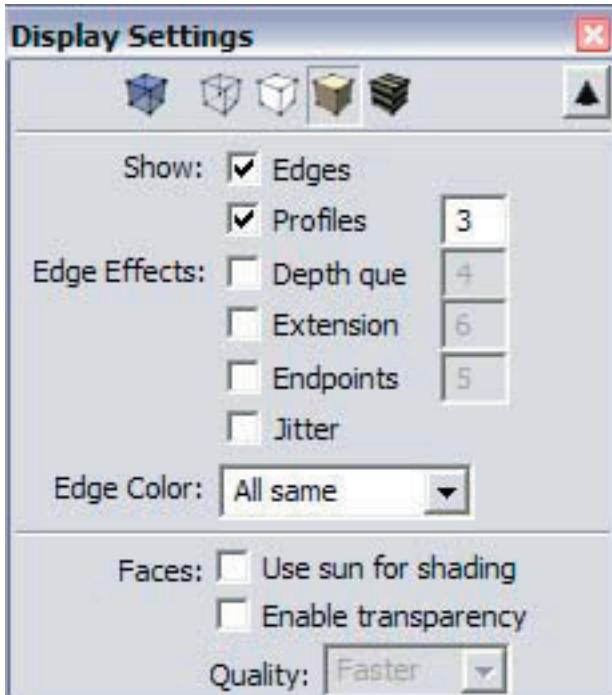
The Slideshow menu item starts a TourGuide slideshow. See the Pages and TourGuide topic for further information.

### Page Manager

The Page Manager menu item invokes the Page Manager dialog box.

## Display Settings Dialog Box

The Display Settings dialog box controls how your SketchUp model is rendered. This dialog box contains items pertaining to edge and face style and color. Activate the Display Settings dialog box from the Window menu.

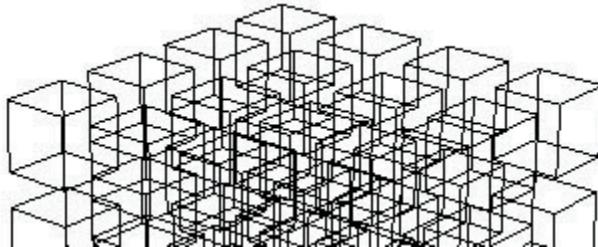


## Face Rendering

SketchUp contains a variety of face rendering styles allowing you to manipulate the amount of material and textures displayed on the screen.

### Wireframe

The Wireframe face rendering style is used to display the model as a collection of lines. Faces are not displayed in Wireframe mode.



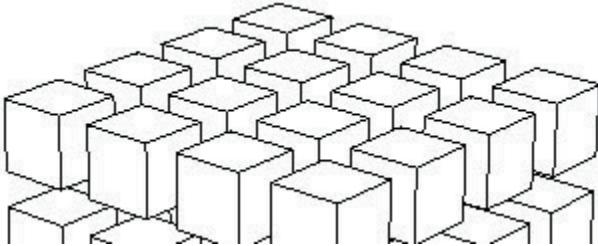
**Note** - You cannot use face modification tools, such as the Push/Pull Tool, on a wireframe rendered model.

### Hidden Line

The Hidden Line face rendering style is used to display faces in the model without any shading or textures.

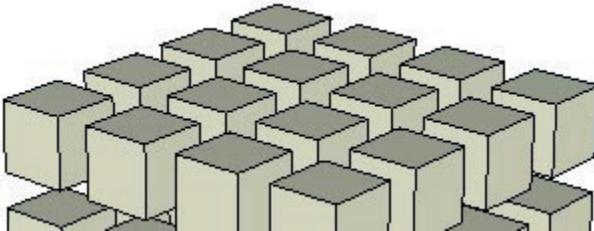


**Tip** - Use this face rendering style when creating black and white printouts which you might want to modify further with traditional media, or which you might use as an underlay for hand drawings.



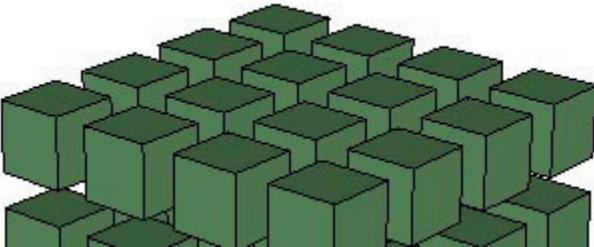
## Monochrome

Monochrome face rendering style is used to display the model as an assembly of edges and faces, just as with the Hidden Line face rendering style. However, Monochrome mode provides default shading and is useful for shadow studies whereby you turn the face front and face back to white and then display shadows.



## Shaded

The Shaded face rendering style is used to display faces in the model as shaded to reflect a light source. Any color that had been applied to a face will be displayed. Remember that both sides of faces can have different colors. The default colors (as specified in the Color panel of the Model Info dialog box) are displayed when no color has been applied to a face.

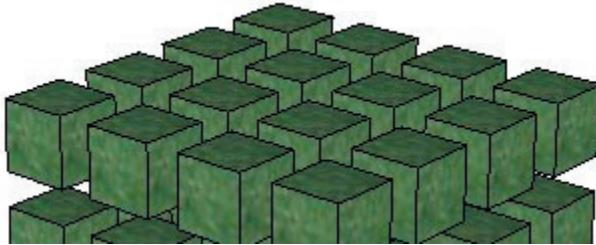


## Shaded with Textures

The Shaded with Textures face rendering style is used to display the faces in model with the texture images that have been applied to a face.



**Tip** - *Textures can slow down SketchUp's performance. Use other face rendering styles, such as the Shaded face renderings style, when performance slows dramatically. Switch back to the Shaded with Textures face rendering mode to create your final output.*

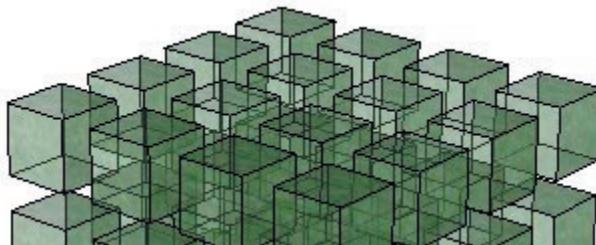


## X-Ray Mode

The X-Ray Mode is used to display all faces in SketchUp with a global transparency. This option allows you to see through the model's faces and edit edges behind faces and can be combined with any of the above face rendering styles.

You can easily visualize, select, and snap to points and edges that would otherwise be hidden behind faces when modeling with X-ray mode. Remember, however, that it is not possible to select and infer faces that would otherwise be hidden.

Faces can not cast Shadows when using X-ray mode. Shadow display will default to ground plane shadows only when X-ray mode is enabled. X-ray mode is different than material transparency.



## Edge Rendering

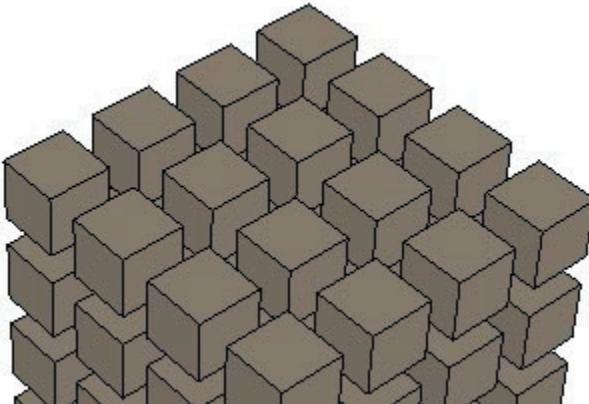
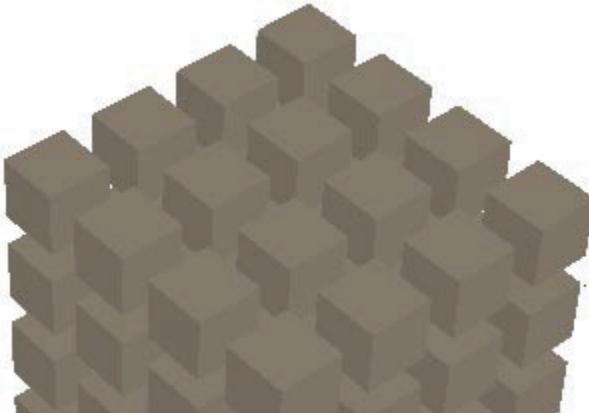
SketchUp contains a variety of edge rendering styles allowing you to manipulate the edge appearance on the screen.



**Tip** - Models with perfectly straight digitally drawn lines are often perceived to be in a finished state. Edge Rendering Styles are useful in conveying that a model is still in conceptual state and, therefore, is open for feedback.

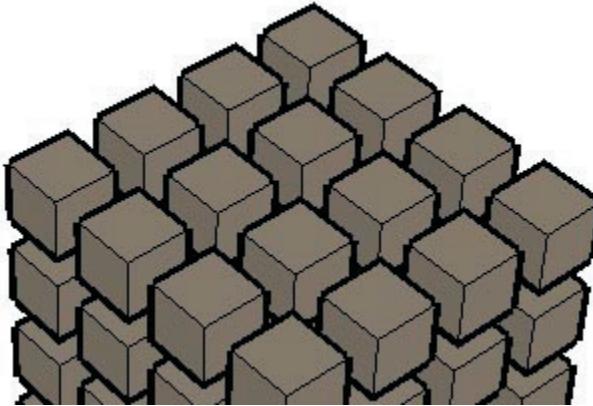
### Displaying Edges

The Edges check box is used to toggle the display of edges in your model. The following images shows a series of cubes without and then with edges.



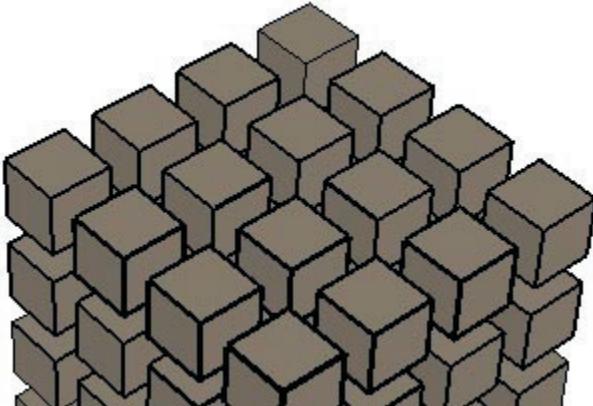
## Profile Lines

The Profile Lines edge rendering style is used to emphasize the outlines or profiles of major shapes in your model. This style is particularly useful in ensuring the 3D nature of geometry is emphasized and borrows from a proven traditional media drawing technique. Enter a thickness, in pixels, for the profile lines in the pixels field. The following image shows a series of cubes with edges and profile edges.



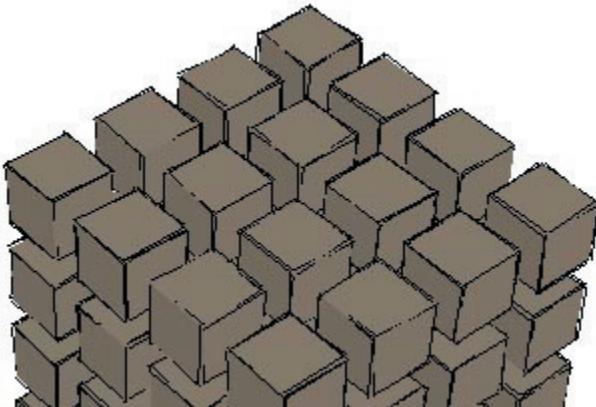
## Depth Cue

The Depth Cue edge rendering style is used to emphasize the lines of geometry in the foreground over the lines of geometry in the background. The following image shows a series of cubes with edges and depth cue edges on. Notice that the foreground edges on each cube get progressively thinner from front to back and top to bottom. Enter a thickness, in pixels, for the depth cue lines in the pixels field. This thickness determines the thickness of the forward-most lines (such as for the bottom middle cube in the following image) of your model.



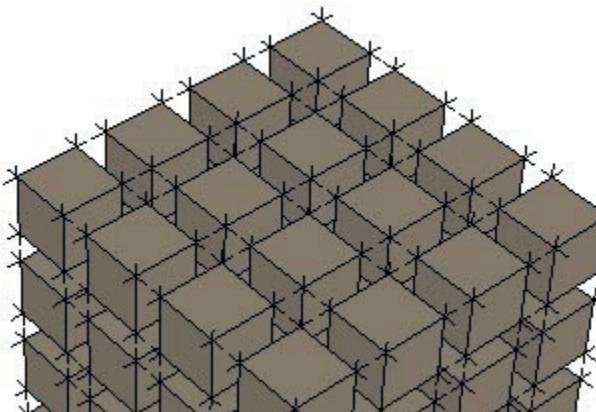
## Jitter Lines

The Jitter Lines edge rendering style is used to render each line multiple times at a slight offset, giving your model hand-drawn sketched appearance. This edge rendering style does not affect inference behavior. The following image shows a series of cubes with edges and jitter edges.



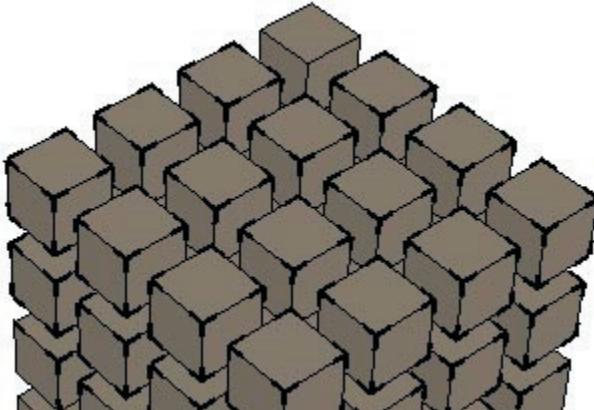
## Extension Lines

The Extension Lines edge rendering style is used to extend each line slightly past its endpoint, giving your model hand-drawn sketched appearance. This edge rendering style does not affect inference behavior. Enter a length, in pixels, for the extension lines in the pixels field. The following image shows a series of cubes with edges and extension edges.



## Endpoints

The Endpoints edge rendering style places additional line thickness at the endpoints of lines. Enter a length, in pixels, for the length of the emphasized endpoints in the pixels field. The following image shows a series of cubes with edges and endpoints edges.



**Tip** - Experiment with combinations of edge renderings to see how they appeal to you.

## Edge Colors

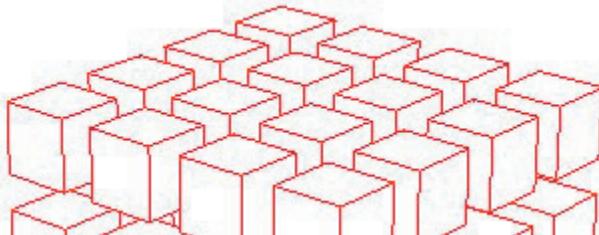
SketchUp also allows you to manipulate the edge color on the screen.



**Note** - Edge Colors are only available when using Shaded and Shaded with Textures Face Rendering Styles. Inference alignments to edges are not available when edges are hidden.

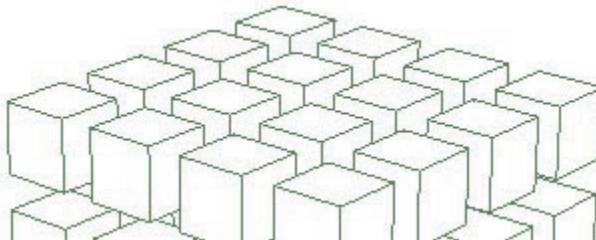
## All Same

The All Same option is used to display all edges using the Edges color as defined in the Colors panel of the Model Info dialog box. This option does not actually change any edge color assignments you may have made, but preserves them if you choose to view them again.



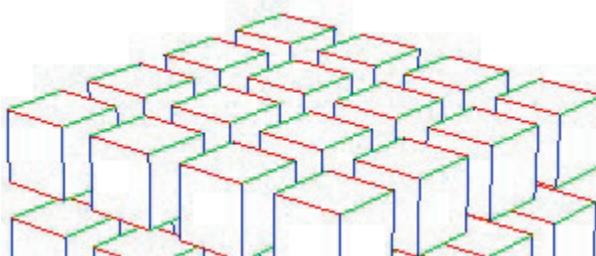
## By Material

The By Material option is used to display edges using an assigned material color. The following image contains edges painted with a grey material.



## By Axis

The By Axis option is used to display edges in colors corresponding to the color of the Drawing Axes to which they are parallel. This option is helpful in determining when edges are not aligned to an axis. The following image has lines colored red, green, and blue corresponding to the axes to which they are aligned.



## Additional Face Rendering Options

The Display Settings dialog box contains additional rendering options affecting the rendering of your model.

### Use sun for shading

The Use Sun for Shading option is used to shade the faces of your model based on the position of the sun. SketchUp uses standard shading that follows the camera when Use Sun for Shading is not enabled.

### Enable Transparency

The Enable Transparency option is used to enable or disable global material transparency.

## Quality

The Quality options are used to select the quality of transparency display between faster, medium, and nicer. Each option differs in its speed and quality of transparency sorting. The Faster display sacrifices sorting accuracy to provide a faster rendering update rate. Conversely, the Nicer display performs additional calculations to correctly sort transparent surfaces.

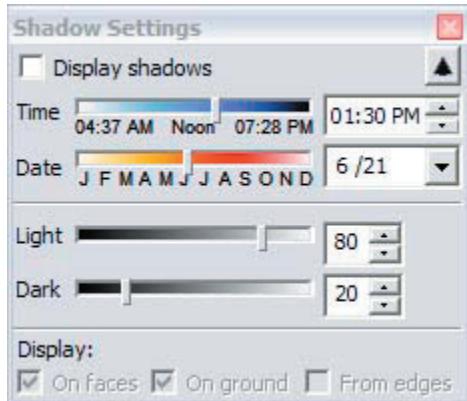
Because SketchUp's transparency system is designed for real-time feedback and display, it may sometimes display transparent faces in an unrealistic way: Faces may appear as if they were in front of other surfaces when they are really behind, and vice-versa.

## Details Arrow

Click on the arrow in the upper-right portion of the Display Settings dialog box to show or hide additional Entity Info details.

# Shadow Settings Dialog Box

The Shadow Settings dialog box is used to control SketchUp's Shadows feature, including display, time and date, and site location and orientation. You can also use the Shadows Toolbar control to control shadows. Activate the Shadow Settings dialog box from the Window menu.



**Note** - Ensure you have set the proper location for your model set through the Location panel of the Model Info dialog box for accurate shadow casting.



**Note** - Daylight savings time is not factored into shadow calculations.

## Shadow Settings Options

SketchUp contains a variety Shadow setting options allowing you to manipulate the use of shadows within your model.

### Display Shadows

The Display Shadows option is used to toggle between displaying and not displaying shadows within your model.

### On Faces

The Faces option is used to enable casting of Face shadows. This feature makes intensive use of your 3D graphics hardware and can cause performance degradation.

### On Ground

The Ground option is used to enable the casting of shadows on the ground plane (the red/green plane).

### From Edges

The From Edges option is used to enable the casting of shadows from edges that are not associated with a face.

### Time

The Time option is used to adjust the time of day used by SketchUp to determine the location of the sun for shadow casting. The slider adjusts the time from sunrise to sunset, with 12:00 noon in the middle of the slider. Type a time into the time text field to set a precise time.

### Date

The Date option is used to adjust the day of the year used by SketchUp to determine the location of the sun for shadow casting. The slider adjusts the date from January 1st to December 31st. Type a date into the date text field to set a precise day. You can specify the date in numerical form (11/8).

### Light

The Light option is used to control the intensity of the light in the model. (it lightens or darkens light). This option effectively lightens and darkens illuminated surfaces.

### Dark

The Dark option is used to control the intensity of light in the model. This option (it lightens or darkens shadows). This option effectively lightens and darkens the areas under shade and shadows.

### Details Arrow

Click on the arrow in the upper-right portion of the Page Manager dialog box to show or hide additional Entity Info details.

## Soften Edges Dialog Box

SketchUp's edges can be softened and smoothed to achieve a real-world appearance. Edges are automatically hidden when softened. Softened edges can also be smoothed rendering the adjoining faces with a smooth tonal gradient. The following series of images contains a column (lower part of each image) that is first softened (middle column) and then smoothed (right-most column).

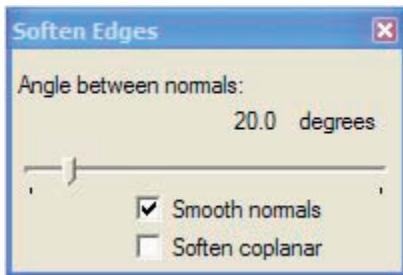


**Note** - Softened edges are hidden automatically (but still exist in your model). Edges that are currently not visible will be displayed, along with any explicitly hidden geometry, when you enable the Hidden Geometry menu item in the View menu.



**Tip** - Triple-click on geometry to ensure you have hidden and unhidden geometry selected before performing a soften or smooth operation.

The Soften Edges dialog box is used to automatically apply or remove softness and smoothing effects to the edges in your model. Activate the Soften Edges dialog box from the Window menu.



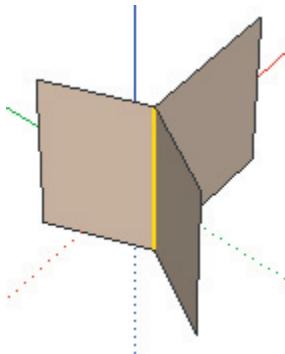
**Tip** - There are several components to achieving a desired result using soften and smoothing effects. Namely, detailed geometry require additional facets (such as a column having many small sides) to achieve a clean soften or smooth effect. However, additional detail can affect the computer's performance. Conversely, adding detail to portions of a model that will be seen at a distance is likely a waste of the computer's resources. A good strategy for creating useful models is to try to use the least amount of geometry in conjunction with soften or smoothing effect to achieve the desired result (whether it be a detailed close-up or a panoramic).

## Softener Edges Options

SketchUp contains a few Soften Edges options allowing you to manipulate the use of softening within your model.



**Note** - Edges that are shared by two or more faces cannot be softened. The following image shows an edge shared by three faces.



## Angle Between Normals

The Angle Between Normals option is used to set the maximum size of all angles that will be smoothed or softened. The higher the setting, the more angles you are likely to smooth or soften with a smooth or soften command. Consequently, the higher the setting the more performance might degrade.

## Smooth Normals

The Smooth Normals option is used to smooth any edges, essentially spreading color and texture over edges, to make the edges appear smooth.

## Soften Coplanar

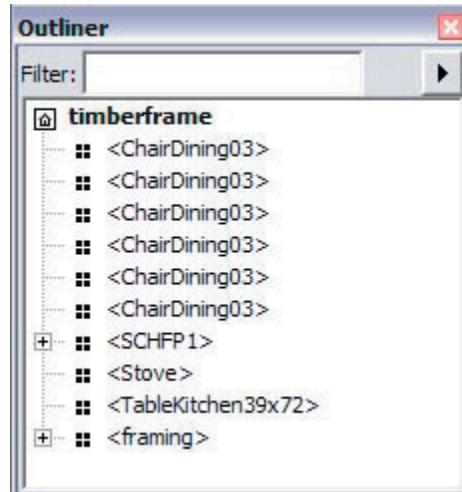
The Soften Coplaner option is used to soften edges, essentially deleting edges, between coplanar surfaces.



**Tip** - Create a three dimensional multi-sided polygon (with more than 5 sides) and experiment with these options to become familiar with softening and smoothing models. For example, create a 6-sided polygon, expand the polygon using the Push/Pull Tool, select the entire 3D polygon, click on Smooth Normals, and move the Angle Between Normals to 61 degrees. Notice that each edge of the polygon looks smooth because the angle between each normal is 60 degrees and, therefore, can be smoothed. Now move the slider to 59 degrees. Notice that the sides no longer look smoothed because the Angle Between Normals only operates on angles below or equal to 59 degrees. Experiment further by turning on the Soften Coplaner option to see edges appear and disappear.

# Component Outliner

The Component Outliner is used to view your Group and Component hierarchies as a hierarchical tree. This feature is great for navigating through large models, restructuring the model hierarchy, locating instances of a particular component, or renaming groups and components. Activate the Component Outliner dialog box from the Window menu.



## Traverse Component and Group Hierarchies

The outliner contains a hierarchy view that reflects your hierarchy of components and groups in your model. To traverse a component or group hierarchy using the hierarchical view:

1. Display the Component Outliner using the Window > Component Outliner menu item. The Component Outliner dialog box opens with a hierarchical view of your model. The Group and Component entities at the root (top-most) level of the hierarchy are displayed. An icon with a plus sign will appear next to any components or groups if additional levels in the hierarchy exists (additional groups or components within the top-level groups or components).
2. Click on the plus (+) icon to list groups or components within other groups or components. Or, optionally, click on the Component Outliner's Details Menu and select the Expand All menu item to expand the hierarchy so all levels are displayed.
3. Click on any group or component name in the hierarchical view to select the group or component in your model.
4. (optional) Double-click on the group or component name in the hierarchical view to edit the group or component in your model.

## Identifying Entries in the Component Outliner Hierarchy

The Component Outliner uses a combination of icons and text to identify groups and component status in a hierarchy.

- Component - Four black squares
- Group - One solid square
- Locked Component - Four grey squares with small lock in lower right corner
- Locked Group - One grey square with small lock in lower right corner
- Open Component - Four hollow squares
- Open Group - One hollow square
- Hidden Component or Group - Name of component or group is in italics

## Moving Items Within the Component Outliner

Use the Component Outliner to reorganize groups and components in the hierarchy. For example, you can move a group, buried deep within the hierarchy, to the top of the hierarchy for easy editing. To move items within the outline:

1. Display the Component Outliner using the Window > Component Outliner menu item. The Component Outliner dialog box opens with a hierarchical view of your model. The Group and Component entities at the root (top-most) level of the hierarchy are displayed. An icon with a plus sign will appear next to any components or groups if additional levels in the hierarchy exists (additional groups or components within the top-level groups or components).
2. Click on the plus (+) icon to list groups or components within other groups or components. Or, optionally, click on the Details Menu and select the Expand All menu item to expand the hierarchy so all levels are displayed.
3. Click and hold on the group or component you want to move in the hierarchy.
4. Move the group or component to a new position in the hierarchy.

## Filtering Groups and Components in the Hierarchical View

The Component Outliner contains a Filter field for displaying only those groups or components containing a specific filter string. This feature helps you to locate only like components or groups. To filter items within the Component Outliner:

1. Display the Component Outliner using the Window > Component Outliner menu item. The Component Outliner dialog box opens with a hierarchical view of your model. The Group and Component entities at the root (top-most) level of the hierarchy are displayed. An icon with a plus sign will appear next to any components or groups if additional levels in the hierarchy exists (additional groups or components within the top-level groups or components).

2. Type a string in the Filter field.
3. Click on the plus (+) icon to list groups or components within other groups or components. Or, optionally, click on the Details Menu and select the Expand All menu item to expand the hierarchy so all levels are displayed. Items containing the filter string will be listed, in red, in the hierarchical view.

## Name Groups and Components Within the Component Outliner

The Component Outliner displays groups by their group names and components by a combination of the definition and specific instance name.

### Naming a Component Definition

As mentioned in the previous paragraph, components are identified in the outliner by a combination of their instance and definition names (the definition name is enclosed in less-than and greater-than symbols). For example, Jim's Office <Cubicle Style 1>. Where you may have several Cubicle Style 1 components in your model, but only one instance represents Jim's Office.

The component definition name generally refers to type of component, such as a certain type of cubicle (Cubicle Style 1) or certain type of video camera (VideoX 8mm camera). This name can either be established when the component is first created, or you can accept the default (Component#X, X being some number) and rename the component definition using the outliner. To rename the component definition:

1. Context-click on the component in the Component Outliner. The context menu for the Component entity is displayed. Notice that this is the same context menu as the context menu displayed when you context-click on an entity in the drawing area (except this context menu has a Rename menu item).
2. Click on Entity Info menu item. The Entity Info dialog box is displayed.
3. Click on the Definition tab to display information about this component's definition.
4. Type a new name in the definition's name field.
5. Click outside of the Entity Info dialog box to save your change.

### Naming a Component Instance

A component instance name is useful when you want to differentiate different component instances from each other in the Component Outliner (the default name for all component instances is "Component"). For example, you might want to have different instance names if several chairs are all from the same definition in your model. To rename the component instance:

1. Context-click on the component in the Component Outliner. The context menu for the Component entity is displayed. Notice that this is the same context menu as the context menu displayed when you context-click on an entity in the drawing area (except this context menu has a Rename menu item).
2. Select the Rename menu item

3. Type the new component instance name in the Component Outliner.
4. Press Enter to save your change.



**Note** - You can also use the Name field at the top of a component's Entity Info dialog box to rename component instances.



**Tip** - Use instance names to differentiate different instances of the same definition in your model. This tip is particularly useful if you want to easily locate a specific component instance in the outliner.

## Naming a Group

You can also name your groups for easy identification in the Component Outliner. To rename a group:

1. Context-click on the group in the Component Outliner. The context menu for the Component entity is displayed. Notice that this is the same context menu as the context menu displayed when you context-click on an entity in the drawing area (except this context menu has a Rename menu item).
2. Select the Rename menu item
3. Type the new component instance name in the Component Outliner.
4. Press Enter to save your change.

## Details Menu

Click on the right arrow next to the Filter field to open the details pop-up menu. This menu allows you perform additional Component Outliner-related functions.

### Expand All

The Expand All menu item is used to expand (open all branches) the entire hierarchical view in the Component Outliner.

### Collapse All

The Collapse All menu item is used to collapse (close all branches) the entire hierarchical view in the Component Outliner.

### Sort by Name

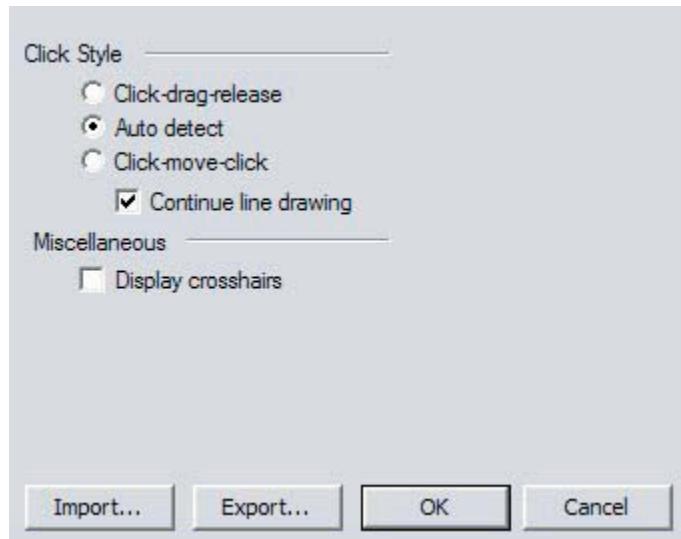
The Sort by name menu item allows you to toggle the sorting of the groups and components alphabetically.

# Application Preferences Dialog Box

SketchUp's application preferences are used to set various global behaviors for SketchUp.  
Activate the Preferences from the Window menu.

## Drawing

The Drawing preferences panel is used to define global mouse (or other *input device*) behavior.



### Click Style

The Click Style options are used to define how your input device reacts to clicks.

**Click-drag-release:** The Click-drag-release option forces the Line Tool to draw by click and holding the mouse button to define the start point of the line, dragging the mouse to extend the line, and releasing the mouse to establish the end point of the line.

**Auto Detect:** The Auto Detect option allows you to use either Click-drag-release and Click-move-click as necessary.

**Click-move-click:** The Click-move-click option forces the Line Tool to draw by clicking and releasing the mouse button to define the start point of the line, moving the mouse to extend the line, and clicking again to establish the end point of the line.

**Continue Line Drawing:** The Continue line drawing option informs the Line Tool that you will usually want to draw a new line at the end of each line. When enabled, the Line tool automatically treats an end point as the start point of a new line, saving you one extra click required to establish a new start point for the second line.

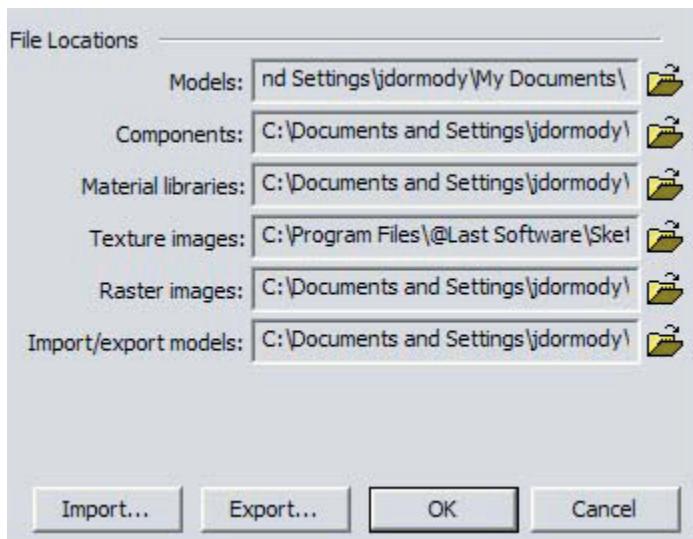
## Miscellaneous

The Miscellaneous options are used to define other global mouse behaviors.

**Display crosshairs:** The Display crosshairs option is used to display a set of colored crosshairs (corresponding to axes colors) when drawing.

## Files

The Files preferences panel is used to define various locations for files used by SketchUp (such as the default location for Components and Materials).



## Models

The Models field is used to define the default location for all of your models. SketchUp uses this location as the starting point for all open and save file operations.

## Components

The Components field is used to define the default location for all of your Components. SketchUp uses this location as the starting point for when you open and save new components.

## Material Libraries

The Materials Libraries field is used to define the default location for all of your Material Libraries. SketchUp uses this location as the starting point for all Components within the Materials Browser.

## Texture Images

The Texture Images field is used to define the default location for all images that can be used as a texture. SketchUp uses this location as the starting point for all File > Insert > Image as Texture operations.

## Raster Images

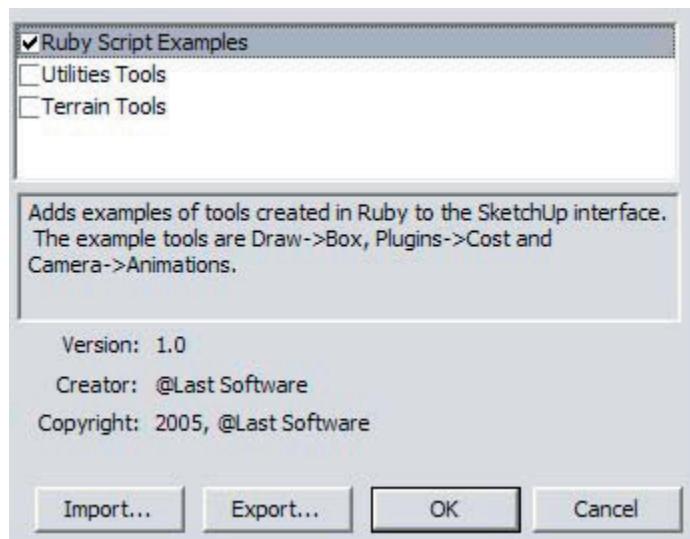
The Raster Images field is used to define the default location for all images that can be used as a texture. SketchUp uses this location as the starting point for all File > Insert > Image... operations.

## Import/Export Models

The Import/Export Models field is used to define the default location for all models that are imported or exported. SketchUp uses this location as the starting point for all File > Insert > DWG/DXF and File > Export > 3D Model... operations.

## Extensions

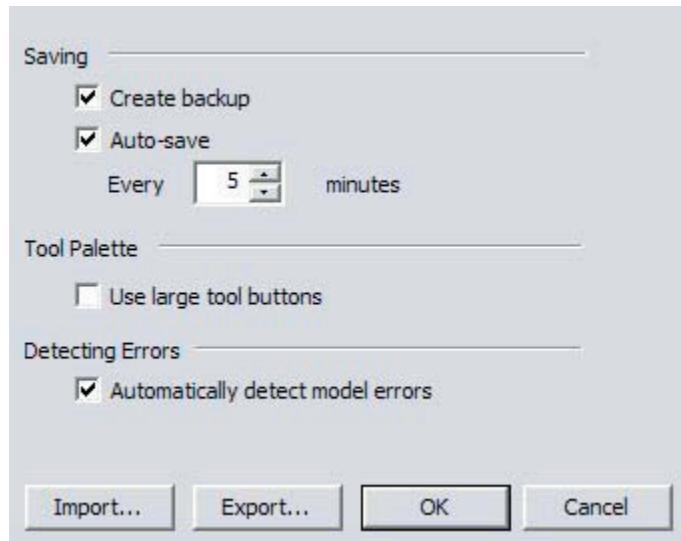
The Extensions preferences panel is used to add or remove SketchUp *extensions* from the user interface. An extension can be an add-on tool or feature created by @Last Software that is not necessarily a part of core SketchUp functionality.



Click in the check box next to the extension that you want to enable in SketchUp. The extension will be available (menu items and optional toolbar) the next time you run SketchUp.

## General

The General preferences panel is used to define global save and user interface settings.



### Create Backup

The Create Backup option is used to automatically create a backup file whenever you save a drawing. The backup file is the previously saved version of the file. It will be saved to the same folder as the drawing file. For example, if your drawing is hotel.skp, the backup will be called hotel(skb).

### Auto-Save

The Auto-Save option is used to automatically save changes to your drawing into a temporary file at specific time interval. This temporary file is available on the file system should SketchUp abnormally exit. Type the auto-save interval, in minutes, into the minutes field.

The temporary file's name will be a combination of the word "AutoSave" with the original filename, such as AutoSave\_hotel.skp. This file is saved in the directory where the original .skp file exists.



**Caution -** @Last Software strongly recommends you keep the Auto-Save option checked to ensure that your valuable work is retained.

SketchUp will check your model for unrecoverable errors during the auto-save, preventing the overwriting of a good auto-save file. Sketchup will display a dialog box in the rare case that unrecoverable errors are found in your model. This dialog box contain the option to quit SketchUp and send a report. @Last Software recommends you click on the quit SketchUp and send report button to terminate SketchUp and preserve your previously valid auto-save file. This report contains valuable information regarding the unrecoverable errors.

SketchUp

## Tool Palette

SketchUp for has two differently sized Tool Bars. The large tool buttons are easier for some users to see and use (they provide larger areas to click with your mouse) and are easier to use with a pen and tablet. Smaller buttons save screen space, allowing a larger Drawing Area.

## Check Model For Problems

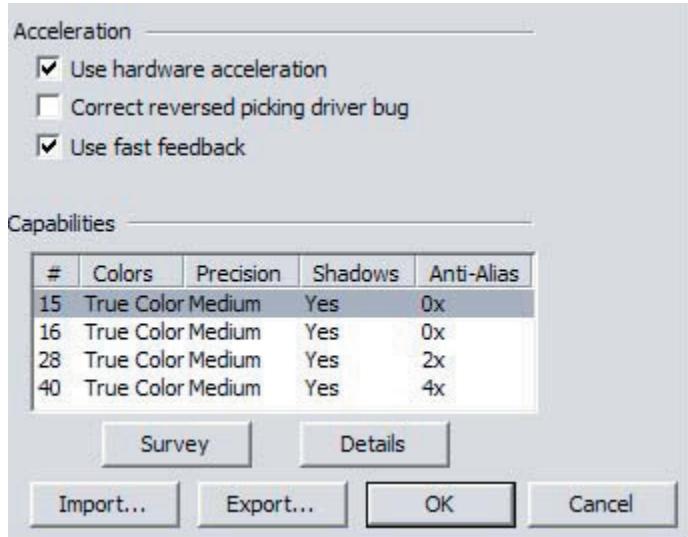
The Check Models for Problems options are used to enable problem checking for your SketchUp models.

**Automatically check models for problems:** The Automatically check model for problems check box toggles the checking for minor problems with your model. These problems can occur given the infinite flexibility that SketchUp provides when designing in 3D. Your model is assessed for problems when it is loaded or saved. @Last Software strongly recommends checking this box to ensure that problems are corrected proactively, allowing your model to perform optimally. You must manually check for problems using the Fix Problems button in the Statistics panel of the Model Info dialog box if you leave this check box unchecked.

**Automatically fix problems when they are found:** Check the Automatically fix problems when they are found check box to have SketchUp fix problems found in your model automatically (without having to display a dialog box). Leave this check box unchecked if you want SketchUp to display a dialog box (with the Always fix my models, Fix it now, and Fix it later options) when a problem is found.

## OpenGL

The OpenGL preferences panel is used to define how the OpenGL standard should be applied within SketchUp.



**Note** - You might notice intermittent streaks or small spots of light within face shadows. A minimal amount of these artifacts is to be expected on most systems, and can vary greatly with the quality of your OpenGL driver.



**Caution** - Certain OpenGL drivers do not fully support casting shadows on faces.



**Caution** - SketchUp's Shadow feature can have a severe performance impact on slower computers or those using software rendering.

## Use Hardware Acceleration

Enable the Use Hardware acceleration option to use the hardware acceleration features of a video card. Unfortunately, only a small percentage of 3D drivers in the consumer video card market are 100% OpenGL compatible and can use this feature (though many cards claim to be 100% OpenGL compatible). Most 3D drivers are designed for games, and are rarely tested using other 3D programs. Consequently, numerous incompatibility problems can occur requiring a fix from the video card manufacturer. Disable this option if you are having problems with the 3D rendering of your models or if your video card is not 100% OpenGL compatible and does not support hardware acceleration.



**Note -** *Hardware Acceleration might only be available on your system for certain resolutions and color depths. Check the system settings for your graphics card to see if it supports hardware acceleration (using the Control Panel on Microsoft Windows).*

@Last software cannot control the quality of the OpenGL driver on your computer system. Video card device drivers are proprietary and are maintained solely by the manufacturer of the video card in your system. Therefore, @Last Software cannot guarantee that SketchUp will work with hardware acceleration on your system. Refer to the OpenGL topic for further information.

## Correct Reversed Picking Driver Bug

The Use face highlight workaround is used to correct an issue whereby some video card device drivers cause SketchUp to select the reverse of faces with the Select Tool. This setting provides a solution to this problem.

## Use Fast Feedback

The Use Fact Feedback option is used to increase performance when editing large models. SketchUp runs a test to assess whether your video card can support this option on startup. SketchUp will activate (check) this option if fast feedback is supported. You can also attempt to use this feature by checking the Use Fast Feedback checkbox manually, however you will be warned by SketchUp that enabling this option could cause problems on your system if fast feedback is not supported.

@Last Software strongly recommends that you set your driver's anti-aliasing setting to application controlled (which is normally the default setting for most graphic card drivers) to ensure that fast feedback functions properly. It is possible that you will see rendering anomalies, such as blurring of the model, when using fast feedback under a non-application controlled setting.

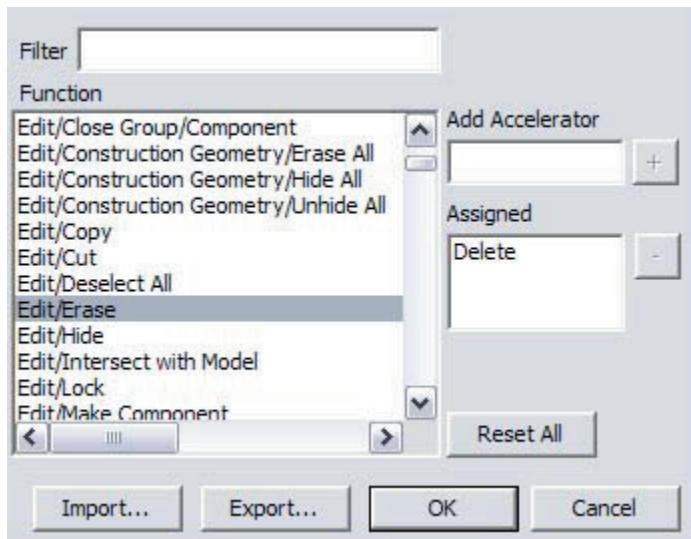
## Capabilities

The capabilities table lists one or more different graphic card settings combinations that might work well within SketchUp. These combinations were selected by SketchUp, on execution, from a large list of settings combinations supported by your graphics card. SketchUp defaults to the most basic of these combinations to ensure your models will be rendered properly. Choose a different combination if you want to alter your graphics capabilities within SketchUp.

**Anti-Alias:** SketchUp generally defaults to a combination with a zero anti-alias value. Higher anti-alias values, such as 2x or 4x, can create smoother looking lines in SketchUp. However, you might experience some flickering and reduced rendering performance if a higher anti-alias value is selected.

## Keyboard Shortcuts

The Shortcuts preferences panel is used to define keyboard shortcuts for most SketchUp commands.



## Assigning Keyboard Shortcuts

Keyboard shortcuts can dramatically speed up your drawing by allowing you to change tools while keeping the mouse cursor near the drawing area. To assign a keyboard shortcut:

1. Select the command to which you will assign a keyboard shortcut (within the Function list). Existing keyboard shortcuts for this command will display in the Assigned list.



**Note -** The list of commands available in the Shortcuts panel is dynamically generated and includes all menu items and active context menu items. Some context menu items become active only after performing another task. For example, select a face to activate the Reverse Faces command and then open Preferences > Shortcuts to set a keyboard shortcut for Reverse Faces.

2. Type the keyboard shortcut key sequence In the Add Accelerator field. Shortcut keys can consist of any keyboard key, except number keys, with a few exceptions. Any key that can be used as a shortcut can also be assigned a modifier key like Shift, Control, or Alt. SketchUp will indicate when a key or key combination cannot be used or is already assigned.

3. Click the plus (+) button.

You can have multiple keyboard shortcut combinations for each command. SketchUp will prompt you before allowing you to assign a keyboard shortcut that is already assigned to another command. Some keys are reserved for use by Microsoft Windows and can not be assigned as a keyboard shortcut.



**Note -** You can save your shortcuts to a .dat file for copying to other computers. See the Files panel in this topic for more information.

## The VCB and Keyboard Shortcuts

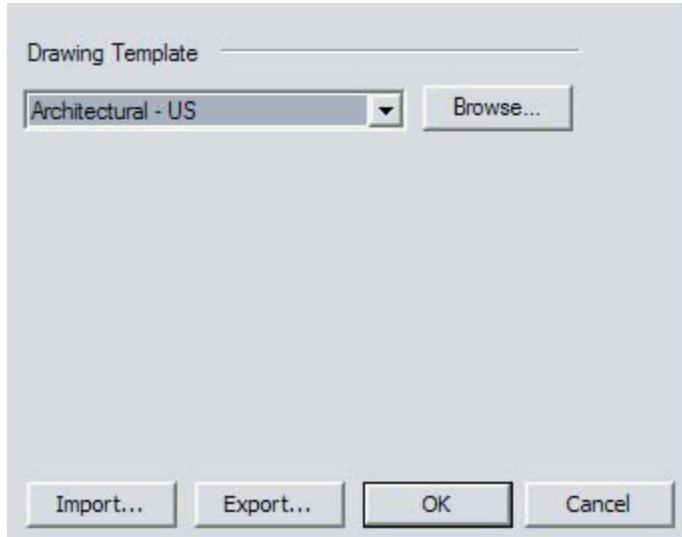
Temporary focus is given to the VCB when you type a number while in the drawing area. Therefore, numbers cannot be used as shortcut keys. The letters S, R, X, and the symbols / and \* can be used within keyboard shortcuts and as you enter values in the VCB. Prepend any of these letters with a number in the VCB to ensure that a command with a corresponding keyboard shortcut letter is not invoked. For example, 7s would be the correct entry to redefine a segmentation in the VCB. However, s7 will activate whatever command is invoked with the letter S as a keyboard shortcut.



**Tip -** Do not use the spacebar or backspace keys as keyboard shortcuts to avoid conflict with values entered in the VCB.

## Template

The Template preferences panel is used to designate a SketchUp file that you would want to use as a template for all of your subsequent models.



### Creating a Template

A template file contains default settings from the Model Info dialog box and base geometry. To create a template:

1. Select File > New menu item to start a new SketchUp file.
2. Modify the Model Info settings to fit your particular needs. For example, you might want to set default units and snaps, activate a ground plane, and set a geographic location.
3. Modify the Preference settings to fit your particular needs.
4. (optionally) Draw any geometry that you want to share among all of your models. For example, you might want all of your models of houses to be created within the same starting terrain (such as when you are creating models of houses within a housing development).
5. Select File > Save to save the SketchUp file. Save this file in the Templates directory (under the SketchUp installation directory) if you want this file to appear in the drop-down list of templates within the Preferences dialog box.

### Automatically Loading a Template

You can load a template SketchUp file manually using File > Load every time you want to create a model. Or, you can identify a file as a template using the Template section of the Preferences dialog box to load the template automatically when you run SketchUp.

Select a template from the list of templates to load a template (if your SketchUp template file is in the Templates directory). Click the Browse button to locate templates that do not appear in the drop-down list.

Your template will load every time you run SketchUp.

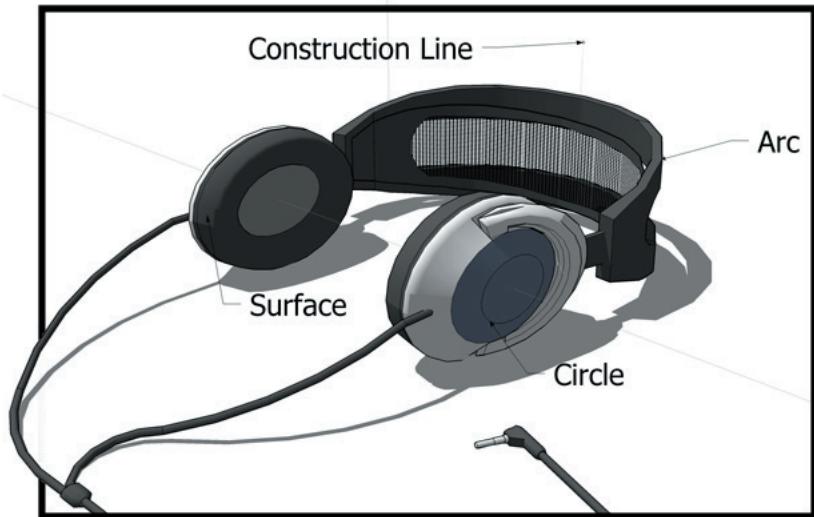
## Importing and Exporting Preferences

You can import and export your keyboard shortcuts and file preferences using the Import and Export buttons at the bottom of each preferences panel. These preferences are exported as a Preferences.dat file, by default, that can be imported into other SketchUp installations. For example, if you use SketchUp on a desktop machine at work, but want to use it at home, export the Preferences.dat file at work and import it on your home machine.

# Entities

This section of the user's guide covers the various entities used to construct a model in SketchUp. These entities include faces, arcs, dimensions, and lines.

Entities are covered in alphabetical order.



# Arc Entities

Arc entities are a combination of multiple line segments connected together to approximate the curvature of the arc. These entities act as a single line in that they can define the edge of a face and also divide a face. Additionally, selecting one segment of the arc selects the entire Arc entity. However, all inference techniques will operate on the Arc as though it were comprised of segments. For example, every point on the arc is inferred as an endpoint of the segment. Draw Arcs with the Arc Tool.



**Note -** You can explode a arc into regular edge segments using the Explode Curve context menu item (see the Arc Context Menu Items section later in this topic).

Arc entities are comprised of a length (also called the base chord), bulge, radius, and number of segments. The following image shows an Arc entity.



**Note -** Arc and Circle entities are special in that they automatically produce softened edges when extruded using the Push/Pull Tool.

## Editing an Arc Entity

You can edit the radius of an Arc entity by using the Move Tool. To edit an Arc entity:

1. Select the Move Tool. The cursor will change to a four-way arrow.
2. Move over the Arc entity to locate the midpoint of the Arc entity.
3. Click and hold on the midpoint of the Arc entity.
4. Move the cursor to adjust the bulge of the Arc entity.



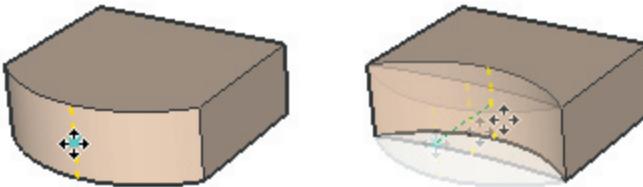
5. Click and hold on the starting or ending point of the Arc.

6. Move the mouse to adjust the radius and length of the arc entity. The radius attempts to stay proportional to the base chord length.

You can also adjust the radius and number of segments using the Arc entity's Entity Info dialog box.

## Editing an Extruded Arc

When you use the Push/Pull Tool to extrude a 2D face that includes an arc, it extrudes a special arc Surface entity which can also have its radius edited. Use the Move Tool to reposition the midpoint edge, and the arc curved face set (as well as the midpoints of the two arc entities that define it) will move accordingly.



### Arc Segmentation

Arcs with more line segments appear to have smoother curvature than arcs with fewer line segments. However, more line segments increases the size of your model and degrades performance. You can often achieve acceptable results by indicating small segmentation and using smoothing and edge softening to create the impression of smoothness.

### Arc Deformation

If an Arc is deformed in a way that destroys its radial definition, such as with a non-uniform scale operation, it will become a non-parametric Curve entity. Polyline Curves can no longer be edited as arcs.

## Arc Context Menu Items

Access context menu items by context clicking on an entity. The arc context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

## Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

## Select

The Select submenu contains several options for selecting geometry related to the entity.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected arc.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected arc.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

## Soften

The Soften menu item is used to soften a faceted surface. Softened edges are not visible unless displayed in profile.

## Divide

The Divide menu item is used to divide an entity into any number of equal segments. This menu item is not available when the arc bounds a curved surface. To divide an entity:

1. Context click on the entity to invoke the entity's context menu
2. Select Divide. A string of red dots appears along the line segment.
3. Drag the cursor back and forth along the length of the entity to increase or decrease the number of divisions. A ToolTip displays the number and length of segments that will be created if you pause briefly on the entity.



**Note -** The number of divisions is also displayed in the Value Control Box (VCB). Type in the number of desired segments followed by the Enter key to manually divide the entity.

4. Click again to divide the entity. The entity will be broken into separate segments.

## Explode Curve

The Explode Curve menu item is used to break the entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

## Convert to Polygon

The Convert to Polygon menu item is used to convert the entity into a polygon arc, which will remain editable yet generate standard facets when extruded.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Point at Center

The Point at Center menu item is used to place a point at the center of your arc or circle entity. This menu item is generated by a Ruby script.

## Arc Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note** - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Radius

This Radius field displays the radius of the selected entity. Type new values in to this field to change the radius of the entity.

## Segments

The Segments field displays the number of segments in the selected entity. Type new values in this field to change the number of segments for the entity.



**Note** - Arc segmentation can only be changed when the Arc entity is part of 2 dimensional geometry (not part of a curved surface).

## Length

The Length field displays the length of the currently selected arc.

## Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Cast Shadows

The Cast Shadows menu item allows the entity to cast a shadow.

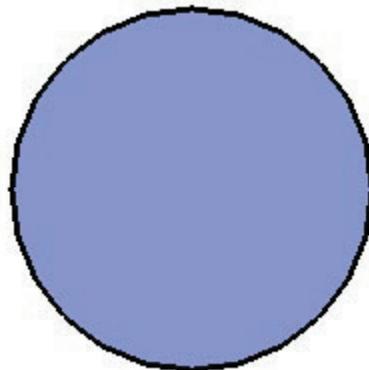
# Circle Entities

Circle entities are a combination of multiple line segments that are connected together to form a circle. These entities act as a single line in that they can define the edge of a face and also divide a face. Additionally, selecting one segment of the arc selects the entire Circle entity. However, all inference techniques will operate on the Circle as though it were comprised of segments. For example, every point on the circle is inferred as an endpoint of the segment. Draw circles with the Circle Tool.



**Note** - You can explode an Circle into regular edge segments using the Explode Curve context menu item (see the Circle Context Menu Items section later in this topic).

Circle entities are comprised of a radius and number of segments. The following image shows a Circle entity with 24 segments.



**Note** - The model in the previous image contains two entities: a Face entity (in the middle) surrounded by a Circle entity (the circular edge).

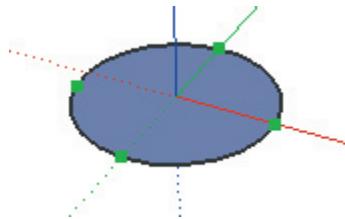


**Note** - Arc and Circle entities are special in that they automatically produce softened edges when extruded using the Push/Pull Tool.

## Editing a Circle Entity

You can edit the radius of an Circle entity, that does not yet bound a face, by using the Move Tool. To edit an Circle entity:

1. Select the Move Tool. The cursor will change to an four-way arrow.
2. Move over the Circle entity to locate one of the four cardinal points of the Circle entity.



3. Click and hold on the cardinal point of the Circle entity.
4. Move the cursor to adjust the radius of the Circle entity.

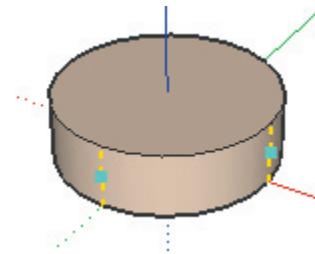
You can also adjust the radius and number of segments using the Circle entity's Entity Info dialog box.



**Tip** - *The inference engine sometimes can get distracted, preventing it from snapping to the center of a circle. You can encourage a center point inference by hovering the mouse cursor over the edges of the circle and then moving it towards the center point.*

## Editing an Extruded Circle

When you use the Push/Pull Tool to extrude a 2D face that includes a circle, it extrudes a special cylindrical Surface entity which can also have its radius edited. Use the Move Tool to reposition one of the four cardinal edges, and the cylindrical curved face set radius (as well as the radii of the two circle entities that define it) will be adjusted accordingly.





## Circle Segmentation

Circles with more line segments appear to have smoother curvature than circles with fewer line segments. However, more line segments increases the size of your model and degrades performance. You can often achieve acceptable results by indicating small segmentation and using smoothing and edge softening to create the impression of smoothness.

## Circle Deformation

If an Arc is deformed in a way that destroys its radial definition, such as with a non-uniform scale operation, it will become a non-parametric Curve entity. Polyline Curves can no longer be edited as arcs.

## Circle Context Menu Items

Access context menu items by context clicking on an entity. The circle context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Select

The Select submenu contains several options for selecting geometry related to the entity.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected face.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected face.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

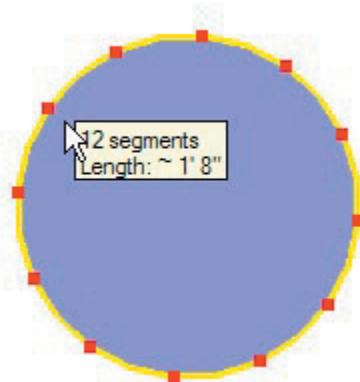
## Soften

The Soften menu item is used to soften a faceted surface. Softened edges are not visible unless displayed in profile.

## Divide

The Divide menu item is used to divide an entity into any number of equal segments. This menu item is only available for entities that are not connected to other geometry. To divide an entity:

1. Context click on the entity to invoke the entity's context menu
2. Select Divide. A string of red dots appears along the line segment.
3. Drag the cursor back and forth along the length of the entity to increase or decrease the number of divisions. A ToolTip displays the number and length of segments that will be created if you pause briefly on the entity.



**Note -** The number of divisions is also displayed in the Value Control Box (VCB). Type in the number of desired segments followed by the Enter key to manually divide the entity.

4. Click again to divide the entity. The entity will be broken into separate segments.

## Explode Curve

The Explode Curve menu item is used to break the entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

## Convert to Polygon

The Convert to Polygon menu item is used to convert the entity into a polygon arc, which will remain editable yet generate standard facets when extruded.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Point at Center

The Point at Center menu item is used to place a point at the center of your arc or circle entity. This menu item is generated by a Ruby script.

## Circle Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note** - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Radius

This Radius field displays the radius of the selected entity. Type new values in to this field to change the radius of the entity.

### Segments

The Segments field displays the number of segments in the selected entity. Type new values in this field to change the number of segments for the entity.



**Note** - Circle segmentation can only be changed when the Circle entity is part of 2 dimensional geometry (not part of a curved surface).

### Length

The Length field displays the length (circumference) of the currently selected circle.

## Hidden

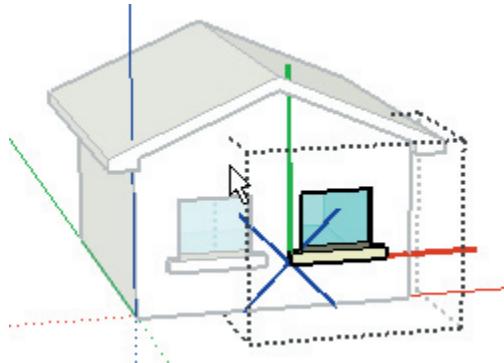
The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Cast Shadows

The Cast Shadows menu item allows the face of the entity to cast a shadow.

# Component Entities

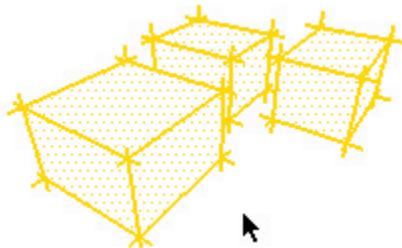
Components are entities that can hold other entities. Components are similar to Group entities, but they are commonly used to combine several entities as a single entity for the purposes of reuse in multiple models. Use the Make Component menu item from the Edit menu to create a component from the currently selected entities.



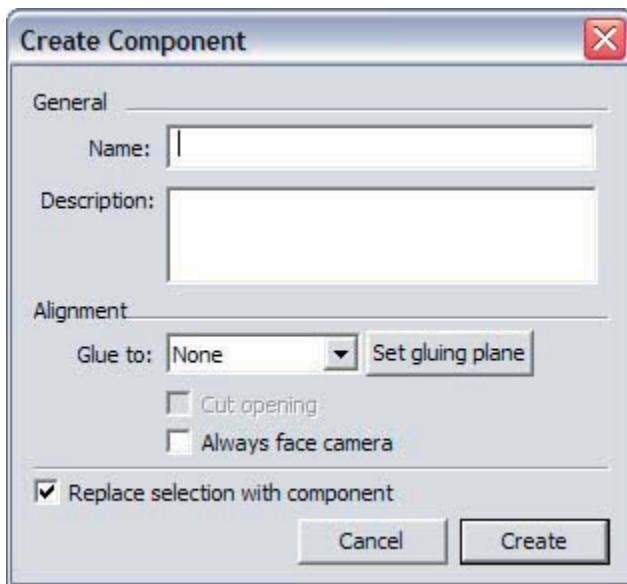
## Creating a Component

Components are useful for creating reusable models to be placed within other models. The most important issue to consider when creating components is how you want them to be placed when inserted into the model from the Component Browser. The component axes will dictate both component orientation on insert and cutting plane (for components that are to automatically cut holes in faces, such as windows). To create a component:

1. Select the Select Tool. The will change to an arrow.
2. Click and hold the mouse button a short distance away from the entities you want to select to start a selection box.
3. Drag the mouse to the opposite corner of the selection starting point.
4. Release the mouse button when all of the elements are either partially included (left-to-right selection) or fully included (right-to-left selection) in the selection box.

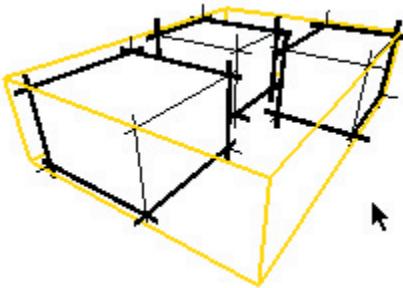


5. Select the Make Component menu item from the Edit Menu. Alternatively, context click on the currently selected entities and select Make Component from the context menu. The Create Component dialog box is displayed.



6. Fill out the fields in the dialog box. Ensure you make all of the appropriate selections and check all of the appropriate boxes before continuing. Specifically, decide whether the component should glue to faces in a specific orientation and cut openings. See The Create Component Dialog Box for further information.

7. Click the Create button. SketchUp adds the newly created component to the In Model Component Library.



**Note** - The Make Component operation disconnects any geometry that was connected to selected geometry prior to placing the selected geometry in the component. The disconnected geometry is maintained outside of the component's context.



**Note** - You can make component hierarchies by grouping other Component entities within a component. Additionally, you can mix your hierarchies by including components and groups within other components and groups.



**Tip** - Create components that can be attached or glued to a surface in context (on a surface) to ensure that the cutting plane is established correctly.



**Video** - Video tutorials on this topic are available at  
<http://www.sketchup.com/training/tutorials.php>

## The Create Component Dialog Box

The Create Component dialog box is displayed when you attempt to create a component using the Make Component menu item.

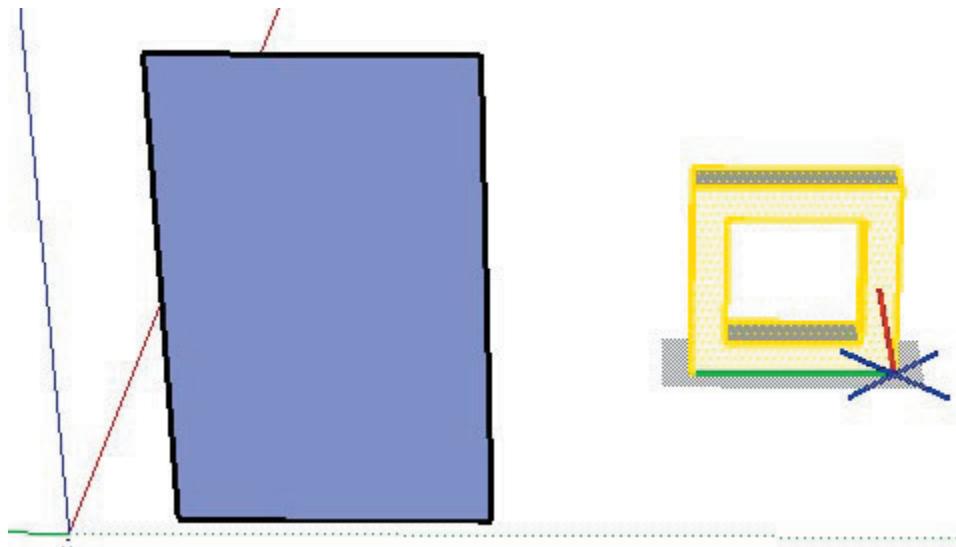
### General

**Name:** The Name field can contain the name of the component definition. All component definitions must have a name.

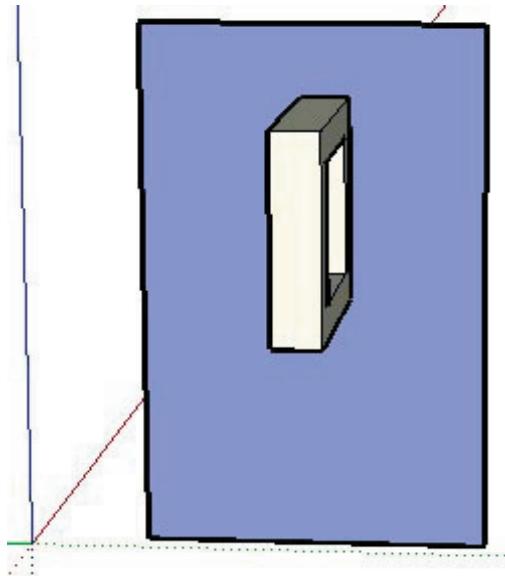
**Description:** The Description field can contain a description of the component.

## Alignment

**Glue to:** The Glue to drop-down list is used to identify the faces where your component can be placed when initially placed from the Component Browser. For example, a standard door might only be glued to faces in the horizontal (blue) plane. A grey gluing plane guide will appear when a specific gluing plane is selected. This plane represents exactly where the component will orient to a face and cut into a face. The following image shows a window component during component creation. Notice that the gluing plane (the grey plane) is parallel to the red/green plane and bottom of the window). This window component was also set to glue to vertical surfaces.



The following image shows what occurs when the previously mentioned window component is placed, from the Component Browser, against a vertical surface. Notice that the window meets the vertical face at the window's bottom because it was created with a gluing plane that is parallel to the bottom of the window.



Usually you want windows and doors to have a gluing plane that is parallel to the front or back of the window or door (not the bottom). You might need to reorient the component's axes at creation, using the Set Plane button, to properly set the gluing or cutting plane.

**Cut Opening:** The Cut Opening option allows the Component to create openings in the face onto which it is placed. For example, a door or window component might be set to cut an opening in any wall where instances of the component are placed.



**Note -** There must be edges along the cutting plane of the component to cut a hole in a face.

**Always face camera:** The Always face camera option allows the component to take on billboard behavior by drawing the component as a 2D form. This option increases performance by eliminating the need to render the component as a 3D model.



**Note -** Components with the Always face camera option enabled cannot have gluing behavior.

**Replace selected:** The Replace selection option causes the currently selected entities to be turned into a Component instance. Uncheck this option to create a component definition in the Component Browser without creating a component instance from selection set.

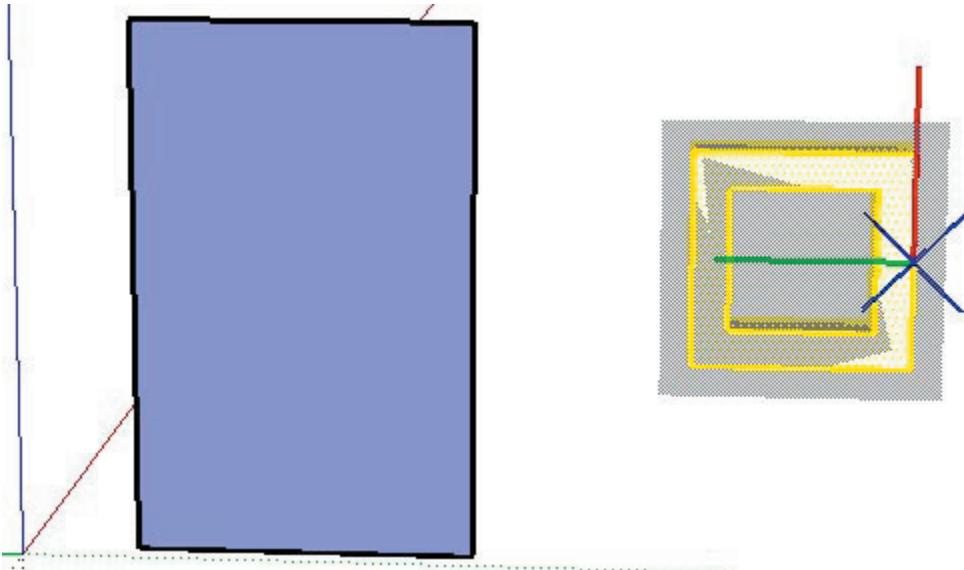
**Set Plane:** The component axis defines how the component inserts and aligns to other geometry or to the camera. The component axis also defines the cutting plane by the orientation of the red/green plane. The Set Plane option is used to specify a different origin for the Component and to modify the orientation of the component when it is placed. To set the origin and plane of a component:

1. Follow the steps 1 through 5 in the Creating a Component section of this topic.
2. Select Show Component Axes in the Components Panel of Model Info dialog box to see the existing axes on a Component before using the set plane option to move the axes.
3. Click on the Set Plane button. The cursor changes to an arrow with axes attached to it.
4. Click on a location of the component to set the origin of the component's axes. The origin and insertion point is now set to this location.



**Note** - When you change the axes you change the orientation of the component when dragged from the Component Browser and the cutting plane. By default, the component's axes are placed as though the component will be inserted from the Component Browser in the exact orientation as the component is in while you create the component. Generally, you will not want to change this orientation unless, for example, you are creating a window component independent of other geometry and in the vertical plane (blue). In this case, the bottom of the window will be on the red/green plane. The red/green plane is the gluing and cutting plane. As such, this window's bottom will therefore want to align and cut into a face when placed in the model from the Component browser. As mentioned previously, however, it is best to create window and door components within the context of a surface type where they will ultimately be placed, such as a wall, so you do not have to reorient the axes. Follow the last two steps in this list if you need to reoriented the component axes.

5. (optional) Move the mouse around the origin to redefine the orientation of the component. The axes of the component will rotate suggesting a new orientation for the component. The cut plane will also move to represent where the component will cut into a face when placed vertically or horizontally. For example, if you rotate the axes such that red is up and green is to the left, the component will be inserted horizontally to its orientation when created. The following image shows a window component during component creation. The component's axes have been reoriented such that the cutting plane is now parallel to the front of the window.



6. (optional) Click to set the new orientation.

## Component Definitions and Instances

A Component Definition and a Component Instance are created when a component is created using the Create Component dialog box. Components Definitions define or provide a blueprint for how all components of a specific type, called instances, appear and behave within the Drawing Area (whether they can be glued to faces, where insertion points appear, and so on). Component Definitions are represented by thumbnail images in the Component Browser.

Component Instances are components definitions that have been inserted in the Drawing Area (called instancing). Component Instances all look like and have the same default behavior as the Component Definition, but, once brought into the Drawing Area, can be rotated, scaled, and painted independently of other instances.

## Inserting Components

There are a variety of different ways to insert component instances in SketchUp.

### Inserting Components from Component Browser

Pre-defined Components are most often inserted from the Components Browser into a model. See the Component Browser for further information.

### Insert a Component From an External SketchUp (.skp) File

You can also insert a component from an external SketchUp file. To insert a component from an external SketchUp file:

1. Select the File > Import > 3D Model... menu item. The Open file dialog box appears.
  2. Select the type of file to import from the Files of Type drop-down list (.skp).
  3. Click Open. The cursor changes to the Move Tool anchored to the component's insertion point.
-  **Tip -** *The component axes origin is the default insertion point for a Component. Change the location of the component's axes before you insert the component to change the default insertion point.*
4. Move the mouse to the location in the Drawing area where you want to place the component.
  5. Click the mouse button again to release the component.

### Insert a Component From the File Explorer

Finally, you can also insert a component from the File Explorer. To insert a component from the File Explorer:

1. Locate the icon representing the file you want to insert.
2. Click and hold the mouse button on the icon.
3. Drag the icon into the Drawing Area. The cursor changes to the Move Tool anchored to the component's insertion point.

4. Release the mouse button to place the component into your model.

## Editing a Component Instance as a Whole

You can edit the component as a whole or edit individual entities within a component. Editing or modifying the component instance as a whole affects only the component instance, not the component definition or other instances.

### Scaling a Component

Scaling a component as a whole scales the individual component instance, not the component definition, allowing you to have many differently scaled instances of the same component in your model.

A component can become skewed when you scale the component in multiple directions. You can reset both a components scale and skew using the Reset Scale and Reset Skew context menu items.

### Flipping a Component

You can flip (or mirror) a component along its axes using the Flip Along component context menu item. Choose Component's Red, Component's Blue, or Component's Green depending on the direction to flip the component.

### Rotating a Component Using the Move Tool

Components can be rotated using either using the Move tool or Rotate tool. The first rotation method rotates the component about its center of mass and in the planes of the component bounding box. The second rotation method allows you to specify precise rotate planes and center of rotation. See the Rotate Tool for further information on this second rotation method. To rotate a component using the Move Tool:

1. Select the Move Tool. The cursor will change to a four-way arrow.
2. Move the cursor over a face that is perpendicular to the desired axis of rotation. Four rotation handles and a protractor appear on the face.
3. Click on a rotation handle
4. Rotate the Component.

## Editing Entities Within a Component Instance

Editing the entities within a component requires you to enter the component's context. Editing or modifying the entities within a component instance affects the component definition and other instances of the component. To edit the entities within a component:

1. Select the Component Instance > Edit Component command from the Edit Menu to edit the component. Alternatively, context click on the currently selected component and select Edit Component from the context menu. An edit bounding box will surround the component and entities exterior to the component will turn grey.



**Tip - Double-click on the component to edit the component.**

2. Make changes to entities within the component. Any changes while in the context of the component affects each component instance and the component definition. You can also perform inference alignments to geometry outside of the component while you are editing the component.

3. Select the Close Group / Component command from the Edit Menu to end the edit session. Alternatively, context click on the component's bounding box and select Close component from the context menu.



**Tip** - Click outside of the component to close the component.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Exploding a Component

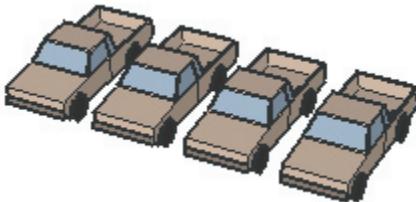
You can explode a Component entity to break it back into its original entities. To explode a component:

1. Select the Select Tool. The will change to an arrow.
2. Select the component you want to explode.
3. Select the Component Instance > Explode command from the Edit Menu. Alternatively, context click on the currently selected group and select Explode from the context menu. The Component will be split back into its entities.

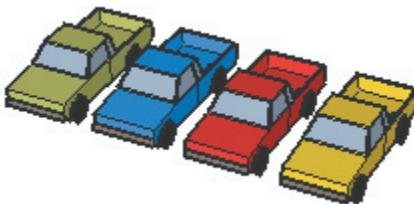
Elements within components that were placed adjacent to other geometry might become joined to elements exterior to the component when the component is exploded.

## Applying Materials to Components

Any geometry inside a component that is painted with the default material will be painted when you paint the instance as a whole. Therefore, you can have entities within a component painted individually while other elements painted using the material assigned to the entire component. The following image contains four cars all enclosed in components. The tires, bumper, and windshield have been painted black when editing the component.



Each entire component instance was then painted after being edited, turning each of the faces with the default material to the color applied on the group (green, blue, red, and yellow from left to right):



## Moving the Insertion Point

The Component Browser switches the insertion point from the origin of the component's axes to another point on the component when you use the move tool to select a component, that is already in your model, using a different (non-origin) point. For example, if you have a 3D rectangle component, the origin and insertion point will be the lower left-hand corner. However, if you insert this component and then move it by a midpoint, the insertion point changes to the mid point (while the origin remains at the lower left hand corner).

This feature is useful when aligning components, such as cabinet components in a kitchen. Consider the following:

1. Insert one kitchen cabinet component with the insertion point at the lower left corner of the component.
2. Insert a second instance of the same cabinet to the left of the previously inserted cabinet (so that the two cabinets touch). Notice that it is hard to accurately place the second cabinet because the insertion point is at the origin at the lower left corner of the component (away from where the two cabinets will touch).
3. Reposition the second kitchen cabinet away from the first.
4. Select the move tool and click the lower-right corner of the second kitchen cabinet component.
5. Now move the second kitchen cabinet such that it is aligned on the left-side of the first cabinet. This process should be easier now that the insertion point has been moved to the lower-right corner of the component. If you drag a third instance of the same cabinet into the model, the insertion point will now be at the lower-right corner (making it easier to place more cabinets to the left of each other).

## Resetting Component Axes

Right click on the component in the component browser and select Reset Insert Point if you want to reset the insertion point to the origin of the component's axes.

## Component Context Commands

Access context menu items by context clicking on an entity. The component context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Lock/Unlock

The Lock menu item is used to lock the component so that it cannot be moved or edited. Additional copies of the component can be dragged into the work area from the component browser, however.

The Unlock menu item is used to unlock the component so it can be edited.

### Edit Component

The Edit Component menu item is used to edit the currently selected component. You can also use the Select Tool to double-click on the component, or select the component and press Enter or Return to edit a component.

### Make Unique

The Make Unique menu item converts the currently selected component into a unique component, with a new definition. The original component definition and all other instances will not be affected when you edit the component after using the Make Unique menu item.

### Close Component

The Close Component menu item is used to end an editing session and close the component. You can also click outside of the component with the Select Tool, or press the ESC key to close the component.

### Explode

The Explode menu item is used to break the entity into the separate entities.



## Unglue

The Unglue menu item is used to free a component from a face (if you've attached a hole-cutting component to a face).

## Reload

The Reload menu item is used to update the currently selected component with a new revision from the file system.

## Save As...

The Save As... menu item is used to save the selected component to a separate SketchUp file, with a new name and location in your file system. This menu item is commonly used to create component files that can be included in the component library or reused in other models.

## Change Axes

The Change Axes menu item is used to redefine the origin axes of the selected component. This option is similar to changing the local coordinate system in other 3D applications.

## Reset Scale

The Reset Scale menu item is used to remove any scaling operations applied to the component.

## Reset Skew

The Reset Skew menu item is used to remove any skewing applied to the component.

## Scale Definition

The Scale Definition menu item applies any scaling operations, to the currently selected component, to the component definition in the Component Browser.

## Intersect With Model

The Intersect With Model menu item is used to intersect two elements, such as a box and a tube, and automatically create new faces where the elements intersect. These faces can then be pushed, pulled or deleted to create new geometry.

## Flip Along

The Flip Along menu item is used to mirror the selected geometry along an axis . You can use the Axis Tool to reposition the drawing axes, allowing you to flip in virtually any direction.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

# Component Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note** - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

## Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

## Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.



**Caution** - Locked groups or components cannot be reassigned to a new layer. You will receive a warning message when you attempt to move locked groups or components to a new layer.

## Name

The Component Name displays the name of the currently selected component instance.

## Definition Tab

The Definition Tab displays an optional description in addition to other component definition options.

**Name:** The Name field can contain the name of the component definition.

**Description:** The Description field can contain a description of the component.

**Glue to:** The Glue to drop-down list is used to identify the surfaces where your component can be placed. For example, a standard door might only be glued to surfaces in the horizontal planes.

**Cut Opening:** The Cut Opening option allows the Component to create openings in the face into which it is placed. For example, a door or window component might be set to cut an opening in any wall where they are placed.



*Note - The location where the hole will exist corresponds to the red/green plane for the component. See the Set Plane option, in this topic, for further information.*

**Always face camera:** The Always face camera option allows the component to take on billboard behavior by drawing the component as a 2D form. This option increases performance by eliminating the need to render the component as a 3D model.



*Note - Components with the Always face camera option enabled cannot have gluing behavior.*

**Reload:** Click on the reload button to reload (update) the component instance after its component definition has changed.

## Statistics Tab

The Statistics reports quantities of various elements, such as faces, edges, construction lines, within the component. Select All geometry from the drop-down list to see statistics for all geometry in the group. Alternatively, select Components from the drop-down list to see statistics for all components in the component. Finally, click on Show nested to see components nested within the currently selected component.

## Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Locked

The Locked check box is used to lock the component so that it cannot be moved or edited. Additional copies of the component can be dragged into the work area from the component browser, however.

Unclick the Locked check box to unlock the component so it can be edited.

### Cast Shadows

The Cast Shadows menu item allows the component to cast a shadow.

### Receive Shadows

The Receive Shadows menu item allows the component to receive a shadows cast by other entities.

# Construction Line Entities

Construction Line entities are infinite dashed lines used as guides to draw precisely. Construction Lines do not interfere with regular geometry. Construction lines can also be hidden and erased independent of regular geometry. Draw construction lines with the Tape Measure Tool.

You can use the Move, Rotate, and Erase Tools to reorient a Construction Line. But, you cannot resize a Construction Line because Construction Lines are infinite in length.

## Hiding/Erasing All Construction Lines

Construction lines are usually created as a temporary means to build a portion of your model. Keeping too many Construction Lines in your model can decrease SketchUp's inference accuracy and display performance, so you might want to hide Construction Lines as you work or delete all Construction Lines at once when you have finished your model.

Use the Edit > Construction Geometry > Hide, unhide, and erase to hide, unhide, and erase all construction geometry in the current context.

## Construction Line Context Commands

Access context menu items by context clicking on an entity. The Construction Line context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

## Construction Line Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box:

### **Layer**

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### **Hidden**

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.



# Construction Point Entities

Construction Point entities are finite dashed lines with end points as guides to draw precisely. Construction points do not interfere with regular geometry. Construction points can also be hidden and erased independent of regular geometry. Draw construction points with the Tape Measure Tool.

You can use the Move, Rotate, and Erase Tools to reorient a Construction Point.

## Hiding/Erasing All Construction Points

Construction points are usually created as a temporary means to build a portion of your model. Keeping too many construction points in your model can decrease SketchUp's inference accuracy and display performance, so you might want to hide construction points as you work or delete all construction points at once when you have finished your model.

Use the Edit > Construction Geometry > Hide, unhide, and erase to hide, unhide, and erase all construction geometry in the current context.

## Construction Point Context Commands

Access context menu items by context clicking on an entity. The construction point context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

## Construction Point Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box:

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

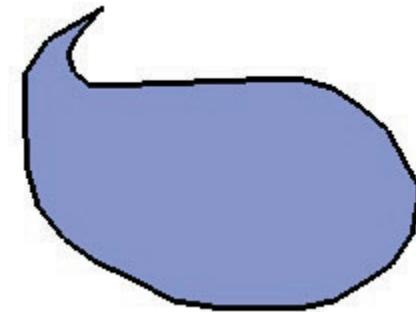
# Curve Entities

SketchUp's Curve entities are a combination of multiple line segments that are connected together. These entities act as a single line in that they can define the edge of a face and also divide a face. Additionally, selecting one segment of the curve selects the entire Curve entity. Draw curves with the Freehand Tool.



**Note -** You can explode a curve into regular edge segments using the Explode Curve context menu item (see the Curve Context Menu Items section later in this topic).

The following image contains a Curve entity. Curves can begin and end at the same point (as in the following image) or start and end at different points.



**Note -** The model in the previous image contains two entities: a Face entity (in the middle) surrounded by a Curve entity (the continuous edge).

## Editing a Curve Entity

You can change the length of a Curve entity, that does not yet bound a face, by using the Move Tool. To edit a Curve entity:

1. Select the Move Tool. The cursor will change to an four-way arrow.
2. Move over the Curve entity to locate an endpoint of the Curve entity.
3. Click and hold on the endpoint of the Curve entity.
4. Move the cursor to adjust the length of the Curve entity.

## Curve Context Menu Items

Access context menu items by context clicking on an entity. The curve context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Select

The Select submenu contains several options for selecting geometry related to the entity.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected curve.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected curve.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

### Soften

The Soften menu item is used to soften a faceted surface. Softened edges are not visible unless displayed in profile.

### Explode Curve

The Explode Curve menu item is used to break the entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

## Convert to Polygon

The Convert to Polygon menu item is used to convert the entity into a polygon arc, which will remain editable yet generate standard facets when extruded.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Curve Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note** - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Segments

The Segments field displays the number of segments in the selected entity. Type new values in this field to change the number of segments for the entity.



**Note** - Curve segmentation can only be changed when the Curve entity is part of 2 dimensional geometry (not part of a curved surface).

### Length

The Length field displays the length of the currently selected line.

## Hidden

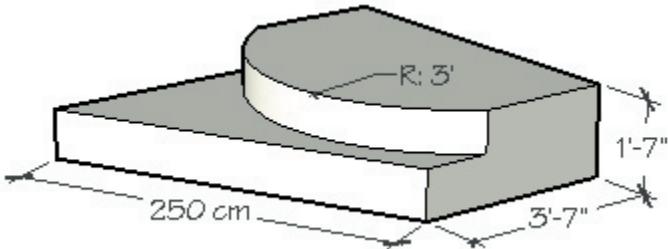
The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Cast Shadows

The Cast Shadows menu item allows the entity to cast a shadow.

# Dimension Entities

Dimension entities are finite lines with length information allowing you to quickly and effectively communicate key dimensions of your model. Dimensions can move and update automatically as you make changes to your model. Draw dimensions with the Dimension Tool.



Each Dimension Object lies in its own plane, which is determined by the entity from which it is drawn and the direction used when it was created.

Dimension display properties, for all dimension entities in your model, can be adjusted using the Dimension panel of the Model Info dialog box.

## Changing Dimension Text

A Dimension entities text displays the dimension by default. You can, however, change the dimension text to include additional information, such as text. Include the symbols <> anywhere in your text to insert the dimension. For example, "The length of this line is <>."



**Caution -** Dimensions that have lost their direct link to geometry or that have had their text edited might not show accurate measurements. Select the Highlight non-associated dimensions option in the Dimension panel of the Model Info dialog box to highlight these dimensions in a specified color.

## Dimension Object Context Menu Items

Access context menu items by context clicking on an entity. The dimension context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Edit Text

The Edit Text menu item is used to edit the dimensions text. Type in the new text and press Enter or Return to accept your changes.

## Dimension Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box:

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

### Style

The Style button is used to invoke the Dimensions panel of the Model Info dialog box for selecting dimension text and leader line type.

# Face Entities

Face entities are flat plane-like entities that combine to form the 3D geometry in a SketchUp model. Faces are automatically created when any three or more intersecting lines or edges are in the same plane (an infinite flat 2D space), or coplanar.

The edges that bound a face remain when you delete a face. However, a face is erased automatically when you delete one of its bounding edges. SketchUp will create new edges and faces using the Auto-fold feature if you alter one of the edges of a face so that it is no longer co-planar with that face.

Draw faces with the Line Tool, Arc Tool, Freehand Tool, Rectangle Tool, Circle Tool, or Polygon Tool. The following image was created simply by joining arcs and lines to form the edges and subsequent faces.



## Face Context Menu Items

Access context menu items by context clicking on an entity. The face context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

## Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

## Select

The Select submenu contains several options for selecting geometry related to the entity.

**Bounding Edges:** The Bounding Edges menu item is used to select all of the edges currently bounding the selected face.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected face.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected face.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

**All with same Material:** The All with same Material menu item is used to select all of the faces with the same material.

## Area

The Area sub-menu of the Face context menu is used to calculate the surface area in your SketchUp model. The results of the calculation are displayed in a Area dialog box.

**Selection:** The Selection menu item is used to calculate the total surface area of the currently selected face.

**Layer:** The Layer menu item is used to calculate the total area for all faces in the layer of the currently selected face.

**Materials:** The Materials menu item is used to calculate the total area of the faces in the entire model, hidden or unhidden, with the material of the currently selected face.

## Intersect With Model

The Intersect With Model menu item is used to intersect two portions of geometry, such as a box and a tube, automatically creating new faces at the intersection. These faces can then be pushed, pulled or deleted to create new geometry.

## Align View

The Align View menu item is used to align the SketchUp Camera to the currently selected face.

## Align Axes

The Align Axes menu item is used to align the Drawing Axes to the currently selected face.

## Reverse

The Reverse menu item is used to flip the front and back sides of the currently selected face.

## Orient Faces

The Orient Faces menu item is used to automatically orient all the connected faces of your model to the orientation of the currently selected face. This option could lead to unexpected results when an edge of a face bounds three or more other faces.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Texture

The Texture submenu contains several options for repositioning textures on a face. This menu item only appears when a texture is painted on the face. See Position Texture Tool for additional information.

**Position:** The Position menu item is used to manipulate textures (reposition, stretch, skew, and so on) directly on a face.

**Reset Position:** The Reset Position menu item is used to reset the texture to its previous state.

**Projected:** The Projected menu item is used wrap images and textures over forms as though projected onto the form.

## Face Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note -** Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

## Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box. There are two default material swatches for faces because faces have both a front and a back side. The left swatch represents the front-side material. The right swatch represents the back-side material.

### **Layer**

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### **Area**

The Area field displays the area of the selected entity. Use the Area > Selection context menu item to calculate the area of multiple selected entities.

### **Hidden**

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

### **Cast Shadows**

The Cast Shadows menu item allows the face of the entity to cast a shadow.

### **Receive Shadows**

The Receive Shadows menu item allows the face of the entity to receive a shadows cast by other entities.

# 3D Polyline Entities

SketchUp's 3D Polyline entities are curve-like entities that do not generate inference snaps or affect geometry in any way. Freehand sketches are usually used for tracing imported drawings, 2D sketching, or for decorating your model. Draw 3D Polyline entities with the Freehand Tool.



**Note** - You can explode a freehand sketch into regular edge segments using the Explode context menu item (see the 3D Polyline Context Menu Items section later in this topic).



**Note** - A 3D Polyline looks just like a Curve entity, but is thinner.

## 3D Polyline Context Menu Items

Access context menu items by context clicking on an entity. The 3D polyline context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Explode

The Explode menu item is used to break the entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

## 3D Polyline Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box:

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

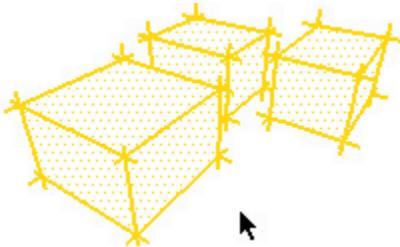
# Group Entities

Groups are entities that can hold other entities. Groups are commonly used to combine several entities as a single entity for the purposes of performing a quick operation with the combination (such as a copy). Use the Make Group menu item from the Edit menu to create a group from the currently selected entities.

## Creating a Group

Groups are useful for combining entities to perform a quick operation such as copy or move operations. To create a group:

1. Select the Select Tool. The will change to an arrow.
2. Click and hold the mouse button a short distance away from the entities you want to select to start a selection box.
3. Drag the mouse to the opposite corner of the selection starting point.
4. Release the mouse button when all of the elements are either partially included (left-to-right selection) or fully included (right-to-left selection) in the selection box.



5. Select the Make Group menu item from the Edit Menu. Alternatively, context click on the currently selected entities and select Make Group from the context menu. The geometry you selected appears grouped within a highlighted bounding box.



**Note** - The Make Group operation disconnects any geometry that was connected to the grouped geometry prior to placing the geometry in the group. The disconnected geometry is maintained outside of the group's context.



**Note** - You can make group hierarchies by grouping other Group entities within a group. Additionally, you can mix your hierarchies by including components and groups within other components and groups.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Exploding (Ungrouping) a Group

You can explode (ungroup) a Group entity to break it back into its original entities. To explode a group:

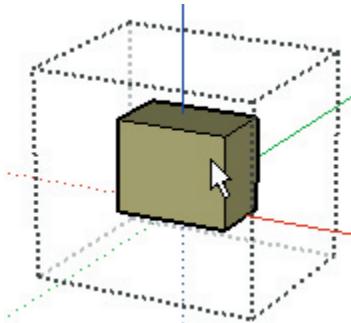
1. Select the Select Tool. The will change to an arrow.
2. Select the group you want to explode.
3. Select the Group > Explode command from the Edit Menu. Alternatively, context click on the currently selected group and select Explode from the context menu. The Group will be split back into its entities.

Elements within groups that were placed adjacent to other geometry might become joined to elements exterior to the group when the group is exploded.

## Editing a Group

Groups can be opened for editing, placing you in the Group's context. To edit a group:

1. Select the Group > Edit Group command from the Edit Menu to edit the group. Alternatively, context click on the currently selected group and select Edit Group from the context menu. An edit bounding box will surround the group and entities exterior to the group will turn grey.



**Tip** - Double-click on the group to edit the group.

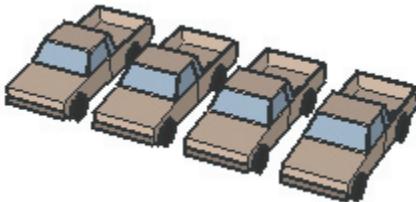
2. Make changes to entities within the group. Any changes while in the context of the group only affects the Group entity. You can, however, perform inference alignments to geometry outside of the group while you are editing the group.
3. Select the Close Group / Component command from the Edit Menu to end the edit session. Alternatively, context click on the currently selected group's bounding box and select Close Group from the context menu.



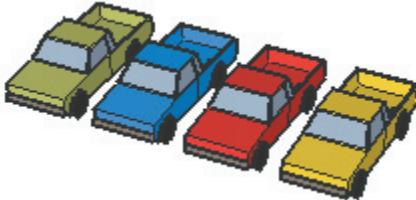
**Tip** - Click outside of the group to close the Group.

## Applying Materials to Groups

Any geometry inside a group that is painted with the default material will be painted when you paint an unexploded group. Therefore, you can have entities within a group painted individually while other elements painted using the material assigned to the entire group. The following image contains four cars all enclosed in groups. The tires, bumper, and windshield have been painted black when editing the group.



Each entire Group entity was then painted after being edited, turning each of the faces with the default material to the color applied on the group (green, blue, red, and yellow from left to right):



## Group Context Commands

Access context menu items by context clicking on an entity. The group context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Lock/Unlock

The Lock menu item is used to lock the component so that it cannot be moved or edited. Additional copies of the component can be dragged into the work area from the component browser, however.

The Unlock menu item is used to unlock the component so it can be edited.

### Edit Group

The Edit Group menu item is used to edit the currently selected group. You can also use the Select Tool to double-click on the group, or select the group and press Enter or Return to edit a group.

### Close Group

The Close Group menu item is used to end an editing session and close the group. You can also click outside of the group with the Select Tool, or press the ESC key to close the group.

### Explode

The Explode menu item is used to break the entity into the separate entities.

### Make Component

The Make Component menu item converts the Group into a new Component entity (component definition).

### Unglue

The Unglue menu item is used to unattach a Group from a face.

### Reset Scale

The Reset Scale menu item removes any scaling operations applied to the Group.

### Reset Skew

The Reset Skew menu item removes any operation that skewed the group (such as scaling the Group multiple times).

### Flip Along

The Filp Along menu item mirrors the selected geometry along the group axis you specify. The groups axes are the axial directions that existed when the group was originally created. You can also use the Axis Tool to reposition the global Drawing Axes, allowing you to flip selected geometry in virtually any direction.

### Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Group Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note** - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

## Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

## Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.



**Caution -** Locked groups or components cannot be reassigned to a new layer. You will receive a warning message when you attempt to move locked groups or components to a new layer.

## Name

The Name displays the name of the currently selected group.

## Statistics Table

The Statistics Table reports quantities of various elements, such as faces, edges, construction lines, within the group. Select All geometry from the drop-down list to see statistics for all geometry in the group. Alternatively, select Components from the drop-down list to see statistics for all components in the group.

## Show Nested

Click on Show nested to see groups nested within the currently selected group.

## Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Locked

The Locked check box is used to lock the component so that it cannot be moved or edited. Additional copies of the component can be dragged into the work area from the component browser, however.

Unclick the Locked check box to unlock the component so it can be edited.



## Cast Shadows

The Cast Shadows menu item allows the face of the entity to cast a shadow.

## Receive Shadows

The Receive Shadows menu item allows the face of the entity to receive a shadows cast by other entities.

# Image Entities

Image entities are essentially special groups consisting of a face with an applied bitmap image material. Image Objects can be moved, rotated and scaled. They can also be stretched horizontally or vertically, but they cannot be made non-rectangular. Import Images using the File > Import > 2D Graphic... menu item.

## Image Formats

SketchUp supports the following formats for import as an Image Object: JPEG, PNG, TGA, BMP, and TIFF. Some formats may be more appropriate than others for certain types of images and applications.

## Inserting Image Entities

Images can be added to SketchUp models in one of two ways. First, you can use the **File > Import > 2D Graphic...** menu item, which will open a File Open dialog allowing you to navigate to the file you want. Alternately, you can simply drag and drop from the File Explorer directly into your drawing area.



**Note -** Image Objects can also be inserted as textures ( *File > Insert > Image as Texture* ). For additional information on using images as textures, see the *Texture Positioning* topic.

## Image Proportions

By default, Image Objects retain the proportions of the file from which they are derived. While inserting an image, you can hold down the Shift key to de-constrain the proportions. You can also use the Scale Tool to alter the image object's proportions after it is placed.

## Image File Size Versus Quality

When you add an Image Object to your scene, the image file it is based on becomes embedded in your SketchUp document. This allows you to send your SketchUp files to others without any data loss through misplaced linked files, but it also means that your files can quickly balloon to unwieldy sizes. When inserting images, try to keep file size as small as possible:

### Resolution

Resolution can have a large impact on image size. Try to use only as much resolution as you need, and no more. You may sometimes find that even a pixelated, low resolution file can be sufficient to provide the information you need from a photograph, sketch, or drawing. You can also cut down on image file size by converting your images to grayscale before inserting them into SketchUp.

Also, the resolution of Image Objects is limited to the largest texture that OpenGL can handle. For most systems this limit is 1024 x 1024 pixels. This should be sufficient for most purposes, but if higher resolution is required, you can always stitch together multiple Image Objects.

### Image File Compression

Another way to minimize file size is to use compressed file formats such as JPEG and PNG. These take up far less space both on disk and inside a SketchUp file.

## Image Object Context Menu Items

Access context menu items by context clicking on an entity. The image context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Explode

The Explode menu item is used to break the entity into the separate face and material. Exploding an Image entity creates a material from the image in the Material Browser.

### Export

The Export menu item is used to save the image entity to a file that can be edited in an image editor.

### Reload

The Reload menu item is used to refresh an Image entity that was manipulated in a separate image editor.

### Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Shadows

The Shadows submenu contains cast and receive shadow options for the Image entity.

**Cast:** The Cast Shadows menu item allows the face of the entity to cast a shadow.

**Receive:** The Receive Shadows menu item allows the face of the entity to receive a shadows cast by other entities.

## Unglue

If you've attached an image object to a face, it cannot be moved off that face. The Unglue command makes it free to move off that face.

## Use As Material

The Use As Material menu item is used to create a material, in the In Model section of the Material Browser, from the image.

## Image Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note -** Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Name

The Name field displays the filename of the Image entity.

### Size in pixels

The Size in pixels field displays the pixel dimensions of the currently selected image.

### Width and Height

The Width and Height fields displays the dimensions of the currently selected image. Type in a new width or height to rescale the Image entity proportionally. Or, click on the lock icon to rescale the Image entity's width and height independently (nonproportional scale).



## Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

## Cast Shadows

The Cast Shadows menu item allows the face of the entity to cast a shadow.

## Receive Shadows

The Receive Shadows menu item allows the face of the entity to receive a shadows cast by other entities.

# Line Entities

Lines, also referred to as edges, form the structural foundation of all models. Draw lines with the Line Tool.

## Editing a Line Entity

You can edit the length of a Line entity, that does not yet bound a face, by using the Move Tool. To edit an Line entity:

1. Select the Move Tool. The cursor will change to a four-way arrow.
2. Move over the Line entity to locate an endpoint of the Line entity.
3. Click and hold on the endpoint of the Line entity.
4. Move the cursor to adjust the length of the Line entity.

You can also adjust the length using the Line entity's Entity Info dialog box.

## Line Context Menu Items

Access context menu items by context clicking on an entity. The line context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Select

The Select submenu contains several options for selecting geometry related to the entity.

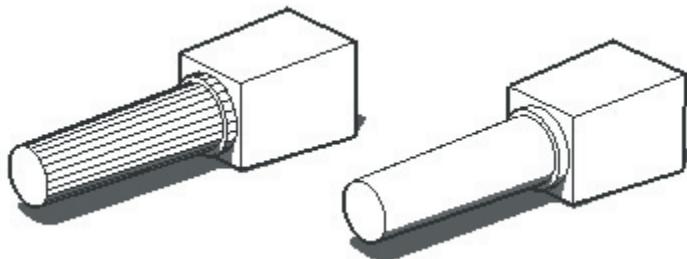
**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected line.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected line.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

## Soften

The Soften menu item is used to soften a edge. Softened edges are not visible unless displayed in profile.



## Divide

The Divide menu item is used to divide an entity into any number of equal segments. This menu item is only available for entities that are not connected to other geometry. To divide an entity:

1. Context click on the entity to invoke the entity's context menu
2. Select Divide. A string of red dots appears along the line segment.
3. Drag the cursor back and forth along the length of the entity to increase or decrease the number of divisions. A ToolTip displays the number and length of segments that will be created if you pause briefly on the entity.



**Note -** The number of divisions is also displayed in the Value Control Box (VCB). Type in the number of desired segments followed by the Enter key to manually divide the entity.

4. Click again to divide the entity. The entity will be broken into separate segments.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Edge Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note -** Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Length

The Length field displays the length of the currently selected line.

### Soft

The Soft option is used to render the edge as a soft edge.

### Smooth

The Smooth option is used to render the faces connected to the edge with a varying tonal value.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

### Cast Shadows

The Cast Shadows menu item allows the entity to cast a shadow.

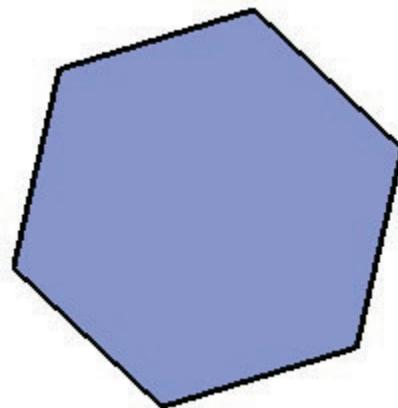
# Polygon Entities

SketchUp's Polygon entities are similar to Faces entities, but can have 3 or more sides. These entities act as a single line in that they can define the edge of a face and also divide a face. Additionally, selecting one segment of the polygon selects the entire Polygon entity. However, all inference techniques will operate on the Polygon as though it were comprised of segments. Draw polygons with the Polygon Tool.



**Note** - You can explode a polygon into regular edge segments using the Explode Curve context menu item (see the *Polygon Context Menu Items* section later in this topic).

Polygon entities are comprised of a radius and number of segments. The following image shows a six-sided polygon.



**Note** - The model in the previous image contains two entities: a Face entity (in the middle) surrounded by a Polygon entity (the six-sided edge).

## Editing a Polygon Entity

You can edit the radius of a circle in which the polygon is inscribed, that does not yet bound a face, by using the Move Tool. To edit a Polygon entity:

1. Select the Move Tool. The cursor will change to a four-way arrow.
2. Move over the Polygon entity to locate the midpoint of a side. At least one midpoint of the polygon will allow you to resize the entity.
3. Click and hold on the midpoint of the Polygon entity.
4. Move the cursor to adjust the radius of the Polygon entity. Click and hold on another midpoint if the polygon does not resize. Try each midpoint until you find the midpoint that will resize the entity.

You can also adjust the radius and number of segments using the Polygon entity's Entity Info dialog box.

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## Editing an Extruded Polygon

When you use the Push/Pull Tool to extrude a 2D face that includes a polygon, it extrudes a special polygonal curved face set which can also have its radius edited. Use the Move Tool to reposition one of the control edges, and the polygonal curved face set radius (as well as the radii of the two polygon entities that define it) will be adjusted accordingly.

### Polygon Deformation

If a polygon is deformed in a way that destroys its radial definition, such as with a non-uniform scale operation, it will become a non-parametric Polyline Curve. Polyline Curves can no longer be edited as polygons.

## Polygon Context Menu Items

Access context menu items by context clicking on an entity. The polygon context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Select

The Select submenu contains several options for selecting geometry related to the entity.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected polygon.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected polygon.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

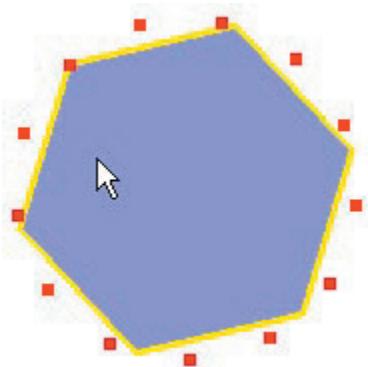
### Soften

The Soften menu item is used to soften a faceted surface. Softened edges are not visible unless displayed in profile.

## Divide

The Divide menu item is used to divide an entity into any number of equal segments. This menu item is only available for entities that are not connected to other geometry. To divide an entity:

1. Context click on the entity to invoke the entity's context menu
2. Select Divide. A string of red dots appears along the line segment.
3. Drag the cursor back and forth along the length of the entity to increase or decrease the number of divisions. A ToolTip displays the number and length of segments that will be created if you pause briefly on the entity.



**Note -** The number of divisions is also displayed in the Value Control Box (VCB). Type in the number of desired segments followed by the Enter key to manually divide the entity.

5. Click again to divide the entity. The entity will be broken into separate segments.

## Explode Curve

The Explode Curve menu item is used to break the entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

## Zoom Extents

The Zoom Extents menu item is used to zoom your view to a distance where the whole entity is visible and centered in the drawing window.

## Point at Center

The Point at Center menu item is used to place a point at the center of your arc or circle entity. This menu item is generated by a Ruby script.

## Polygon Entity Info

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note -** Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Radius

This Radius field displays the radius of the selected entity. Type new values in to this field to change the radius of the entity.

### Segments

The Segments field displays the number of segments in the selected entity. Type new values in this field to change the number of segments for the entity.



**Note -** Line segmentation can only be changed when the Line entity is part of 2 dimensional geometry (not part of a curved surface).

### Length

The Length field displays the length of the currently selected polygon.

### Hidden

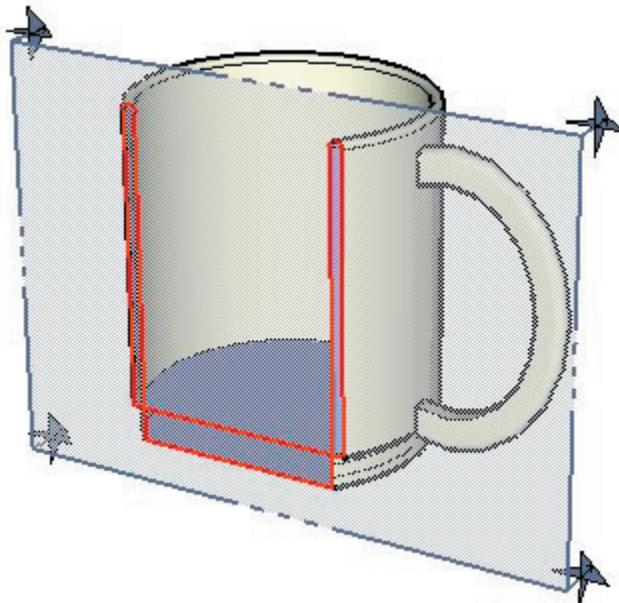
The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

### Cast Shadows

The Cast Shadows menu item allows the face of the entity to cast a shadow.

# Section Plane Entities

Section Planes are special entities that control the SketchUp section cutting effect. Their position in space and in relation to groups and components determine the nature of the section cut effect. The following image shows a rectangular section plane creating a section cut effect in a model of a cup.



## Section Plane Context Commands

Access context menu items by context clicking on an entity. The section plane context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

#### Reverse

Reverses the direction that the section is facing, thereby flipping the Section Cut Effect.

#### Activate Cut

Activates and de-activates the Section Plane Object, as indicated by the check mark. Activating a Section Plane will automatically de-activate all other planes in that context.

#### Align View

Aligns the camera view to the plane of the section cut. This is handy for creating section perspectives. Selecting Align View from the Section Plane Context will align the SketchUp Camera with the selected Section plane.

#### Create Group from Slice

Generates new edges, encapsulated within a Group, wherever the section intersects with a face.

### Section Plane Entity Info Dialog Box

With a Section Plane selected, you can see and change its attributes in the Entity Info dialog box.

#### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

#### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

#### Style

The Style button is used to invoke the Section Planes panel of the Model Info dialog box for selecting sectioning colors and cut line width.

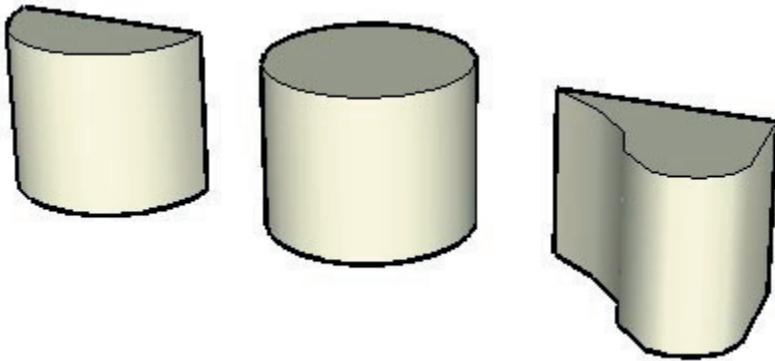
# Surface Entities

Surface entities are a number of faces joined together to give the impression of roundness (using soft edges). Although the individual planar faces that make up a curved face set are still separate entities, they can act like a single group when selected or painted.

There are three types of surfaces, though the Entity Info dialog box will recognize each as just a Surface entity:

- Arc surface. An arc surface is generated when a face with an Arc entity has one or more of its edges extruded (usually with the Push/Pull Tool).
- Cylindrical surface. A cylindrical surface is similar to an arc surface, but is generated when a circular face is extruded to form a cylinder.
- Polyface surface. A polyface surface is generated when a face with a Polyline Curve entity has one or more of its edges extruded (usually with the Push/Pull Tool). You can also create one manually by softening edges with the Erase Tool or using the Soften/Smooth Control. Polyface surfaces do not have special editable properties, but respond to tool operations as a single entity.

The following image contains the three types of surfaces, an arc surface, cylindrical surface, and polyface surface, from left to right.



## Creating an Arc Surface

As mentioned previously, arc surfaces are extruded faces with one or more arc edges. To draw an arc surface:

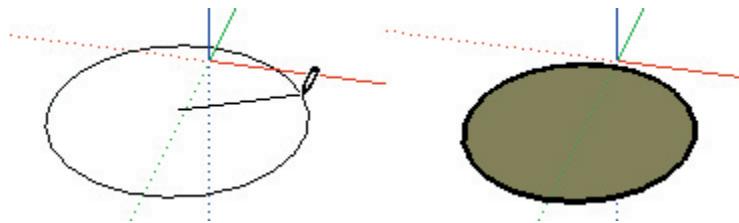
1. Select the Arc Tool. The cursor changes to a pencil with an arc.
2. Click to place the starting point of your arc.
3. Click again to place the ending point of your arc.
4. Move your mouse to adjust the bulge distance. Or, optionally type in values for the chord length, bulge distance, radius, and number of segments in the VCB.
5. Click again to set the bulge distance.

6. Select the Line Tool. The cursor changes to a pencil.
7. Click at one end of the arc set the starting point of your line.
8. Click at the other end of the arc to set the ending point of your line. This step completes a face consisting of an arc and a straight line.
9. Select the Push/Pull Tool. The cursor will change to a 3D rectangle with an up arrow.
10. Click on the face
11. Move the cursor up to create an arc curve in the positive direction or move the cursor down to expand the arc curve in the negative direction.
12. Click again when the arc curve has reached the desired size. The vertical curved surface is an arc surface.

## Creating a Cylindrical Surface

As mentioned previously, cylindrical surfaces are extruded circles. To draw a cylindrical surface:

1. Select the Circle Tool. The cursor changes to a pencil with a circle.
2. Click to place the center point of the circle.
3. Move the mouse out from the center point to define the radius of your circle. As you do so, the radius value is displayed dynamically in the VCB and can be specified by typing in a length value followed by the Enter or Return key. You can also specify the segmentation for the circle in the Value Control Box (VCB).



4. Click a second time to finish the circle.
5. Select the Push/Pull Tool. The cursor will change to a 3D rectangle with an up arrow.
6. Click on the face
7. Move the cursor up to create cylinder in the positive direction or move the cursor down to expand the cylinder in the negative direction.
8. Click again when the cylinder has reached the desired size. The vertical surface is a cylindrical surface.

## Creating a Polyface Surface

As mentioned previously, polyface surfaces are extruded faces with one or more polyline curve edges. To draw an polyface surface:

1. Select the Freehand Tool. The cursor will change to a pencil with a polyline curve.
2. Click and hold to place the starting point of your polyline curve.
3. Drag the cursor to draw a polyline curve. Do not close the curve.
4. Release the mouse button to stop drawing.
5. Select the Line Tool. The cursor changes to a pencil.
6. Click at one end of the polyline curve to set the starting point of your line.
7. Click at the other end of the polyline curve to set the ending point of your line. This step completes a face consisting of an arc and a straight line.
8. Select the Push/Pull Tool. The cursor will change to a 3D rectangle with an up arrow.
9. Click on the face
10. Move the cursor up to create the polyface surface in the positive direction or move the cursor down the expand the polyface surface in the negative direction.
11. Click again when the polyface surface has reached the desired size. The vertical curved surface is an polyface surface.

## Surface Context Menu Items

The context menu items for a Surface entity are the same as the context menu items for an individual face. See the Face entity for more information.

## Surface Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



**Note -** Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box. There are two default material swatches for surfaces because surfaces have both a front and a back side.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

### Cast Shadows

The Cast Shadows menu item allows the face of the entity to cast a shadow.

### Receive Shadows

The Receive Shadows menu item allows the face of the entity to receive a shadows cast by other entities.

# Text Entities

Text entities allow you to annotate your SketchUp model in several ways. There are two types of Text entities: Leader Text, which has leader lines and can be attached to a face, or Screen Text, which remains fixed to a point on your screen.



Text entities can have their own font, color, and size settings. Use the Text Tool to place Text Objects in your model.

## Attaching Text Directly to a Face

Double-click on a face with the Text Tool to attach text, without leader, to the face.

## Text Leaders

Text Objects can have one of four leader arrow styles: None, Dot, Closed arrow, and Open arrow. This can be changed using the context menu or its Entity Info dialog box.



Text leaders are tied to the model, so as you rotate the model, the text information continues to be valid. As you move and adjust surfaces, the notes attached to those surfaces adjust with them. Once a leader arrow is obscured the text will be hidden.

All text interacts with the model in three dimensions, but there are two ways for it to look on the screen. Thus, there are two main styles of leaders: View Based and Pushpin. A View Based leader will always retain its 2D screen orientation. A Pushpin leader is aligned in 3D space, and rotates with your model as you change your view.

### View Based

2D View Based leaders do not change as the model view is changed. Their appearance remains the same based on the screen layout and viewing direction present when they were placed. As you rotate the model, the actual text tries to stay oriented on the screen the same way, and the entire entity will follow whatever it is attached to. When the leader arrow becomes obscured, the whole text entity disappears. This method works well for presentation of still images from a particular vantage point.

## Pushpin

3D Pushpin leaders do change appearance as your view changes because they are drawn in 3D space just like model geometry. As you rotate the model, the leader foreshortens, rotates and hides just like any edge geometry. 3D text can be repositioned in 3D space just like any other edge entity. This method works well for planning studies and models that will be examined using fly-bys.

## Text Object Context Commands

Access context menu items by context clicking on an entity. The text context menu items follow:

### Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entity's attributes.

### Erase

The Erase menu item is used to delete the selected entity from your model.

### Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.

### Edit Text

This allows you to edit the text. Click outside the text editing window to accept the new text.

## Text Object Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities attributes. Following is a list of items in the entity's Entity Info dialog box.



***Note - Some of these items are only visible when the details arrow in the upper right-hand corner is pointing up.***

### Material

The Material swatch displays the material used on the currently selected entity. The default material for an entity is represented by a gray/blue box.

### Layer

The Layer field displays the layer of the currently selected entity. Select a new layer to move the entity to that layer.

### Arrow

The Arrow drop-down list contains a list of arrowheads permitted for the selected Text entity.

### Leader

The Leader drop-down list contains a list of leader-types permitted for the selected Text entity.

### Choose Font

The Choose Font button is used to select a new font for the selected Text entity.

### Text

The Text field contains the text string for the selected Text entity. Click in this field to modify the text. Click in the drawing area to save your changes.

### Hidden

The Hidden check box is used to hide or unhide the entity. Hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled.

# Multiple Entity Context Commands

The commands listed below are shared by most entities, and will generally be available when you bring up a context menu for multiple entities.



## Entity Info

The Entity Info menu item is used to display the Entity Info dialog box for viewing and changing the entities attributes. The Entity Info dialog box displays the number of entities selected.

## Erase

The Erase menu item is used to delete the selected entity from your model.

## Hide/Unhide

The Hide/Unhide menu item hides or unhides the select entity. All hidden entities are displayed in a ghosted pattern, allowing you to select them, if View > Show Hidden Geometry is enabled. This menu item changes to Unhide when selected geometry is hidden. Select Unhide to restore visibility for the entity.



## Select

The Select submenu contains several options for selecting geometry related to the entity.

**Bounding Edges:** The Bounding Edges menu item is used to select all of the edges currently bounding the selected entity.

**Connected Faces:** The Connected Faces menu item is used to select all of the faces currently connected to the selected entity.

**All Connected:** The All Connected menu item is used to select all of the elements currently connected to the selected entity.

**All on same Layer:** The All on same Layer menu item is used to select all of the elements on the current layer.

**All with same Material:** The All with same Material menu item is used to select all of the faces with the same material.

## Area

**Selection:** This calculates the total surface area of the face you clicked on.

**Layer:** This provides the total area of all faces in the layer of the face you clicked on.

**Materials:** This provides the total area of the faces in the entire model, hidden or unhidden, using the material of the face you clicked on.

## Make Component

This creates a Component entity from the selected geometry.

## Group

This creates a Group entity out of the selected geometry.

## Intersect with Model

This command allows you intersect two elements, such as a box and a tube, and automatically create new faces where the elements intersect. These faces can then be pushed, pulled or deleted to create new geometry.

## Reverse Faces

This flips the direction of each selected face so that they point in the opposite direction.

## Flip Along

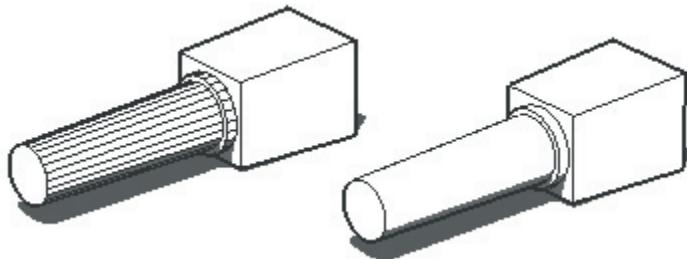
This mirrors the selected geometry along the axis you specify. You can reposition the Drawing Axes, allowing you to flip in virtually any direction.

### Zoom Extents

Zooms your view to a distance which makes the entities visible, and centers them in the drawing window.

### Soften

The Soften menu item is used to soften a edge. Softened edges are not visible unless displayed in profile.



### Soften/Smooth Edges

This option is available when the selection includes multiple edges and/or faces. It brings up the soften/smooth dialog, which lets you adjust the apparent rendered smoothness of geometry based on the angle you choose.

### Explode Curve

The Explode Curve command breaks the arc/circle/polygon entity into regular edges. An exploded entity will appear the same, but it can no longer be edited, and will no longer respond to tool operations as a single entity.

### Zoom Extents

Zooms your view to a distance which makes the whole entity visible, and centers it in the drawing window.

## Entity Info Dialog Box

Select an entity and either context click on the entity or select Window > Entity Info to view and change the entities' attributes. Following is a list of items in the entity's Entity Info dialog box.

### Material

The Material swatch displays the material used on the currently selected entities. The default material for an entity is represented by a grey/blue box.

### Layer

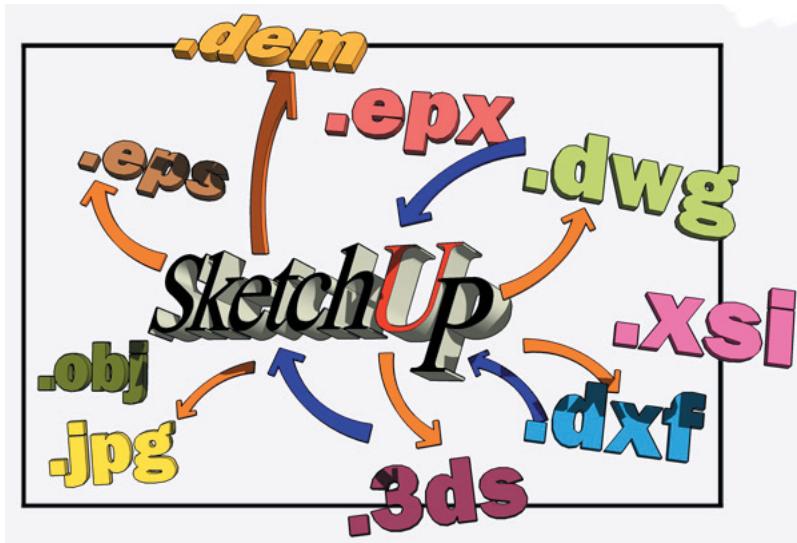
The Layer field displays the layer of the currently selected entities. Select a new layer to move the entities to that layer.



**Caution -** *Locked groups or components cannot be reassigned to a new layer. You will receive a warning message when you attempt to move locked groups or components to a new layer.*

# Input and Output

This section of the user's guide covers the various mechanisms for inputting items, such as graphic images, into your SketchUp file and outputting or presenting your model once it is finished.



The items in this section are divided into the following categories:

- Importers and Exporters
- Presentations (TourGuide)
- Printing

The majority of the items in this section are found in the File menu.

# 3D Model (DWG/DXF) Import

SketchUp allows you to import 3D models, such as DWG and DXF CAD files, for placement within your model. Import a file using the Import > 3D Model... menu item in the File menu.



**Note -** *@Last Software is a member of the OpenDWG Alliance, allowing SketchUp to offer the most reliable DWG file translation available.*

## Importing DWG/DXF Files

To import a 3D model file:

1. Select the File > Import menu item from the File menu. The Open dialog box is displayed.
2. Select the type of file to import from the Files of Type drop-down list.
3. (optional) Click on the Options... button to modify the import options, such as units, for the incoming file. See the following Import Options section for further information.
4. Click OK to import the file. The Import Results dialog box will appear containing details of the imported model.



**Note -** *It can take several minutes to import a large file because SketchUp's native geometry is very different from most CAD software and the conversion process is calculation-intensive.*

5. Click OK in the Import Results dialog box. The model will appear in the drawing area at the origin.



**Note -** *The imported model's entities will be enclosed within a group if geometry existed in the drawing area prior to importing the model.*

6. (optional) Click on the Zoom Extents Tool to locate the imported model if it is not currently displayed in your drawing area.

## Supported CAD Elements

SketchUp must translate the elements within a CAD file into SketchUp entities when importing a CAD file. Supported CAD elements include lines, arcs, circles, polylines, faces, entities with thickness, 3D faces, and nested blocks. CAD layers are also supported.

SketchUp does not support AutoCAD regions, XREFs, hatching, dimensions, text, and proprietary ADT or ARX objects. These CAD elements will be ignored on import.

You can change any unsupported elements into primitive CAD drawing elements, within your CAD application, if you need to import these elements into SketchUp. For example, you can explode ADT walls and extrusions in Autodesk Architectural Desktop so they will import as faces in SketchUp. Some elements might need to be exploded multiple times in your CAD package to be translated into SketchUp entities.

## Reducing Import File Size

Try to keep the size of imported files to a minimum. Importing very large CAD files can take a long time because each CAD element must be analyzed and converted into a SketchUp entity. Also, once imported, a complex CAD file can slow SketchUp's performance because lines and faces in SketchUp have a lot more "intelligence" than their CAD equivalents.

### Strategies for Minimizing CAD File Size

SketchUp models can be designed to be as accurate as models in CAD. However, SketchUp is not designed for the same type of line-intensive drawings done in CAD software. Therefore, it is a good idea to clean up and import only the CAD content that is absolutely necessary into SketchUp.

Another strategy is to keep different levels of detail in separate CAD files. For example, one imported CAD file can contain site plan information, another can have a floor plan, and a final file can have a specific detail.

## Import Options

Some CAD files might contain non-standard units, coplanar faces, or inconsistently oriented faces. You can force SketchUp to automatically analyze and repair these issues during the import process.

### Scale

Some CAD formats, such as DXF, save data in generic units.

**Units:** Select the unit type used in the original CAD file to import geometry at the correct scale.



**Caution -** SketchUp only recognizes faces of .001 square inches and larger. It is possible to have create very small faces on import if you select millimeters as your SketchUp units but the model was originally intended to be displayed in feet. Therefore, use a large unit type, such as feet or meters, if you do not know the units used in the original file. Resize the model as necessary after it is imported.

**Preserve Drawing Origin:** The Preserve drawing origin option is used to define where the DWG/DXF file will be placed when it is inserted into a SketchUp file. Check this box if you want to place the DWG/DXF file at the origin, as originally defined in the DWG/DXF file. Do not check this box if you want the DWG/DXF file to be placed near the SketchUp origin.

### Geometry

**Merge Coplanar Faces:** Planes can import into SketchUp with triangulated lines. Removing these lines manually can be tedious. The Merge coplaner faces option is used to have SketchUp automatically remove these lines.

**Orient Faces Consistently:** The Orient faces consistently option is used to analyze the direction of imported faces and orient these faces so that they are uniform in direction.

# 3D Model (3DS) Import

SketchUp allows you to import 3D Studio (3DS) files. Import a file using the Import > 3D Model... menu item in the File menu.

## Importing 3DS Files

To import a 3DS model file:

1. Select the File > Import menu item from the File menu. The Open dialog box is displayed.
2. Select the type of file to import from the Files of Type drop-down list.
3. (optional) Click on the Options... button to modify the import options for the incoming file. See the following Import Options section for further information.
4. Click OK to import the file.



**Note -** *It can take several minutes to import a large file because SketchUp's native geometry is very different from most CAD software and the conversion process is calculation-intensive.*

5. Click OK in the Import Results dialog box. The model will appear in the drawing area at the origin.
6. (optional) Click on the Zoom Extents Tool to locate the imported model if it is not currently displayed in your drawing area.

## Import Options

### Geometry

**Merge Coplanar Faces:** Planes can import into SketchUp with triangulated lines. Removing these lines manually can be tedious. The Merge coplanar faces option is used to have SketchUp automatically remove these lines.

### Scale

**Units:** The Units drop-down list is used to import 3DS geometry at the correct scale. A 3DS file has a field which indicates its units value. Sometimes this value is explicitly set in the 3DS file (such as millimeters) and sometimes it is missing and set to a generic value. The Model Units list item will import your 3DS file with the unit value it finds in your 3DS file. SketchUp will import your file using inches if a units value is not set in the 3DS file. SketchUp will import your 3DS file at the unit you choose if you pick anything other than Model Units.

For example, a face that is 1cm x 1cm will be imported into SketchUp as 1cm x 1cm if your 3DS file has its units value set to centimeters and you select Model Units in this drop-down list. A 1cm x 1cm face will import into SketchUp with dimensions of 1in x1in if you choose inches in the drop-down list.



**Caution** - SketchUp only recognizes faces of .001 square inches and larger. It is possible to have create very small faces on import if you select millimeters as your SketchUp units but the model was originally intended to be displayed in feet. Therefore, use a large unit type, such as feet or meters, if you do not know the units used in the original file. Resize the model as necessary after it is imported.

# 3D Model (DEM) Import

SketchUp allows you to import a digital elevation models (DEM) containing point data relating to terrain elevations. Unfortunately, there is not a single standardized file format for digital elevation models. However, SketchUp supports the following two important file formats: USGS DEM and spatial data transfer standard (SDTS).

The USGS DEM file format is an ASCII format for storing geographical information. These files can have an extension of .dem. You will have to add the .dem extension to your USGS DEM files if they do not have this extension.

SDTS is a complex geographical data format. SDTS models usually consist of 20 or more files, all with the extension of .ddf, stored in a single directory. You can select any of the \*.ddf files in a SDTS directory to import the entire set of files.

Import a USGS DEM or SDTS file using the Import > 3D Model... menu item in the File menu.

## Importing DEM File

To import a DEM file:

1. Select the File > Import menu item from the File menu. The Open dialog box is displayed.
2. Select the type of file to import from the Files of Type drop-down list.



**Caution -** You must add the .dem file extension to your DEM files to open these files. Additionally, you can select any file with the .ddf extension from your directory to open DDF files.

3. (optional) Click on the Options... button to modify the import options, such as points, for the incoming file. See the following Import Options section for further information.
4. Click OK to import the file. The Import Results dialog box will appear containing details of the imported model.



**Note -** The file import time corresponds to the number of maximum points defined in the Import Options dialog box: the greater number of maximum points, the longer it takes the file to load.

5. Click OK in the Import Results dialog box. The model will appear in the drawing area at the origin.



**Note -** The imported model's entities will be enclosed within a component.

6. (optional) Click on the Zoom Extents Tool to locate the imported model if it is not currently displayed in your drawing area.

## DEM Import Options

DEM files can contain thousands of points yielding a very detailed, yet very slow, SketchUp model. The CAD files might contain non-standard units, coplanar faces, or inconsistently oriented faces. You can force SketchUp to automatically analyze and repair these issues during the import process.

### TIN

**Points and Faces:** The Points and faces fields are used to identify the number of points, and subsequent faces, that will be created upon import. The fewer the points and subsequent faces, the better the performance in SketchUp. However, the fewer the points and subsequent faces, the lesser the detail of the imported model.



**Note -**  $(2 * \text{points}) + 1 = \text{approximate number of faces created upon import.}$

### Color

**Generate Gradient Texture:** The Generate gradient texture field is used to apply a gradient texture to the imported DEM model. This texture will be darker in lower areas of the model and lighter in high areas.

# 2D Graphic Import

SketchUp allows you to import 2D raster images, including JPEG and BMP, for placement within your model. Import a file using the Import menu item in the File menu.

## Importing 2D Graphic Files

To import a 2D graphic file:

1. Select the File > Import > menu item from the File menu. The Import 2D Graphic dialog box is displayed.
2. Select the type of file to import from the Files of Type drop-down list or select the All Image Files option to see a list of all your 2D raster images.
3. Locate the file you want to import.
4. (optional) Click the Use As Texture check box if you want your image to be used as a texture upon import (this will require you to paint the image on a an existing surface). See the following Importing 2D Graphics as Textures section for additional information.
5. Click OK to import the file. The image will appear in the drawing area. Your cursor, which will be an arrow, will be anchored to one of the image's corners.
6. Click anywhere in the drawing area to anchor the point of the image held by the cursor.
7. Drag your mouse away from the anchor point to increase the size of the image.
8. Click again to finish placing your image.

## Importing 2D Graphics as Textures

To import a 2D graphic file as a texture:

1. Select the File > Import menu item from the File menu. The Import 2D Graphic dialog box is displayed.
2. Select the type of file to import from the Files of Type drop-down list or select the All Image Files option to see a list of all your 2D raster images.
3. Locate the file you want to import.
4. Click the Use As Texture check box if you want your image to be used as a texture upon import (this will require you to paint the image on a an existing surface). See the following Importing 2D Graphics as Textures section for additional information.
5. Click OK to import the file. The image will appear in the drawing area. Your cursor, which will be the paint bucket, will be anchored to one of the image's corners. The paint bucket cursor only appears when the image is over a surface where it can be painted.
6. Click on a surface to anchor the point of the image held by the cursor.
7. Drag your mouse away from the anchor point to increase the size of the image on the face.
8. Click again to paint your image on the face. The image will repeat, if necessary, to fill the entire face. A thumbnail of the image appears in the Materials Browser's In Model section to indicate that the image is a texture. Refer to the Paint Bucket Tool for additional information on working with textures.

# 3D Model (DWG/DXF) Export

SketchUp can export 3D geometry to several AutoCAD formats: DWG r12, DWG r13, DWG r14, DWG r2000, DWG r2004, DXF r12, DXF r13, DXF r14, DXF r2000, and DXF r2004.

SketchUp uses the industry standard OpenDWG Alliance file import/export model libraries to ensure maximum possible compatibility with AutoCAD. Export a file from the File menu.

## Exporting 3D DWG/DXF Files

To export a 3D DWG or DXF file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File name field.
3. Select the export type from the Export type drop-down list.
4. (optional) Click on the Options button. The AutoCAD Export Options dialog box is displayed.
5. (optional) Adjust the options in the AutoCAD Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## AutoCAD Export Options Dialog Box

The AutoCAD Export Options dialog box contains options for exporting AutoCAD files.

### AutoCAD Version

The AutoCAD section of the AutoCAD DWG/DXF Export Options Dialog Box is used to select the version of exported DWG/DXF file.

### Export

The Export section of the AutoCAD Export Options dialog box contains the various entities that can be exported into your DWG/DXF file.

SketchUp can export faces, edges (wires), dimensions, text, or construction geometry in any combination.

## 3D DWG/DXF Export Details

The 3D DWG and DXF export is used to export SketchUp files for use in other CAD programs such as AutoCAD. The following details apply to this export:

- Faces in SketchUp are exported as a triangulated polyface mesh with interior splframe hidden lines (if applicable). This conversion helps to simulate the appearance of your native SketchUp file, even when all exported faces are triangular.

- SketchUp uses the current units set in the Units Panel of the Model Info dialog box as a reference for translation to a DWG or DXF file. For example, if the current unit setting is Decimal and Meters, SketchUp will export the DWG file accordingly. AutoCAD must be set to decimal for the units to translate correctly as meters.
- When exporting, duplicate line entities will not be created on top of a p-line entity.

# 3D Model (3DS) Export

The 3DS format is native to the original DOS-based 3D Studio modeling and animation application. Although obsolete in many ways, the 3DS format is still widely used and offers a direct way to export simpler SketchUp models into a wide range of 3D modeling packages. Because 3DS preserves material assignments, texture mapping, and camera position, it can often allow ideas generated in SketchUp to be transferred with greater fidelity than formats designed for CAD.

## Exporting 3DS Files

To export a 3DS file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File name field.
3. Select the 3DS export type from the Export type drop-down list.
4. (optional) Click on the Options button. The 3DS Export Options dialog box is displayed.
5. (optional) Adjust the options in the 3DS Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## 3DS Export Options Dialog Box

The 3DS Export Options dialog box contains options for exporting 3DS files.

### Geometry

**Export (Single Object):** The Export (Single object) item is used to export your SketchUp model as a single 3DS mesh. This single mesh will be simple to select and manipulate once imported into another application. Because there is a limitation in the 3DS format that meshes can only have 65,536 vertices and faces, if this limit is exceeded, more than one mesh will be created with emphasis on keeping the number of meshes to the minimum amount necessary.

**Export (Objects By Geometry):** The Export (Objects by geometry) item is used to analyze and break up your SketchUp model into 3DS objects based on the connections of faces, groups, and component definitions. All top level components and group instances will be exported as separate meshes. For example, if you have 5 top level components and 3 top level groups, 8 meshes will be created. For geometry that is not contained within components/groups, meshes will be created based on the connection of faces – those entities that are connected will be exported as separate meshes. Note that the hierarchy of components/groups is not respected with this option. As with the Single object option, if any of the meshes exceed the 65,536 vertices/face limit, additional meshes will be created as necessary.

**Export (Objects In Hierarchy):** The Export (Objects in hierarchy) item is used to export the complete component and group hierarchy of the SketchUp model. The faces of each component and group are exported as a single mesh, excluding any nested components or groups which are exported as individual meshes. This option preserves the name of the component or group using the following rule for components: if an instance name exists, that

name will be used. If an instance name does not exist, the definition name will be used. For example, if you have a component instance named "MyOffice" which contains twenty faces (connected or not) and a nested component instance named "MyChair", the twenty faces would be exported as a single mesh under the name "MyOffice" and the faces of the chair would be exported as a single mesh under the name "MyChair." Top-level geometry that is not contained within components or groups is exported as a single mesh. If any of the meshes exceed the 65,000 vertices/face limit, additional meshes will be created as necessary (as with the Single object option).

**Export Only Current Selection:** The Export only current selection checkbox is used to export the currently selected geometry. The entire SketchUp model is exported when there is no selection. Additionally, the entire model is exported when this checkbox is not checked.

**Export Two-Sided Faces:** The Export two-sided faces checkbox is used to export faces twice, once for the front and once for the back. This option doubles the number of polygons in the resulting 3DS file and can slow down rendering. However, this option ensures that your model will appear more like it appears in SketchUp. Both faces will always render, and materials applied to front and back faces are preserved. When this option is selected, SketchUp will weld the vertices of the front faces together and the vertices of the back faces together according to the logic for texture mapping and welding outlined in the following Export texture maps option.

**Export Standalone Edges:** Standalone Line entities (lines not connected to any faces) are somewhat unique to SketchUp and not supported by many other 3D programs or the 3DS format. The Export standalone edges check box creates very thin rectangles that appear as standalone lines in the 3DS files. Unfortunately, this option can result in invalid texture coordinates requiring UV mapping to be re-applied before the scene is rendered. Also, certain exported standalone edge geometries can create invalid 3DS files altogether. The VRML format may better suit your needs if you need to export standalone edges.

## Materials

**Export Texture Maps:** The Export texture maps checkbox is used to assign texture maps to 3DS materials whenever the corresponding SketchUp material uses a texture image. The 3DS format allows only one UV mapping coordinate per vertex, meaning you can not have a different UV mapping on two faces that share the same vertex. Given this limitation of the 3DS format, a tradeoff between preserving texture coordinates and welding geometry is sometimes necessary and reflected in the following two sub-options: Favor Preserving texture coordinates and Favor Welding Vertices.

**Favor (Preserving Texture Coordinates):** When the Preserving texture coordinates option is selected, and UV mappings are different, the exporter breaks up geometry so that each coplanar group of polygons has its own set of vertices. In this case, vertices will not be welded together and no smoothing of faces will be evident. If UV mappings are identical, vertices will be welded.



**Note - Use this option if the preservation of textures is of most importance.**

**Favor (Welding Vertices):** When the Welding Vertices option is selected, the exporter will weld together vertices which will keep your geometry intact and allow smoothing to be preserved. However, when two faces share the same vertex but have different UV mappings, this option will only allow one of the texture mappings to be preserved.



**Note - Use this option if the preservation of smoothing and welding is of most importance.**

**Use "Color by Layer" Materials:** The 3DS file format does not support layers directly. The Use "Color by Layer" Materials checkbox is used to assign 3DS materials based on your model's layer assignments in SketchUp. Check this checkbox to export materials based on layers, such as when exporting to other rendering applications.

## Cameras

**Generate Cameras From Pages:** The Generate Cameras from Pages checkbox is used to create a camera for the default view as well as any SketchUp Pages which have been defined. The current SketchUp view is exported with the name "Default Camera," and other Page camera definitions are output using their Page Name.

## Scale

**Units:** The units drop-down list is used to determine the unit size in the 3DS output.

## Known Issues and Limitations

### 2 Sided Faces

Faces are only visible from the front in some programs while both sides of a face are always visible in SketchUp. These two-sided faces can appear to be missing in other applications if not facing the correct direction. You can manually reorient your faces in SketchUp using the Reverse command or the Orient Faces command to make face directions uniform for a given set of connected geometry.

Additionally, you can use the Export two-sided checkbox to correct this issue.

### Flipped Textures

Textures and materials exported with the two-sided/geometry checkbox can appear flipped in certain applications (such as, Maya version 6.0 and 6.5). A 3DS mesh does not store any normal data so normals must be computed by the target system. Some importers might compute the normals incorrectly resulting in the appearance of flipped textures.

### Texture Map Names

The 3DS format was designed in an era when DOS-based systems were prevalent. Therefore, 3DS is not capable of storing texture filenames that exceed the 8.3 DOS character limit. This issue can be a limitation to anyone who uses modern OS filename capabilities to manage large, complex projects or file libraries.

SketchUp creates unique names for each texture by adding unique suffixes to file names. For example, a file named "corrugated metal.jpg" will be described in the 3DS file as "corrugat.jpg". Any other files that use the same first six letters are truncated and the suffix "01," "02," and so on, are added.

## Visibility

Only entities that are currently visible in SketchUp are output to the 3DS file. Rendering display options are not taken into account, which means that faces are output even if SketchUp is in wireframe mode. Faces are not output, however, if they are hidden or if they reside on a layer that is hidden.

## Multiple Cameras

3DS format can not store an orthographic camera. This format will be simulated through a perspective camera with a very small field of view or a very large lens length. The eye point will be moved as far away as is possible to yield the same width and height of the projection plane. Certain applications might have problems with a small field of view. For example, Maya does not read the field of view at all and only lets it become as small as 2.5.

## Layers

The 3DS format does not support layers. Any layers you assign in SketchUp will be lost. The DWG format might be a better solution if you need layers to export. Alternately, you can export with the "Color by Layer materials" option enabled, which might allow you to easily select and organize geometry in your model based on SketchUp layers once your model is imported into another application.

## Units

SketchUp exports 3DS files at the units specified in the 3DS export options dialog box. This setting can affect the way geometry is described within the 3DS file. For example, a 1 meter cube in SketchUp will export to 3DS with sides of length 1 when units are set to meters. If you change the export units to centimeters, the same cube will export to 3DS with a length of 100.

The 3DS format contains extra information that indicates the original units using a scale factor. This information allows an application that reads 3DS to automatically adjust the size of 3DS models to the correct "real world" size.

Unfortunately, many applications ignore this extra unit scale information meaning that the centimeter cube imports as 100 times larger than the 1 meter cube, instead of at the same size. Worse, it is not always clear in which unit 3DS files are saved, resulting in considerable trial and error. In these cases, the best work-around is simply to export files at the units setting expected by the 3DS importing application.

# 3D Model (VRML) Export

VRML 2.0 (Virtual Reality Modeling Language) is a 3D scene/object description format often used to exchange data between 3D applications and to publish 3D information to the web. VRML files can store SketchUp geometry in the form of edges, faces, groups, materials and textures, transparency, camera views, and lights.

## Exporting VRML Files

To export a VRML file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File name field.
3. Select the VRML export type from the Export type drop-down list.
4. (optional) Click on the Options button. The VRML Export Options dialog box is displayed.
5. (optional) Adjust the options in the VRML Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## VRML Export Options Dialog Box

The VRML Export Options dialog box contains options for exporting VRML files.

### Appearance

**Output Texture Maps:** If enabled, SketchUp will export texture information to the VRML file. When disabled, it will export colors only. When publishing VRML files to the web, you may want to edit the files so that textures read from the relative location rather than from your local hard drive. Also, VRML texture and material names cannot have blanks, so SketchUp will use the underscore character instead.

**Ignore Back of Face Materials:** The Ignore Back of Face Materials checkbox is used to export both front and back faces with the front material. SketchUp exports VRML files with double faces so that files will be viewable from any viewpoint.

**Output Edges:** The Output Edges check box exports displayed edges as VRML edge entities.

**Use "Color by Layer" Materials:** The VRML file format does not support layers directly. The Use "Color by Layer" Materials checkbox is used to assign VRML materials based on your model's layer assignments in SketchUp. Check this checkbox to export materials based on layers, such as when exporting to other rendering applications.

**Use VRML Standard Orientation:** The Use VRML Standard Orientation checkbox is used to export a file that conforms to the VRML standard. The VRML standard considers the XZ plane to be horizontal (the ground plane) whereas SketchUp considers the XY plane to be the ground plane.

**Generate Cameras:** The Generate Cameras checkbox is used to create a camera for the default view as well as any SketchUp Pages which have been defined. The current SketchUp view is exported with the name "Default Camera", and other Page camera definitions are output using their Page Name.

**Allow Mirrored Components:** The Allow Mirrored Components check box is used to export components that have been mirrored or resized so they are an opposite of the original component.

**Check for Material Overrides:** The Check for Material Overrides check box is used to determine if any of the faces, edges, or components within this component contain references to the default material or default layer.

# 3D Model (FBX) Export

SketchUp allows you to export your models as Kaydara (FBX) files. Export a FBX file using the Export > 3D Model... menu item in the File menu.

## Exporting a FBX File

To export a FBX model file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File name field.
3. Select the FBX export type from the Export type drop-down list.
4. (optional) Click on the Options button. The FBX Export Options dialog box is displayed.
5. (optional) Adjust the options in the FBX Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## The FBX Export Options Dialog Box

The XSI Export Options dialog box contains options for exporting XSI Files.

### Geometry

The Geometry section of the FBX Export Options Dialog box is used to identify the geometry that will be exported.

**Selection only:** The Selection only checkbox is used to export the currently selected geometry. The entire SketchUp model is exported when there is no selection. Additionally, the entire model is exported when this checkbox is not checked.

**Triangles only:** The Triangles only checkbox is used to break the output into triangles instead of multi-sided faces.

**Swap YX coordinates (Y is up):** The Swap YZ coordinates check box is used to output the model with the vertical direction as Y. This orientation is the default orientation for some applications, such as Maya. Do not check this box if you want your model to be output with the Z axis in the vertical direction.

**Export faces double-sided:** The Export faces double-sided checkbox is used to export faces twice, once for the front and once for the back. This option doubles the number of polygons in the resulting XSI file and can slow down rendering. However, this option ensures that your model will appear more like it appears in SketchUp. Both faces will always render, and materials applied to front and back faces are preserved.

**Units:** The units drop-down list is used to determine the unit size in the OBJ output.

# 3D Model (XSI) Export

SketchUp allows you to export your models as Soft Image (XSI) files. Export a XSI file using the Export > 3D Model... menu item in the File menu.

## Exporting a XSI File

To export a XSI model file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File namefield.
3. Select the XSI export type from the Export type drop-down list.
4. (optional) Click on the Options button. The XSI Export Options dialog box is displayed.
5. (optional) Adjust the options in the XSI Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## The XSI Export Options Dialog Box

The XSI Export Options dialog box contains options for exporting XSI files.

### Geometry

The Geometry section of the XSI Export Options Dialog box is used to identify the geometry that will be exported.

**Selection only:** The Selection only checkbox is used to export the currently selected geometry. The entire SketchUp model is exported when there is no selection. Additionally, the entire model is exported when this checkbox is not checked.

**Triangles only:** The Triangles only checkbox is used to break the output into triangles instead of multi-sided faces.

**Swap YX coordinates (Y is up):** The Swap YZ coordinates check box is used to output the model with the vertical direction as Y. This orientation is the default orientation for some applications, such as Maya. Do not check this box if you want your model to be output with the Z axis in the vertical direction.

**Export edges:** The Export edges checkbox is used to write SketchUp line entities as XSI line entities. Edges are ignored when this checkbox is not checked. Usually this checkbox is not checked because most applications will ignore edges on import.

**Export faces double-sided:** The Export faces double-sided checkbox is used to export faces twice, once for the front and once for the back. This option doubles the number of polygons in the resulting XSI file and can slow down rendering. However, this option ensures that your model will appear more like it appears in SketchUp. Both faces will always render, and materials applied to front and back faces are preserved.

**Units:** The units drop-dow list is used to determine the unit size in the OBJ output.

SketchUp

# 2D Graphic Export

SketchUp allows you to export 2D raster image files in JPG, BMP, TGA, TIFF, and PNG raster file formats. Export a file using the Export > 2D Graphic... menu item in the File menu.



**Note - Shadows cannot be saved with a 3D model. None of the 2D vector output formats support rendering features such as shadows, textures, or transparency. The only export formats that will represent shadows as seen on your display are pixel-based raster images and animations.**

## Exporting a Raster Image

To export a 2D graphic file:

1. Manipulate the view of the model so that it appears as you would like it to appear in the exported file. SketchUp will export the model view as it is currently displayed, including display style, edge rendering, shadows, and point of view.
2. Select File > Export > 2D Graphic. The Export 2D Graphic dialog box is displayed.
3. Enter a file name for the exported file in the File name field.
4. Select the export type from the Export type drop-down list.
5. (optional) Click on the Options button. The Export Image Options dialog box is displayed.
6. (optional) Adjust the options in the Export Image Options dialog box.
7. (optional) Click the OK button.
8. Click the Export button.

## Export Image Options Dialog Box

The Export Image Options dialog box contains options for exporting raster image files.

### Image Size

**Use View Size:** The Use view size is used to set your output to a 1:1 (real world) scale.

**Width/Height:** The Width and Height fields are used to control the size of image as measured in pixels or specific units.

### Rendering

**Anti-Alias:** When enabled, SketchUp will smooth the exported image. This method can take longer, but it helps to reduce jagged lines and pixilated artifacts in images.

## JPEG Compression

The slider bar for a JPEG export is used to define the level of JPEG compression quality. The level of quality becomes lesser and the file size is smaller as the slider is moved to the left. Conversely, the level of quality becomes higher and the file size is larger if the slider is moved to the right.

# 2D Graphic (DWG/DXF) Export

SketchUp allows you to export your models as 2D vector drawings, including DWG and DXF. Export a file using the Export > 2D Graphic... menu item in the File menu.

## Exporting a DWG or DXF File

To export a 2D DWG or DXF graphic file:

1. Manipulate the view of the model so that it appears as you would like it to appear in the exported file.



**Note -** SketchUp exports the model using the current point of view with the exception of unsupported features such as textures and shadows.

2. Select File > Export > 2D Graphic. The Export 2D Graphic dialog box is displayed.
3. Enter a file name for the exported file in the File name field.
4. Select the DWG or DXF export type from the Export type drop-down list.
5. (optional) Click on the Options button. The DWG/DXF Hidden Line Options dialog box is displayed.
6. (optional) Adjust the options in the DWG/DXF Hidden Line Options dialog box.
7. (optional) Click the OK button.
8. Click the Export button.

## DWG/DXF Hidden Line Options Dialog Box

The DWG/DXF Hidden Line Options dialog box contains options for exporting 2D DWG and DXF Files.

### Drawing Scale & Size

The Drawing Scale & Size section of the DWG/DXF Hidden Options Dialog box contains scaling options for your model.

**Full Scale (1:1):** The Full Scale (1:1) option is used to set your output to a 1:1 (real world) scale.

**In Drawing/In Model:** The In Drawing/In Model fields are used to scale your model for export. The first measurement, labeled In the Model, is the measurement of the exported geometry. The second measurement, labeled In Drawing , is the actual measurement of the object in real scale. For example, for a scale of  $1/4" = 1'$ , simply enter 1 inch In Model equals 4 feet In Drawing.



**Note -** It is not possible to print a perspective image to scale. Ensure perspective is disabled and you are using one of the Standard Views in the Camera > Standard submenu, to enable this scale option.

**Width/Height:** The width and height fields are used to enter a custom page size for your file.

## AutoCAD Version

The AutoCAD section of the AutoCAD DWG/DXF Export Options Dialog Box is used to select the version of exported DWG/DXF file.

## Profile Lines

The Profile Lines section of the DWG/DXF Hidden Options Dialog box contains options for exporting profile lines.

**None:** The None option is used to export profile lines at standard width.

**Polylines with width:** The Polyline with width option is used to export profile lines as AutoCAD polyline entities.

**Wide line entities:** The Wide line entities option is used to export profile lines as AutoCAD wide line entities. This option is only available when saving AutoCAD 2000 or later DWG files.

**Separate on a layer:** The Separate on a layer option is used to output profile lines onto their own layer. This option is useful if you would like to plot profile lines using a different pen weight or quickly change the line width of all profile lines in a separate CAD or Illustration program.



**Note -** The Separate on a layer option creates a single additional layer for profile edges. SketchUp Layer assignments do not translate directly when exporting 2D hidden line vectors.

**Width:** The Width field is used to specify an exact width for the Polyline with width option. This option is only available when Polyline with width is checked and Automatic is unchecked.

**Automatic:** The Automatic option is used to set the width of profile lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Polyline with width is checked.

## Section Lines

The Section Lines section of the DWG/DXF Hidden Options Dialog box contains options for exporting section lines.

**None:** The none option is used to export section slice lines at standard width.

**Polyline with width:** The Polyline with width option is used to export section slice lines as AutoCAD polyline entities.

**Wide line entities:** The Wide line entities option is used to export section slice lines as AutoCAD wide line entities. This option is only available when saving AutoCAD 2000 or later DWG files.

**Separate on a layer:** The Separate on a layer option is used to output section slice lines onto their own layer. This option is useful if you would like to plot section slice lines using a different pen weight or quickly change the line width of all section slice lines in a separate CAD or Illustration program.



**Note** - The Separate on a layer option creates a single additional layer for section slice edges. SketchUp Layer assignments do not translate directly when exporting 2D hidden line vectors.

**Width:** The Width field is used to specify an exact width for the Polyline with width option. This option is only available when Polyline with width is checked and Automatic is unchecked.

**Automatic:** The Automatic option is used to set the width of section slice lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Polyline with width is checked.

## Extension Lines

The Extension Lines section of the DWG/DXF Hidden Options Dialog box contains options for exporting extension lines.

**Show extensions:** Some CAD applications might have problems recognizing endpoints and intersections of lines with SketchUp extensions. The Show extensions option is used to toggle the export of line extensions.

**Length:** The Length field is used to specify an exact length for line extensions. This option is only available when Show extensions is checked and Automatic is unchecked.

**Automatic:** The Automatic option is used to set the length of extension lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Show extensions is checked.

**Always Prompt for Hidden Line Options:** The Always Prompt for Hidden Line Options is used to automatically display the Hidden Line Options dialog box when you export a 2D DWG or DXF file.

**Defaults:** The Defaults button is used to return the items in the Hidden Line Options dialog box to the default settings.

# 2D Graphic (PDF/EPS) Export

SketchUp allows you to export your models as 2D vector drawings, including PDF and EPS. Export a file using the Export > 2D Graphic... menu item in the File menu.

## Exporting a PDF or EPS File

To export a 2D PDF or EPS graphic file:

1. Manipulate the view of the model so that it appears as you would like it to appear in the exported file.



**Note -** SketchUp exports the model using the current point of view with the exception of unsupported features such as textures and shadows.

2. Select File > Export > 2D Graphic. The Export 2D Graphic dialog box is displayed.
3. Enter a file name for the exported file in the File name field.
4. Select the PDF or EPS export type from the Export type drop-down list.
5. (optional) Click on the Options button. The PDF/EPS Hidden Options dialog box is displayed.
6. (optional) Adjust the options in the PDF/EPS Hidden Options dialog box dialog box.
7. (optional) Click the OK button.
8. Click the Export button.

## PDF/EPS Hidden Line Options Dialog Box

The PDF/EPS Hidden Options dialog box contains options for exporting 2D PDF and EPS Files.

### Drawing Size

The Drawing Size section of the PDF/EPS Hidden Options Dialog box contains size and scaling options for your model.

**Full Scale (1:1):** The Full Scale (1:1) option is used to set your output to a 1:1 (real world) scale.

**Width/Height:** The width and height fields are used to enter a custom page size for your file.

**In Hidden-Line Output/In Sketchup:** The In Hidden-Line Output/In SketchUp fields are used to scale your model for export. The first measurement, labeled In Hidden-Line Output, is the measurement of the exported geometry. The second measurement, labeled In SketchUp, is the actual measurement of the object in real scale. For example, for a scale of 1/4" = 1', simply enter 1 inch In SketchUp equals 4 feet In Hidden Line Output.



**Note -** It is not possible to print a perspective image to scale. Ensure perspective is disabled and you are using one of the Standard Views in the Camera > Standard submenu, to enable this scale option.

## Profile Lines

The Profile Lines section of the PDF/EPS Hidden Options Dialog box contains options for exporting profile lines.

**Show Profiles:** The Show profiles option exports any lines that are displayed in profile as thicker lines in the 2D vector file. All lines are output normally, without profile thickness, when this option is disabled (regardless of the screen display).

**Match Screen Display (Auto Width):** The Match screen display (auto width) automatically sets the width of profile lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Show profiles is checked.

**Width:** The Width fields are used to specify an exact width for the profile lines. This option is only available when Show Profiles is checked and Match Screen Display (Auto Width) is unchecked.

## Section Lines

The Section Lines section of the DWG/DXF Hidden Options Dialog box contains options for exporting section lines.

**Specify Section Line Width:** The Specify section line width option is used to adjust settings for Section Slice lines that are output.

**Match Screen Display (Auto Width):** The Match screen display (auto width) automatically sets the width of section lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Specify section line width is checked.

**Width:** The Width fields are used to specify an exact width for the section lines. This option is only available when Specify Section Line Width is checked and Match screen display (auto width) is unchecked.

## Extension Lines

The Extension Lines section of the PDF/EPS Hidden Options Dialog box contains options for exporting extension lines.

**Extend Edges:** Some CAD applications might have problems recognizing endpoints and intersections of lines with SketchUp extension lines. The Extend edges option is used to toggle the export of line extensions.

**Match Screen Display (Auto Width):** The Match screen display (auto width) automatically sets the width of extension lines by matching the output to the proportions you see in the SketchUp Drawing Area. This option is only available when Extend edges is checked.

**Width:** The Width fields are used to specify an exact width for the extension lines. This option is only available when Extend Edges is checked and Match screen display (auto width) is unchecked.

**Always Prompt for Hidden Line Options:** The Always prompt for hidden line options is used to automatically display the Hidden Line Options dialog box when you export a 2D PDF or EPS file.

**Map Windows fonts to PDF base fonts:** This option is used to select PDF fonts that correspond to the Windows fonts used in the model.

**Defaults:** The Defaults button is used to return the items in the Hidden Line Options dialog box to the default settings.

## 2D PDF/EPS Export Details

The PDF and EPS export is used to export vector SketchUp files for use in other vector-based editing programs like Adobe Illustrator.



**Caution -** Some graphic features of SketchUp, including textures, shadows, smooth shading, backgrounds, and transparency, cannot be exported to PDF and EPS.

### Text and Dimensions

SketchUp will attempt to output text annotation and notes to your 2D drawing. Please note the following limitations:

- Text and Dimension entities that are obscured (partially or totally) by other geometry in the SketchUp Drawing Area export on top of the geometry.
- Text and Dimension entities that are partially clipped by the edges of the SketchUp Drawing Area are not exported.
- Certain fonts may not export exactly.

# Piranesi Epix Export

Piranesi is a painting application that enables you to create stunning renderings from your SketchUp models. For more information, please visit the Piranesi Web Site: [www.informatix.co.uk/piranesi.htm](http://www.informatix.co.uk/piranesi.htm)

SketchUp allows you to export your models as Epix files for use in Piranesi. Export an Epix file using the Export > 2D Graphic... menu item in the File menu.

## Exporting an Epix File

To export a Epix file:

1. Manipulate the view of the model so that it appears as you would like it to appear in the exported file.



**Note -** SketchUp exports the model using the current point of view with the exception of unsupported features such as textures and shadows.

2. Select File > Export > 2D Graphic. The Export 2D Graphic dialog box is displayed.
3. Enter a file name for the exported file in the File name field.
4. Select the Epix export type from the Export type drop-down list.
5. (optional) Click on the Options button. The Export Epx Options dialog box is displayed.
6. (optional) Adjust the options in the Export Epx Options dialog box dialog box.
7. (optional) Click the OK button.
8. Click the Export button.



**Note -** Your display must be set to 32-bit color in order to correctly export Epix files.

## The Export Epix Options dialog box

The Export Epix Options dialog box contains options for exporting Epix files.

### Image Size

The Image Size section of the Export Epix Options dialog box contains size and scaling options for your model.

**Use View Size:** The Use View Size option is used to set your output to the exact dimensions of your SketchUp drawing area.

**Width/Height:** The width and height fields are used to enter a custom page size for your file.



**Tip -** SketchUp cannot export Epix files with compression enabled. Re-saving your file in Piranesi can reduce file size considerably.

## EPIX

The EPIX section of the Export Epix Options dialog box contains options for the contents of the exported file.

**Export Edges:** The Export Edges check box is used to maintain the edge rendering style from the drawing area when saving your drawings as an Epix file.



**Note -** *Edges will not appear in your Epix file if edges are disabled using Window > Display Settings.*

**Export Textures:** The Export Textures check box is used to maintain textures from the drawing area when saving your drawings as an Epix file.



**Note -** *The Export Textures option is only applicable when materials have been assigned to faces and SketchUp is in the Shaded with Textures display setting.*

**Export Ground Plane:** The Export Ground Plane check box is used to create a ground plane in the Epix file without explicitly modeling a ground in SketchUp.

# 3D Model (OBJ) Export

The OBJ file format is a 3D file format created by Wavefront for use in their Advanced Visualizer™ product. These files are text-based and support free-form and polygonal geometry. An additional .mtl file describes materials defined in the .obj file.

## Exporting OBJ Files

To export an OBJ file:

1. Select File > Export > 3D Model. The Export Model dialog box is displayed.
2. Enter a file name for the exported file in the File name field.
3. Select the OBJ export type from the Export type drop-down list.
4. (optional) Click on the Options button. The OBJ Export Options dialog box is displayed.
5. (optional) Adjust the options in the OBJ Export Options dialog box.
6. (optional) Click the OK button.
7. Click the Export button.

## OBJ Export Options Dialog Box

The OBJ Export Options dialog box contains options for exporting OBJ files.

### Geometry

**Export only current selection:** The Export only current selection checkbox is used to export the currently selected geometry. The entire SketchUp model is exported when there is no selection. Additionally, the entire model is exported when this checkbox is not checked.

**Triangulate all faces:** The Triangulate all faces checkbox is used to break the output into triangles instead of multi-sided faces.

**Export two-sided faces:** The Export two-sided faces checkbox is used to export faces twice, once for the front and once for the back. This option doubles the number of polygons in the resulting OBJ file and can slow down rendering. However, this option ensures that your model will appear more like it appears in SketchUp. Both faces will always render, and materials applied to front and back faces are preserved.

**Export edges:** The Export edges checkbox is used to write SketchUp line entities as OBJ line entities. Edges are ignored when this checkbox is not checked. Usually this checkbox is not checked because most applications will ignore edges on import.

## Material Properties

**Swap YZ coordinates (Y is up):** The Swap YZ coordinates check box is used to output the model with the vertical direction as Y. This orientation is the default orientation for some applications, such as Maya. Do not check this box if you want your model to be output with the Z axis in the vertical direction.

**Units:** The units drop-down list is used to determine the unit size in the OBJ output.

## Known Issues and Limitations

### OBJ file names

OBJ files do not support spaces in their filenames. Sketchup will replace any space with an underscore (\_) when the file is saved.

### Triangulation errors

SketchUp will export one face entity as one polygon in an OBJ file. This ratio of faces to polygons can create some problems for some OBJ importers (polygons might appear missing or reversed). Use the Triangulate faces option to create additional polygons for importers that seem to have problems with the OBJ file exported from SketchUp. Additionally, you can import the OBJ file into an application that does not have problems with the SketchUp faces to polygons ratio and then export that file as another file type to be used in the less-qualified application.

### Flat set hierarchy

The OBJ output supports a flat set membership hierarchy meaning that the format identifies which objects belong to any set. This output does not support a tree hierarchy because it cannot identify if one particular set is actually a component of another set. This limitation is a limitation of the OBJ format.

### NURBS

SketchUp does not output nurbs or any of the advanced OBJ entities. SketchUp outputs polygon faces only.

# Section Slice Export

SketchUp can save the active Section Slice as its own 2D file in the DWG and DXF formats. As with all vector information, section slice export has the advantage of precision. Export an animation using the Export > Section Slice... menu item in the File Menu.

## Exporting Section Slices

To export a section slice:

1. Double-click on the Section Plane entity whose section slice you want to export.
2. Select File > Export > Section Slice. The Export 2D Section Slice dialog box is displayed.
3. Enter a file name for the exported file in the File name field.
4. Select the export type from the Export type drop-down list.
5. (optional) Click on the Options button. The Section Slice Export Options dialog box is displayed.
6. (optional) Adjust the options in the Section Slice Export Options dialog box.
7. (optional) Click the OK button.
8. Click the Export button.

## 2D Section Slice Options Dialog Box

The 2D Section Slice Export Options dialog box contains options for exporting 2D section slice files.

### Projection

The projection section contains options for selecting the type of exported file.

**True Section (Orthographic):** This option outputs the section slice as a true orthographic drawing. This is useful for creating templates for CD drawings or any other time you want to generate accurate, measurable slices.

**Screen Projection (WYSIWG):** This projects the section cut as you see it on your SketchUp screen, including any perspective distortion. This is most useful for diagrams that need not be measured.

### Drawing Scale & Size

**Full Scale (1:1):** When enabled, This outputs your section slice to CAD at a true 1:1 scale.

**Width/Height:** These values represent the overall dimensions of the slice output, and will update depending on the slice being exported and its scale.

**In Drawing/In Model:** These controls allow you to specify a certain drawing scale, and follow architectural conventions. The first measurement, labeled 'In Drawing' is the measurement of the exported geometry. The second measurement, labeled 'In Model' is the actual measurement of the object in real scale.

For example, for a scale of  $1/4" = 1'$ , simply enter 1 inch in the output equals 4 feet in SketchUp.



**Note -** You cannot export a perspective screen projection to scale. Also, even if the view is a parallel projection, only faces whose normals are perpendicular to the view angle will be measurable.

## AutoCAD Version

The AutoCAD section of the AutoCAD DWG/DXF Export Options Dialog Box is used to select the version of exported DWG/DXF file.

### Section Lines

**None:** Outputs section slice lines at normal width.

**Polylines with width:** Outputs lines as poly-line entities.

**Wide line entities:** Outputs lines as wide line entities. This option is only available when saving AutoCAD 2000 or later DWG files.

**Separate on a layer:** The Separate on a layer option is used to output profile lines onto their own layer. This option is useful if you would like to plot profile lines using a different pen weight or quickly change the line width of all profile lines in a separate CAD or Illustration program.



**Note -** The Separate on a layer option creates a single additional layer for profile edges. SketchUp Layer assignments do not translate directly when exporting 2D hidden line vectors.

**Width/Automatic:** When enabled, this automatically sets the width of profile lines by matching the output to the proportions you see in the SketchUp display. If disabled, you can specify an exact width.

**Always Prompt for Hidden Line Options:** When enabled, the options dialog will come up each time you output a section slice. When disabled, SketchUp will use whatever options were used the last time by default.

# Animation Export

You can export TourGuide presentations as video animation files (Audio Video Interleave or AVI files) or as series of image file, one file for each frame per second of your presentation. Export an animation using the Export > Animation... menu item in the File Menu.



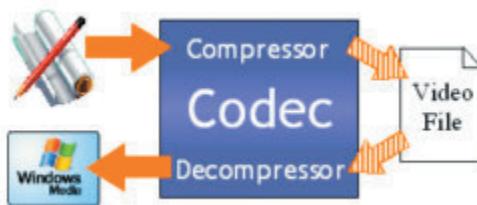
**Tip -** Use the Animation Export feature when you need to provide a presentation of your model to a client who does not have SketchUp or the SketchUp viewer. Or, export a video animation for post-processing in another presentation software package. Use SketchUp for simple presentations or when the customer has the SketchUp viewer.

## Selecting Animation Type

SketchUp supports both single video animation file export (in the form of a Audio Video Interleave or AVI file) and multiple image file export.

### Single Video File Animations

SketchUp allows you to export a TourGuide presentation as a single video animation file (Audio Video Interleave or AVI files). These files can be played using any compatible software video player or incorporated into a professional presentation using additional software presentation tools. SketchUp utilizes special third-party software called codecs (compressor/decompressor) to create these files.



Each codec included with SketchUp implements a compression technique designed for a specific medium such as Web video, Email, or CD-ROM.

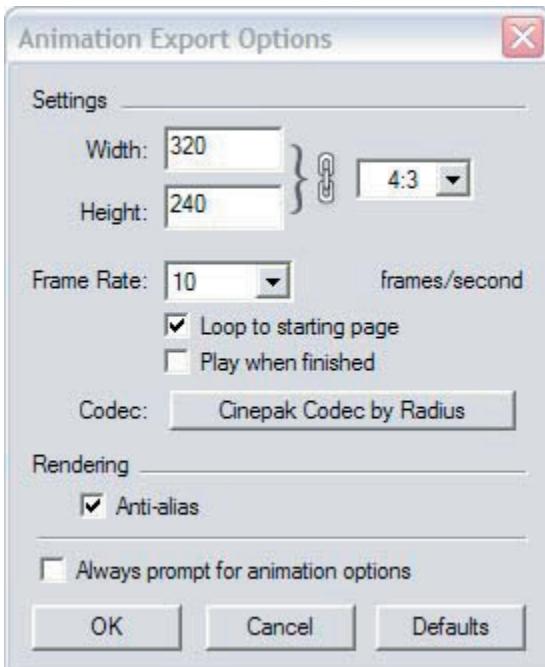
## Multiple Image File Animations

SketchUp also allows you to export a TourGuide presentation as a series of progressive image files. The number of files exported depends on the frame rate for the presentation. For example, a 5 minute presentation exported at 10 frames per-second will yield 3000 files (60 seconds x 5 minutes x 10 frames per-second). Multiple image file animations are typically generated to use as input into video animation software products for post-processing.

## Exporting TourGuide Presentations as Animations

Animation files can be used to present your model without the need for SketchUp. Or, you import your files into another presentation or animation product to further enhance your TourGuide presentation. To Export a TourGuide presentation as an animation file (or series of image files):

1. Select the File > Export > Animation... menu item. The Export Animation Dialog box is displayed.
2. Select the export type from the Export type drop-down list.
3. Enter the name for the exported file or series of files. The animation exporter will combine this name with a sequencing number for each image file that is created as part of a multiple image file animation. For example, if you export a 1 minute presentation called mypres.jpg, at 10 frames per-second, you will get 600 files, each named mypres0001.jpg, mypres0002.jpg, mypres0003.jpg all the way to mypres0600.jpg.
4. Click on the Options... button. The Animation Export Options dialog box is displayed.



5. Modify animation export options as necessary. See Animation Export Options in this topic for further information.

6. Click on the Export button to export the presentation as an animation.

## The Animation Export Options Dialog Box

The Animation Export Options dialog box contains options for exporting Animation files.

**Width/Height:** The Width and Height fields are used to control the frame size of the animation, as measured in pixels. The smaller the width and height values, the smaller the overall size, in kilobytes, in the video animation file.



**Tip -** Computer screens can handle resolutions of 1024x768 pixels and above. However, 640x480 pixels is considered a full-size video. Typical videos are created at an even lower frame size. For example, 320x240 yields acceptable data rates for CD-ROM, and transfers acceptably to videotape while 240x180 is acceptable for Web streaming. Values larger than 640x480 are not recommended unless specifically required such as when creating Full Quality Digital Video similar to that of professional videographers.



**Note -** Most personal computers can transfer 5 to 7 megabytes of data per second from the hard drive. The larger the frame size, the larger the resulting video animation file. Large video animation files can slow down your computer resulting in skipping.

**Aspect Ratio Lock:** The Aspect Ratio Lock icon is used to maintain a fixed proportion of video at any frame size. Click on this lock to use a nonproportional frame size in your video animation.

**Frame Rate:** The Frame Rate drop-down list is used to select the number of frames of animation per second (fps) of time in the presentation. The more frames per-second, the smoother the video animation. However, doubling the frame rate generally doubles the rendering time as well as the final size of the video. A setting between 8 and 10 is considered the minimum required for to produce smooth movement in your video animation. You might consider using a frame rate of 3 frames per-second while testing video animation export.



**Note -** There are frame rate requirements for certain mediums, such as 29.97 fps for television in the United States, 25 fps for television in Europe, 24 fps for film, and so on.



**Note -** Most personal computers can transfer 5 to 7 megabytes of data per second from the hard drive. The larger the frame rate, the larger the resulting video animation file. Large video animation files can slow down your computer resulting in skipping.

**Loop to Starting Page:** The Loop to Starting Page option is used to generate an additional video segment that animates from the last page of your presentation to the first (forming a continuous loop of video).

**Anti-Alias:** The Anti-Alias option is used to smooth the exported image. This option can increase the export time, but helps to reduce jagged lines and pixelated artifacts in images.

**Codec:** The Codec option is used to specify which codec, or compression plug-in, to apply to your video animation.

**Play When Finished:** The Play When Finished option is used to start your default video player and play exported video animation file after it has been created.

**Always Prompt for Animation Options:** The Always Prompt for Animation Options option is used to display the Animation Export Options dialog box every time a video animation is created.

# Pages and TourGuide™

Your SketchUp file can contain one or more pages. A page consists of your model and a series of page-specific settings, such as a specific point of view, shadow, display setting, and section cut. For example you can have your model on one page with a morning shadow and on another page with your model and a evening shadow. Then, use the TourGuide™ feature of SketchUp to cycle through the pages and watch the shadow transition from morning to evening (this is called a shadow study). Link several pages together, each with different settings, to form a presentation or TourGuide™ tour of your model. Activate the Page Manager from the Window menu to add, delete, and update pages.

## TourGuide™

TourGuide is a powerful™ presentation tool enabling you to create compelling presentations of your model. TourGuide is used to gradually transition the model from one state, in one page, to another state in another page. Configure TourGuide settings using the Tourguide panel of the Model Info dialog box.

### Running a TourGuide Tour

SketchUp contains controls for starting, stopping, and pausing a TourGuide™ tour. To Run a tour:

1. Select the View > Tourguide > Play Slideshow menu item. The Tourguide™ controls are displayed and TourGuide™ starts to cycle through your pages.
2. Press the Pause button to pause the slide show.
3. Press the Stop button to stop the slide show.



**Note** - You can also context click on a page tab and select Slideshow to begin a presentation.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

## Saving a Presentation as an Animation

SketchUp presentations can be exported as a video animation, or series of progressive raster images, using the File > Export > Animation... menu item. Refer to Animation Export Options for further information.

# Printing

SketchUp allows you to print your designs using any Windows-compatible printing device. You can also Print to Scale and span a print across multiple sheets, allowing you to output a large drawing from a standard printer. Printing is a three-step process:

1. Select the paper size for your printer using File > Print Setup...
2. Configure the output size and preview your output using File > Print Preview...
3. Print your Model using File > Print...

## The Print Setup Dialog Box

The Print Setup dialog box is used to select and configure your printer and final output orientation. Select File > Print Setup... to invoke the Print Setup dialog box.

### Printer

The Printer section of the Print Setup dialog box is used to select a specific printer and printer settings. Select the printer to use from the Name drop-down list. Click on the Properties button to configure the printer for printing (the properties dialog box is different for each printer-type).

### Paper

The Paper section of the Print Setup dialog box is used to select the paper size used for your output.

**Size:** The Size drop-down list contains different paper sizes that are supported by your printer. Select the paper size that you want to use from this list.

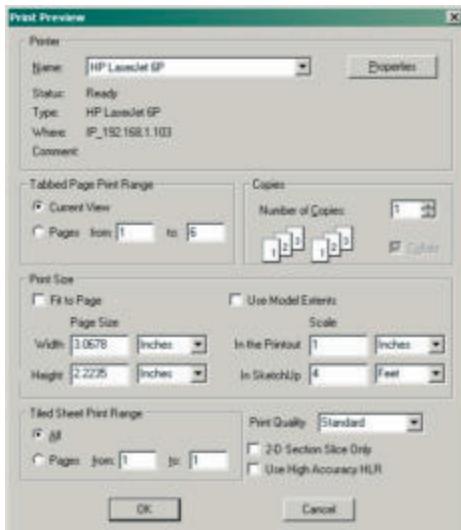
**Location:** The Location drop-down list contains different trays in your printer. Select the tray that contains the paper you want to use.

### Orientation

The Orientation section is used to select the orientation (landscape or portrait) for your output.

## The Print Preview and Print Dialog Boxes

The Print Preview and Print dialog boxes contain the same series of options. The only difference is that the Print Preview dialog box outputs your model to the screen instead of to a printer. Items configured in one of these dialog boxes automatically appear in the other dialog box. Select File > Print Preview... or File > Print... to invoke the Print Preview and Print dialog boxes.



The Print Preview and Print dialog boxes contain several printing options to consider when printing a SketchUp model.

### Printer

The Printer section of the Print dialog box is used to select a specific printer and printer settings. Select the printer to use from the Name drop-down list. Click on the Properties button to configure the printer for printing (the properties dialog box is different for each printer-type).

### Tabbed Page Print Range

The Tabbed Print Range section of the Print dialog box is used to choose between printing the model in the current Draw Area, or printing all models on all pages in the file.

### Copies

The Copies section of the Print dialog box is used to select the number of copies to print. Click the collate button if you want each copy to be collated (instead of printing several page ones, then several page twos, and so on).

### Print Size

The Print Size section of the Print dialog box is used to adjust the physical size of your output.

**Fit to Page:** The Fit to Page option is used to size the model to fit on a single sheet of paper. This option must be disabled in to specify a different size or scale.

**Page Size:** The Page size fields are used to enter a custom page size for your output.

**Scale:** The Scale fields are used to scale your model for printing. The first measurement, labeled In the Model, is the measurement of the exported geometry. The second measurement, labeled In SketchUp, is the actual measurement of the object in real scale. For example, for a scale of  $1/4'' = 1'$ , simply enter 1 inch In the Printout equals 4 feet In SketchUp.



**Note -** *It is not possible to print a perspective image to scale. Ensure perspective is disabled and you are using one of the Standard Views in the Camera > Standard submenu, to enable the scale option.*



**Note -** *Even though paraline projection is used to get an output that is to scale, not all angles will be measurable to a scale. This issue is the result of SketchUp using foreshortening, or a technique of shortening lines to create a 3D effect on a 2D medium such as paper.*

**Use Model Extents:** The Use Model Extents option is used to print only the model as viewed using the Zoom Extents Tool. This option might discard any surrounding empty background.

### Tiled Sheet Print Range

If the current Scale is larger than the paper size of your printer or plotter, the entire model can be printed by printing on several pieces of paper. These pieces of paper can then be taped together to create the final scaled model. For example, tiling lets you print proofs of a large model, such as a B size ( $11'' \times 17''$ ), on a printer that uses a smaller paper size, such as an A size ( $8.5'' \times 11''$ ). Tiling also lets you print banners that are made up of multiple pages.

Tiling is available when you use the Scale fields to scale a model to be larger than the current paper size available in your printer.

You can also print selected page tiles within the set by entering a page number range in the Pages from fields. Page tiles are numbered top to bottom beginning at the top left of the drawing page.

You can display a preview of tile pages using Print Preview in the File Menu.

Printing large tiled output can be taxing on your computers resources.

### Print Quality

You can expedite the printing of print jobs and rough drafts by selecting the appropriate print quality.

#### Draft

The Draft setting is used to produce quick images (usually output has jagged lines).

#### Standard

SketchUp<sup>®</sup>

The Standard setting is used to produce output with a the optimum balance of speed and quality.

#### **High Definition**

The High Definition setting is used to produce high-quality output.

#### **Ultrahigh Definition**

The Ultrahigh Definition setting is used to produce the highest-quality output. This setting can slow down your system, but is best for measured drawings or ultra-fine prints.

#### **Large Format**

The Large Format setting produces images that are optimized for large format output or that are meant to be viewed from several feet away. Resolution and line weights are set accordingly. This setting is useful for plotters and tiled sheets.

#### **2D Section Slice Only**

The 2D Section Slice Only option is used to output only the section slice outlines in your model (if present).

#### **Use High Accuracy HLR**

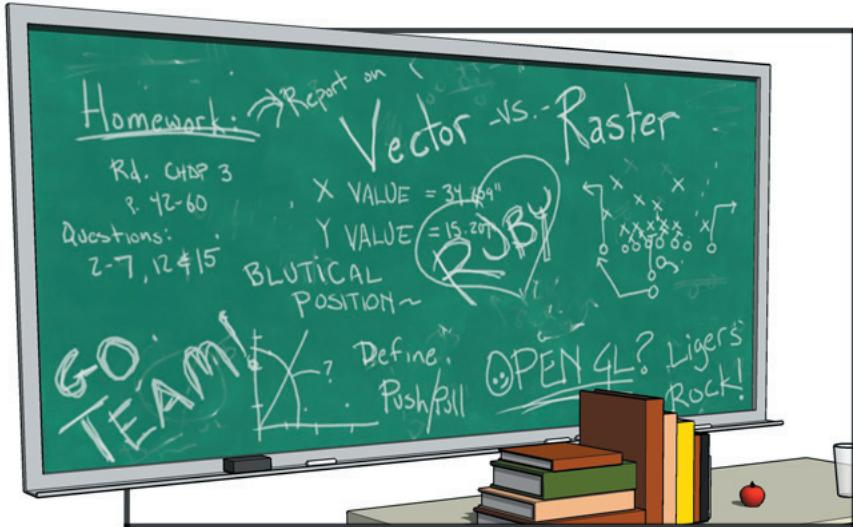
The Use High Accuracy HLR option is used to send the model information to the printer as vector information.



**Video** - A video tutorial on this topic is available at  
<http://www.sketchup.com/training/tutorials.php>

# Technical Info

This section of the documentation contains additional detailed information about various aspects of SketchUp.



# Raster File Formats

Raster images are made up of individual square colored pixels that, when combined, form some image. SketchUp supports several different raster image formats, each of which has its own particular advantages and should be selected depending on the task you want to perform.

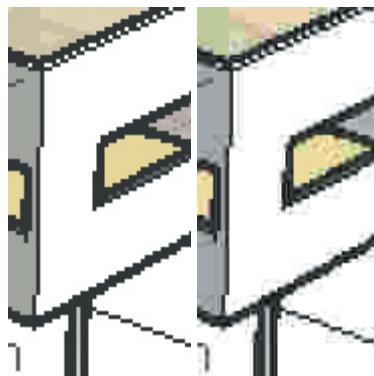
## JPEG (Joint Photographic Experts Group)

JPEG image files enjoy wide support across many applications, computer platforms, and digital cameras. The most prominent feature of JPEG is its compression scheme, which can reduce file size to a fraction of other formats. This compression scheme makes the JPEG format particularly useful for sending images, such as digital photographs, using email or for viewing on the world wide web.



**Tip - Consider using JPEG if your image contains lots of textured areas or fine detail**

JPEG compression works so well because it analyzes an image, divides it into blocks, and then discards the information which the human eye does not readily perceive. However, This high level of compression yields a lower image quality and therefore, sometimes resulting in blocky patterns of color referred to as *artifacts*. JPEG images are referred to as *lossy* because data is actually lost during the compression process and an uncompressed JPEG file will never be identical to the original file.



The previous image shows the artifacts resulting from JPEG compression (in the right-most portion of the image).

Fortunately, you can adjust the amount of JPEG compression to suit your needs using the File > Export > 2D Graphic... Options... > Image Quality slider bar.



**Tip - Resaving a JPEG image multiple times can result in compounded artifacts. Work with original raster images in a non-lossy format, such as PNG, before saving to JPEG for final output.**

## PNG (Portable Network Graphics)

The PNG format was developed for transferring images efficiently over the world wide web without data loss. Like JPEG, PNG does use compression, but it is a non-lossy variety that is free from artifacts.



**Tip** - Consider using PNG for models that have large areas of solid, uniform, colors (such as non-textured SketchUp models).

## TIFF (Tagged Image File Format)

The TIFF file format is often used to exchanged files between applications and computer platforms (Mac and Personal Computers). There are a large number of variations to the TIFF format available, and therefore TIFF might not be as universally accessible as other formats such as PNG and JPEG. SketchUp supports as wide a range of TIFF formats as possible, and you should test compatibility with other applications before exporting to TIFF file. 1-bit raster files, or TIFF files with gif encoding, are not compatible with SketchUp.

Also, while formats such as PNG and JPEG are viewable by any web browser and can therefore be sent to virtually any other person with good chance of success, TIFF files may require additional software to be viewed on some systems. TIFF is primarily used for high-resolution printing from desktop publishing applications.



**Note** - SketchUp does not support TIFF files at 1 bit per pixel. Images compressed using RLE will expand to uncompressed sizes once imported into SketchUp.

## BMP (Bitmap)

The BMP file format was created for Microsoft Windows operating systems primarily as the format used for the Microsoft Windows clipboard and for viewing (especially as wallpaper). BMP does not use compression. Therefore, BMP can be useful for creating temporary files to be further modified in other applications. However, BMP tends to create very large files that are not suitable for archive, web, and email use.

## TGA (Truevision Advanced Raster Graphics Adapter or Targa)

TGA files were developed by AT&T for use with Truevision video boards, which brought high resolution and full color capability to early computers. Today, all personal computers have graphics capabilities that easily surpass the early Targa boards. Nonetheless, the TGA format remains in use today for its combination of non-lossy compression, stability, and alpha channel support.

## Epix

Epix is the native format of Piranesi which is an architectural painting application designed to work with images generated from 3D models. See Piranesi Epix Export for more information.

# 2D Vector File Formats

2D vector file formats have advantages over bitmap image export in that they produce drawings that are resolution independent and can be easily imported and modified in many other programs. These file formats are useful for creating a set of 2D construction documents, plotting a perspective in a large format, or for import into vector illustration software for further refinement .

## EPS

Encapsulated PostScript format is based on PostScript, a graphics description language developed by Adobe as a standard way for graphics programs and print devices to communicate. EPS is widely used in the graphic design and publishing industries.

## PDF

Adobe® Portable Document Format (PDF) is the open standard for worldwide electronic document distribution. PDF preserves all the fonts, formatting, graphics, and color of the source document, regardless of the application and platform used to create the document. Also, PDF files are compact and can be shared, viewed, navigated, and printed exactly as intended by anyone with free Adobe Acrobat® Reader® software.



**Caution -** Some graphic features of SketchUp, including textures, shadows, smooth shading, backgrounds, and transparency, cannot be exported to PDF and EPS.

## DWG (AutoCAD Drawing File)

DWG was created by AutoDesk as the file format for their AutoCAD program. There are many different versions of DWG, including DWG r12, r2000, and r2004.

## DXF (Data Exchange File)

DXF is a 2D graphics file format supported by most all Macintosh and PC-based CAD applications. DXF was created by AutoDesk as the method for exchanging CAD data between their AutoCAD application and other CAD applications.

## The Anatomy of an Epix File

The native file format of Piranesi is referred to as an Epix (Extended Pixel) file. In addition to storing the actual rendered image, Epix embed additional information from the original 3D model. This information allows Piranesi's paint tools to intelligently render the image. There are three "channels" of an Epix file:

**RGB:** The first, known as the RGB Channel, contains the color of each pixel. This is the same data typically stored in other raster image formats. (In fact, Epix files are readable in most image editors as TIFF files.)

**Depth:** The second, known as the Depth Channel, stores the distance of each pixel from the eye point. This information helps Piranesi understand the surface topology under the image and allows it to apply textures, scale objects, lock orientation, and many other capabilities dependant on the 3D surfaces of your model.

**Material:** The third, a Material Channel stores the material for each pixel. This lets you paint one part of your rendering loosely without having to worry about painting another by mistake.

In General, Piranesi expects a flat shaded, non-textured Epix file. Some of the rendering modes in SketchUp, such as Wireframe and Hidden Line, don't really work well for Piranesi, and are disabled during export in favor of shaded output.

Other features of SketchUp, such as edges and textures, are also different from that which Piranesi expects, yet may be desirable in many cases. While adding support for Epix files in SketchUp, we have tried to adhere to Piranesi's expectations whenever possible while retaining enough flexibility to accommodate the varying needs of different artists and renderers.

# SketchUp and OpenGL

3D applications, such as SketchUp, require abundant system resources. Aside from having a fast CPU and large amounts of RAM, your graphics card and graphic card drivers must be 100% OpenGL compliant.

## What is OpenGL?

OpenGL is the industry-standard, used in numerous software applications and games, to draw 3D geometry. Most Microsoft Windows and Apple OS X operating systems come with a software-based OpenGL driver. However, these drivers rely heavily on the CPU to perform the rendering calculations of OpenGL (a task that is not done efficiently by most CPUs).

Many graphics card manufacturers have also built cards that support the OpenGL standard. These cards perform the rendering calculations using a specialized chip called the Graphics Processing Unit or GPU (instead of relying on the CPU). These chips significantly enhance OpenGL performance upward of 3000 percent. This performance enhancement is known as Hardware Acceleration.

## Enabling Hardware Acceleration

SketchUp will take advantage of hardware acceleration if your computer has a 100% OpenGL compatible video card.



**Tip -** *If the Use Hardware Acceleration option is not available in SketchUp, verify that the control panel settings for your video card's device driver has hardware acceleration enabled.*

Unfortunately, only some 3D drivers in the consumer video card market are 100% OpenGL compatible and can use this feature (though many cards claim to be 100% OpenGL compatible). Most 3D drivers are designed for games, and are often not tested using other 3D programs. Consequently, incompatibility problems can occur requiring a fix from the video card manufacturer. Disable this option if you are having problems with the 3D rendering of your models or if your video card is not 100% OpenGL compatible and does not support hardware acceleration.



**Note -** *Hardware Acceleration might only be available on your system for certain resolutions and color depths. Check the system settings for your graphics card to see if it supports hardware acceleration (using the Control Panel on Microsoft Windows).*



**Note -** *@Last Software strongly recommends that you set your display colors to a 32 bit color depth (using Control panel > Display Properties) to assure that your SketchUp model will render accurately when using hardware acceleration.*

@Last software cannot control the quality of the OpenGL driver on your computer system. Video card device drivers are proprietary and are maintained solely by the manufacturer of the video card in your system. Therefore, @Last Software cannot guarantee that SketchUp will work with hardware acceleration on your system.

# Compatibility Issues

OpenGL incompatibility is a significant system configuration issue leading to problems with SketchUp. Difficulties with Sketchup tools, performance, and rendering (such as mysterious graphics appearing on your screen) are usually the result of a video card not fully supporting OpenGL (despite claims by the manufacturer), an out-of-date video card driver, or incompatibility with 32-bit color depth. A temporary solution is to disable hardware acceleration in SketchUp while troubleshooting the problem.



**Caution -** Consult the Graphics Card Compatibility section of the Readme file (included with SketchUp) for additional details regarding compatibility issues for specific graphics cards.

## Common Symptoms of Sub-Standard OpenGL Cards

Following are several symptoms of incompatible OpenGL video card.

### Streaking or dark artifacts when using face-casting shadows

Some OpenGL drivers contain a bug related to face-casting shadows. Update your device driver if you see streaking or dark mysterious graphics appearing on the screen when using face-casting shadows.

### SketchUp crashes

Many 3D video card cards and drivers are designed for video games, but are not 100% OpenGL compatible (despite manufacturer claims) and only contains a subset of the features needed for other 3D applications. These noncompatible cards and drivers can cause SketchUp and other 3D applications to crash.

### When selecting faces, the reverse faces are picked

This is a common OpenGL bug to which we have found a work-around. The work-around may be enabled by selecting "Correct Reversed Picking Driver Bug" in the OpenGL tab of the Preferences Dialog. This is recommended over disabling 3D Hardware Acceleration.

### Missing axes, all lines visible and dashed, and strange texture colors (blue bricks) in 16 bit mode

The device drivers for the ATI Rage Mobility (commonly used on laptops) are not 100% OpenGL compatible. Use software rendering by turning off hardware acceleration to solve this problem..



**Caution -** Consult the Graphics Card Compatibility section of the Readme file (included with SketchUp) for additional details regarding compatibility issues for specific graphics cards.

# Codec Lists

SketchUp's animation export feature uses codecs (COmpressor/DECompressor) to create the appropriately sized video depending on your needs. Codecs differ in their intended use, compression method, and handling pictures and sound. If you plan to edit your video, you might want to choose a codec that provides higher quality at the expense of file size and compatibility. Conversely, if you plan to email your video or play it off a CD, you might want to select a codec that provides lower data rates at the expense of quality. Some codecs are also designed specifically for a certain device, such as a digital video camera.

You can also choose to create uncompressed video by not using a codec. However, a large uncompressed animation file might require a fast system and very large amounts of disk space.

Following is an explanation of codecs that are included with SketchUp.

## Cinepak Codec by Radius

This is the default codec for SketchUp. Although not as advanced as modern codecs, videos compressed using Cinepak will work reliably on any platform, play smoothly from CD-ROM, and will provide good file size compression. Cinepak is asymmetrical and lossy.

## Intel Indeo Video R3.2

This codec was designed for video playback from CD-ROM discs. It is comparable in quality to the Cinepak codec, but Cinepak is preferable due to wider support.

## Microsoft Video 1

This codec originally shipped with Video for Windows (Windows 3.1) and is low quality and rarely used today.

## Intel Indeo Video 4.5

This is a lossy codec that provides good compatibility, speed, and flexibility with average image quality. Advanced features include a quick compression option, keyframe control, transparency, and the ability to adapt to the available bandwidth. (Access to some of these requires software available separately from Intel.)

## Intel Indeo Video 5.10

This is a lossy codec that provides good compatibility, speed, and flexibility with average image quality. Advanced features include a quick compression option, keyframe control, transparency, and the ability to adapt to the available bandwidth. (Access to some of these requires software available separately from Intel.)

## Full Frames Uncompressed

This option allows you to export your animation without any compression.

# Bug Splat

BugSplat ([www.bugsplatsoftware.com](http://www.bugsplatsoftware.com)) is a 3rd party software application that is integrated in SketchUp for Microsoft Windows 2000 and XP to help improve the quality of the product. BugSplat helps @Last Software troubleshoot unrecoverable SketchUp errors or *crashes*. BugSplat provides a mechanism for Microsoft Windows users to send information about all crashes to @Last Software for troubleshooting.

## How Does it Work?

In the rare event that SketchUp crashes you will see a dialog, similar to Microsoft Window's crash report dialog. This dialog will give you the option to send us the information regarding the crash. Here is what is collected if you decide to send us the information:

- Your version of the SketchUp executable.
- Your language setting, such as English.
- A "stack trace" of the bug allowing us to see the exact line of code where SketchUp crashed.
- A list of SketchUp code dependencies.
- (optional) Your name and email address.
- (optional) A description of what you were doing before the crash.



**Note -** If you happen to look at the BugSplat Software website, you will see a security statement that is aimed at assuring clients of BugSplat - in this case @Last Software – that the information on our crashes (e.g., how many we have, and so on) will be protected. The above information is all that we are collecting.

## Loading An Offline Crash Report

If a crash occurs when you are not connected to the Internet, BugSplat displays a dialog box indicating that a zip file has been created on your local hard drive. This dialog box will also indicate the name and location of the zip file (normally AtLastCrashMMDDYY\_HHMMSS.zip in your Windows temporary directory). Go to [http://www.bugsplatsoftware.com/post/post\\_form.php](http://www.bugsplatsoftware.com/post/post_form.php) and follow the instructions on the Web page to send this file to BugSplat Software when you reconnect to the Internet. Or email the zip file to @Last Software directly at support@sketchup.com.

# SketchUp Ruby API and Console

SketchUp contains a Ruby application programming interface (API) for users who are familiar with (or want to learn) Ruby scripting and want to extend the functionality of SketchUp. This interface allows users to create tools, menu items, and other macros, such as automated component generators, to be included in the menus within SketchUp. In addition to the API, SketchUp also includes a Ruby console which is an environment where you can experiment with Ruby commands or *methods*.

For additional information on the Ruby programming language, visit [www.rubycentral.com](http://www.rubycentral.com).

Click on Help > Ruby Help for additional information on SketchUp Ruby API.

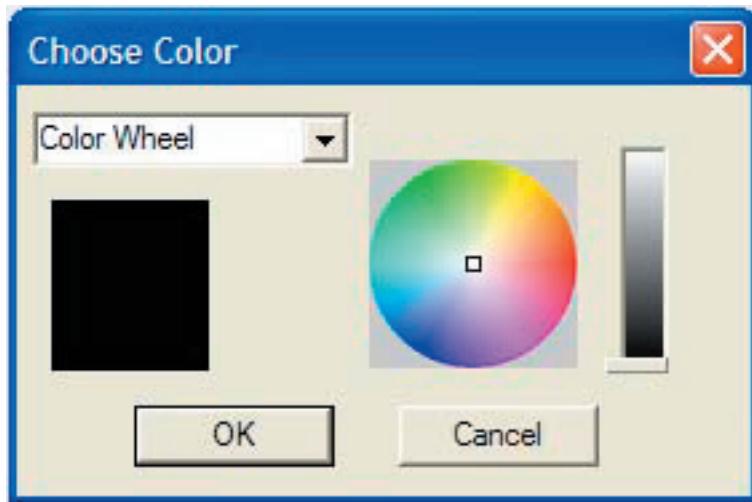
@Last Software provides unlimited technical support for SketchUp via email. Support is limited to the English language. Currently, we do not offer technical support for the Ruby Application Programmers Interface (API) or for any Ruby scripts created by third parties. We encourage posting Ruby API questions to our SketchUp Ruby API Forum. General Ruby information may be obtained at <http://www.ruby-lang.org/en/>.

@Last Software reserves the right to change this policy at any time.

# Color Pickers

The Choose Color dialog box contains a variety of different color controls that combine to give you an incredibly versatile tool for experimenting with color in your SketchUp models.

You can activate the Choose Color dialog box by clicking on any color swatch in the SketchUp user interface (such as in the Colors section of the Model Info dialog box).



## Color Picker Menu

SketchUp allows you to choose between four color pickers: Color Wheel, HLS, HSB, and RGB. Click on the color picker that you want to use in the drop-down menu within the Choose Color dialog box.

### Active Color Well

The Active Color Well, which is below the Color Picker menu, contains the current color selection.

The contents of the Active Color Well are maintained as you switch between pickers, allowing you to use different pickers to create your colors.

## Color Pickers

SketchUp contains four color pickers, used to select materials that do not have texture (just color).



*Note - Ensure that the Shaded or Shaded with Textures display style is selected to see the colors that you have applied to entities.*

## Color Wheel

The color wheel arranges color hue radially around the wheel, with the highest saturation at the outer edge of the wheel. To change the brightness of the color, slide the value slider to the right of the wheel up or down.

To use the Color Wheel, select the color you want by clicking your mouse over that color in the wheel. You can also click and drag the cursor around the Color Wheel to quickly browse through many different colors which will dynamically appear in the Active Color Well.

## HSB (Hue, Saturation & Brightness) and HLS (Hue, Light, and Saturation)

The HSB and HLS Color Pickers allows you to pick colors from the HSB (Hue Saturation and Brightness) color range. HSB often gives you a more intuitive color model for selecting desaturated colors.

Adjust the sliders for Hue, Saturation and Brightness until you see the color you want in the Active Color Well.

Sometimes it is easier to mix desaturated colors using the HSB or HLS Color Picker in conjunction with one of the other color pickers. Switch to another color picker to make your rough color selection, then switch back to the HSB or HLS Color Picker to fine tune your color.

## RGB (Red, Green & Blue)

The RGB Color Picker allows you to select colors from the RGB (Red, Green and Blue) color range. RGB colors are traditionally used when modeling color on a computer screen and represent the closest approximation of the actual range of colors recognizable by the human eye. RGB has a wide color gamut and is one of the most effective color spaces to use in SketchUp.

Drag the sliders left and right for each of the component colors (red, green and blue) to create the color that you want to use.

You can type in exact numerical values for red, green or blue in the value boxes to the right of the sliders if you want to precisely match a color that you have specified elsewhere.



## Glossary

### A

Angular Rotation Value - The value in the VCB that is displayed when measuring an angle with the Protractor Tool.

Autofold - The process of skewing, distorting, or folding geometry by selecting a specific point on an edge or face and moving the points. See also Sticky.

Axonometric - Meaning “measurable from the axes.” An axonometric projection is a view of a model in which lines appear parallel in both 3 dimensional and 2 dimensional space, and lines have a length that is to some scale, (for example 1? in SketchUp = 1? in the real world).

### C

CAD - See Computer Assisted Design

Camera - A software implementation of the user’s point of view. See also Point of View.

Camera Angle - See Point of View.

Camera Tool - A software tool used to alter the user’s point of view of a SketchUp model.

Chord Length - The distance between the starting point and the ending point of an Arc entity.

Component entity - An entity that contains other entities. Components are models that are saved as SketchUp files for reuse in other SketchUp files.

Component Brower - The Component Brower is used to instance [Component](#) entities from component definitions, including those components that you create and a variety of pre-built Components that you can use in your models.

Component Definition - The blueprint that defines the appearance and behavior of all component instances. Component definitions are created and stored in the Component Brower either as part of a Component Library or when you create a component.

Component Instance - An instance of a component definition, such as a single instance of an office chair component. You can have multiple component instances from the same component definition.

Component Library - A series of related component definitions, such as a component library of doors or windows.

Computer Aided Design - Software used for precision design. CAD is traditionally rigid and is often used for creating construction documents from which you would create the actual item you are designing.

Construction Tools - Tools used to create construction geometry and prepare physical documentation.

**Context** - An isolated realm that separates geometry within from anything outside. When you start a SketchUp model you are working within the model context. When you make a Group or Component you are essentially creating other contexts inside of the model context. When you edit groups or components you are in the group or component context. Certain commands or operations, such as the Unhide All command, scaling using the Measure Tool, and placing active Section Planes, are limited to a particular context. Additionally, when you create and select other entities, you are essentially in the entity's context.

**Context Menu** - A menu of menu items or commands available in a particular context, such as when a line or component is selected.

**Cutting Behavior** - The ability of component entities to cut holes in faces. The component's cutting behavior is established when the component is initially created.

**Cutting Plane** - A plane defining the exact location where the component will cut into a face.

**Coplanar** - Refers to entities that exist in the same plane.

**Curved Face Set** - A series of faces joined to make up a curved surface, such as a tube or arch.

## D

**Dialog Stack** - Dialog boxes arranged such that they form a stack.

**Distort** - Twisting out of natural shape.

**Drawing Area** - The area within the Drawing Window where you create your model. See also Drawing Window.

**Default Material** - A material assigned to all newly created faces. The default material is different for the front and back of faces.

**Drawing Tool** - A software tool used to create entities and geometry from scratch.

**Drawing Window** - The entire application window for SketchUp. The Drawing Window contains menus, toolbars, the status bar, and the Drawing Area. See also Drawing Area.

## E

**Edge-Based Modeling** - A form of modeling in which a surface is automatically created for 3 intersecting coplanar edges.

**Edge entity** - Edges bound faces within geometry. The term edge and line are used interchangeably.

**Entity** - The smallest graphical building blocks in SketchUp. Entities are used to create geometry which, in turn, is used to create models.

**Extrude** - The action of thrusting out or growing a form.

**Extrusion** - A form that has been extruded. See Extrude.

## F

**Functional Terrain** - Terrain that has no portion of itself bending back upon itself creating overhangs, underhangs or caves

**Face entity** - A planer entity bounded by 3 or more intersecting coplanar edges or lines.

**Face Shadows** - Shadows projected onto horizontal faces by vertical faces based on the sun's angle in the sky.

## G

**Geometry** - The combination of 3 or more entities. Geometry in SketchUp usually refers to a indistinguishable portion of a component or model.

**Ground Plane** - A flat or level surface representing the ground. In SketchUp, the ground plane is represented by the exact location of the red and green axes.

**Group entity** - An entity that contains other entities. Groups are commonly used to combine several entities into a single entity for the purposes of performing a quick operation, such as a copy and paste.

**Gluing Behavior** - The ability of component entities to align to faces in a specific orientation. The component's gluing behavior is established when the component is initially created.

**Ground Shadows** - Shadows projected onto the ground plane by vertical faces based on the sun's angle in the sky.

## H

**Healing** - The concept of joining two faces by erasing, or "healing" a line that intersects a face. Creating one face from two.

## I

**Inference** - The identification of relationships between entities in the drawing area. These relationships are identified and pointed out to the user by the inferencing engine and can be used as references for drawing in 3D space.

**Inference Engine** - Functionality built into SketchUp to locate or infer points from other points in your model, such as the center of a circle, the midpoint of a line, a line that is perpendicular to the ground plane, a point on a face, a point on an edge, and so on. The Inference Engine notifies you of these points by using both color indicators and on-screen messages indicating the location of the cursor as you draw an entity. For example, SketchUp displays the string ?From Point? when the line you are drawing is in plane with another point.

**Insertion Point** - The point where the cursor will grab and insert the component in to a model from the component browser.

**Instance** - see Component Instance.

**Intersection** - The concept of splitting faces and edges to create additional independent faces and edges by intersecting the face or edge with a line.

## L

**Layer** - Layers are used to control the visibility of geometry within large models. A SketchUp layer is an attribute with a name, such as "Layer0," "Layer9," or "Chairs." Elements can be assigned different layers.

SketchUp<sup>®</sup>

Layer Manager - A dialog box used to apply and manage layers in your model

## M

Manager - A dialog box used to store and manipulate a specific aspect of SketchUp functionality such as pages, components, or materials.

Material - Software-simulated paints that have both color and texture attributes. For example, a brick material might contain the color of red and a rough appearance or texture similar to real brick.

Modal Dialog Box - A dialog box that temporarily prohibits the user's interaction with the application. Modal dialog boxes usually require the user to perform some action prior to returning to normal application use.

Model - The contents of a SketchUp file. See also component.

Modification Tool - A software tool used to modify existing entities or geometry.

Modeless Dialog Box - A dialog box that does not prohibit the user's interaction with the application. See also Modal Dialog Box.

Move Point - The point where you click on the entity with the Move Tool.

## O

One-Point Perspective - In central perspective or one point perspective there is only one vanishing point (vp), which is located straight in front of the viewer: it is the vanishing point for the direction of view. This is not determined by our view, but the flat faces and tunnel walls of all objects within view. All lines drawn in the blue axes are exactly perpendicular to the ground plane and all of the lines drawn in the red and green axes are exactly perpendicular to the blue axes when viewing a 2 or 3 dimensional drawing viewed in one point perspective.

Origin - The point where the Drawing Axes start or originate.

Organic Shape - Geometry or models that have a hand-made or custom appearance.

## P

Page - Similar to a slide in presentation software, a page consists of your model and a series of page-specific settings, such as a specific point of view, shadow, display setting, and section cut. Pages can be combined to form TourGuide presentations in SketchUp.

Paraline - See Axonometric.

Perspective - A distortion of the Camera Angle such that it represents the model as though you were standing at a fixed position and looking at it without moving (certain items appear closer while other items appear to be far away; entities are not to scale).

Plane - A flat or level surface.

Point of View - The user's view of the model.

Polygon Mesh - A surface comprised of polygons, each derived from irregularly spaced points.

POV - see Point of View.



Point of View - The User's view of the model.

Principal Tools - Tools that tend to be used most often in SketchUp.

Pulling - The process of reshaping your model by shrinking a portion of your model back toward its starting point along a single axis.

Pushing - The process of reshaping a your model by expanding a portion away from its starting point and along a single axis.

## Q

## R

Ruby Console - Ruby console which is an environment where you can experiment with Ruby commands or methods.

Ruby Script - A small program, written in the Ruby programming language, that adds functionality to SketchUp.

Ruby Console - Ruby console which is an environment where you can experiment with Ruby commands or methods.

Ruby Programming Language - An object oriented scripting language.

## S

Sandbox - Surfaces created with the Sandbox from Scratch and Sandbox from Contours are referred to as sandboxes because like sand in a sandbox, these surfaces can be sculpted using a special set of tools called sandbox tools. Other terms for sandbox are terrain or Triangulated Irregular Network.

Score - A thin slice allowing for folding.

Section Cut Effect - The result of slicing through your model to see inside and optionally modify its inner working.

Section Plane - Special entities that are used to control the selection, placement, orientation, direction, of the *section slice*.

Section Slice - The edges created by the intersection of geometry at a *section plane*.

Skew - To place at an angle.

Split - The concept of creating two faces from one by dividing the face with a line.

Stickiness - Entities are said to be "sticky" in that when connected to another entity, moving one entity might move or alter the another, attached, entity. See also Autofold.

Surface - A series of joined faces.

## T

Three-Point Perspective - in three-point perspective there are three vanishing points (vp), allowing you to construct a form in any orientation. Three-point perspective is usually used to represent three dimensions in a three-dimensional medium such as SketchUp.

TIN - See Triangulated Irregular Network.

Triangulated Irregular Network - A surface comprised of triangles, each derived from irregularly spaced points. This surface is also referred to as a Sandbox and Mesh.

Triangulation - The orientation of triangles in a TIN (horizontal or vertical). See also Triangulated Irregular Network.

Two-Point Perspective - In two-point perspective there are two vanishing points (vp), which are located to the left and right of the viewer. Two-point perspective is usually used to represent three dimensions on a two-dimensional medium. All lines drawn in the blue axes are exactly perpendicular to the ground plane when viewing a 3 dimensional drawing viewed in 2 point perspective.

## V

Value Control Box (VCB) - The VCB displays dimensional information while you draw. You can also enter values into the VCB to manipulate the selected entity.

## W

Walkthrough Tools - Tools used to view your model as though walking around and in your model

## X

X-Ray Mode - A display setting whereby all faces have an applied global transparency. X-Ray mode is useful for seeing, and sometimes editing, the inner workings of your model

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