SketchUp

User Manual

Version 19 for macOS and Windows

Foreword

This manual is a complimentary private work made from previous manuals, mainly the 2016 version in PDF built by Kārlis Bruņenieks, and from some of the current Help pages (for version 19) of the SketchUp web site.

The author has no links with Google or the Trimble Company.

This manual has been made for the SketchUp community of users to ease the learning of the application offline[1]. It is free for download[2].

All the text and images in this document are © 2019 Trimble, published with requested authorization.

Should Trimble consider publishing this manual as an official Trimble document, the author will be more than happy to make his Word source file available.

Warning: This PDF file is in A4 format. A *Letter* format is in works (all the pix have to be changed from *free position* to *bound-to-text* first*).*

This edition in PDF format, all additional text (in blue) and footnotes are © 2019 Didier Morandi. Corrections and suggestions are more than welcome via mail at didier.morandi@gmail.com.

Toulouse
October 23rd, 2019.

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[1] To be able to read it later on an iPad without Internet connection, download the PDF then open it with iBooks, a copy will be stored locally.
[2] File (PDF, 211 pages, 3.4 Mb) is available from
http://www.didiermorandi.fr/SketchUp/SketchUp_User_Manual_v19.pdf

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User Interface

- Welcome screen
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Welcome screen

Developed for the conceptual stages of design, SketchUp is a powerful yet easy-to-learn 3D software. We think of it as the pencil of digital design. This award-winning software combines a simple, yet robust tool-set that streamlines and simplifies 3D design inside your computer. SketchUp is being used by anyone with the desire to Dream, Design and Communicate in 3D!

The Welcome to SketchUp dialog box appears when you first run SketchUp. This dialog box is primarily used to choose a template for use in SketchUp (templates have predefined settings, such as units of measurement), open an existing project, license a copy of SketchUp Pro, and learn more about SketchUp.
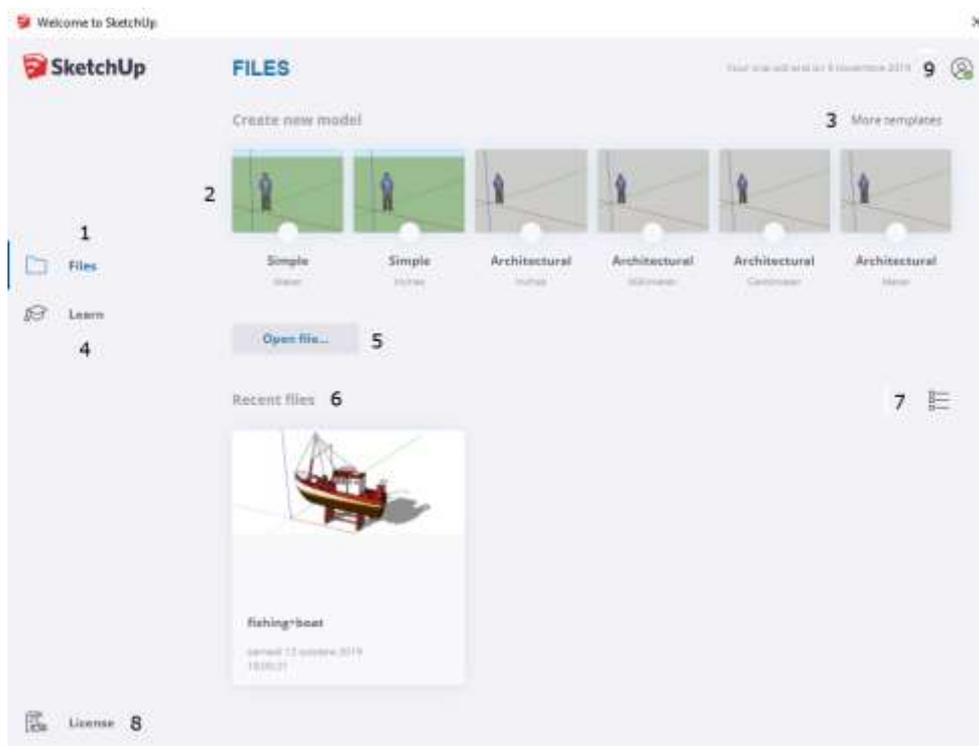
Legend of previous picture:

1 -   **Files** Tab: Display templates to create a new model.
2 - **Template** List. Click to select.
3 - **More Template** Button: Click to display more templates.
4 - **Learn** Tab: Display options to learn SketchUp (Forum, Campus and Videos).
5 - **Open file...** Button: Click to locate and open an existing project.
6 - **Recent files**: History list of previous projects.
7 - **List formatting** Button: Toggle recent files list (summary or details).
8 - **License** Tab: Display license information.
9 - **User Profile** Button: User profile management and Sign Out option.

Main screen
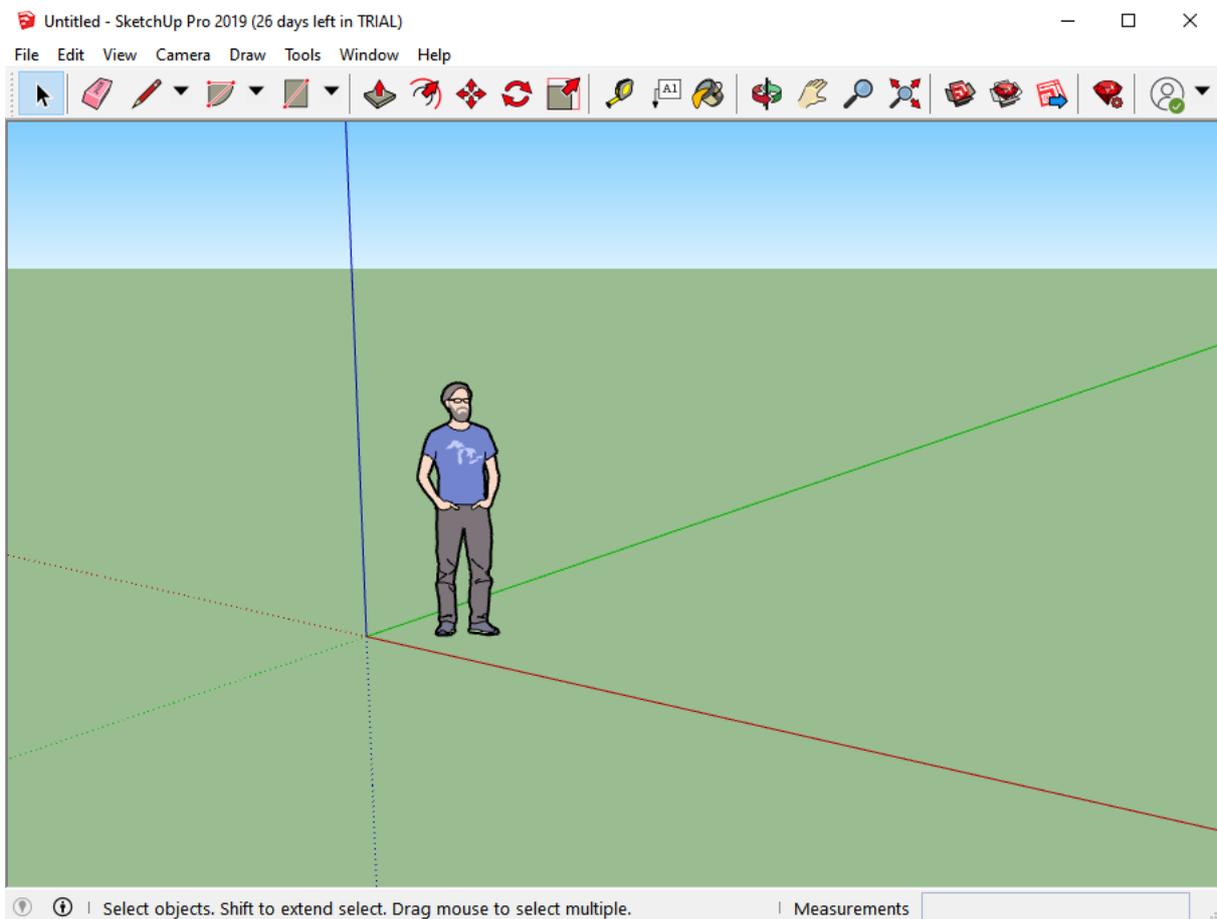
The Main screen displays the following Menu commands :

- File
- Edit
- View

- Camera
- Draw
- Tools
- Windows
- Help

Below the Menu list are toolbars located.

At the bottom of the screen, a status bar displays the following buttons and information :

- Geo-location Button
- Credits Button
- Context sensitive description
- Measurements Field

Toolbars

The toolbars which appear below the menus and sometimes along the left side of the application contain a user-defined set of tools and controls. SketchUp displays only the *Getting Started* tool palettes by default. To show or hide other tool palettes:

1. Select **View > toolbars.** The *toolbars* sub-menu displays.
2. Select any toolbar to show or hide. toolbars with a check mark are currently displayed. toolbars without a check mark are not currently displayed.
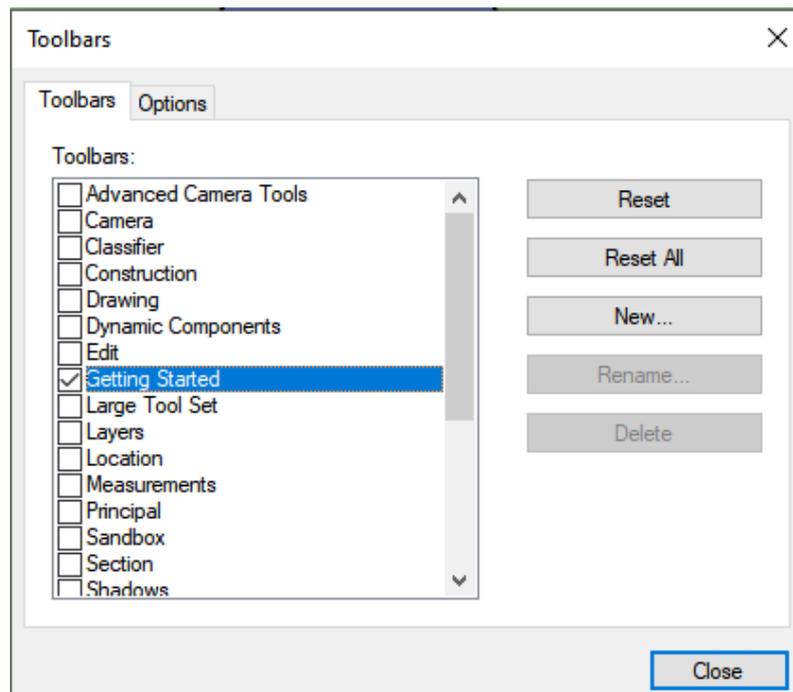
Note: A right click on any toolbar displays the list of all toolbars.
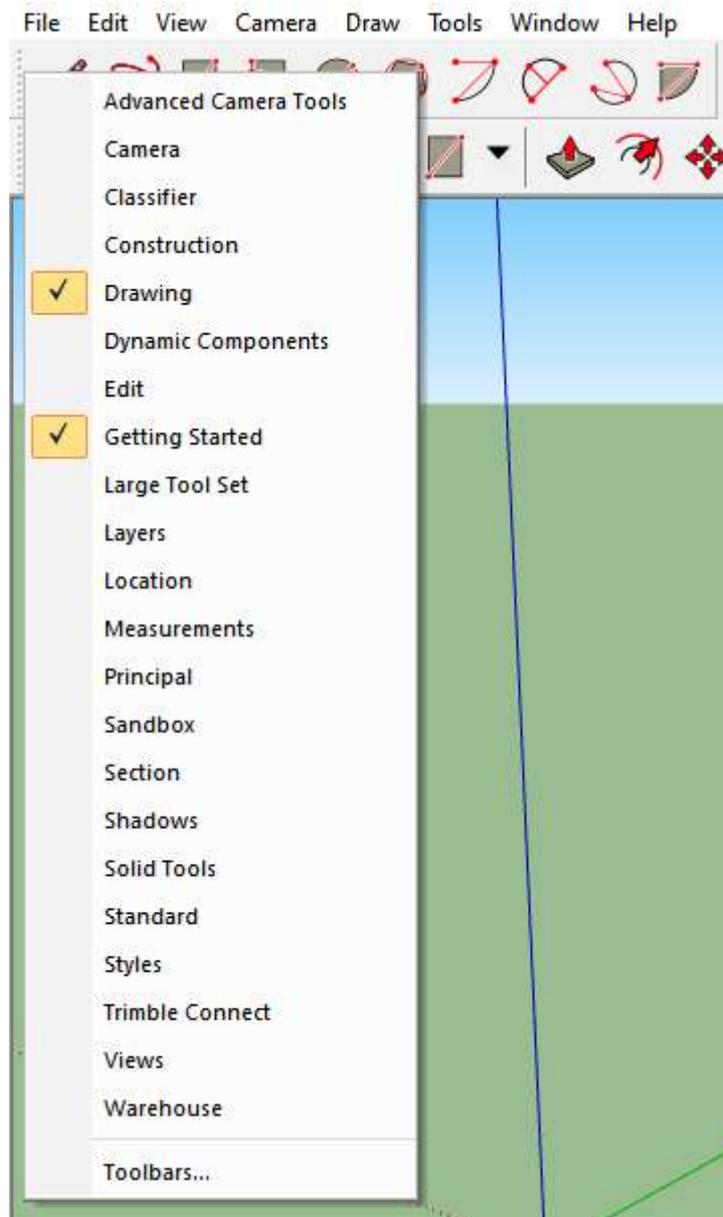
Note: Toolbars can be moved outside the screen of the SketchUp program onto the desktop, if the main SketchUp window is not resized to its maximum size.

Advanced Camera Tools toolbar

The buttons on the *Advanced Camera Tools* toolbar allow you to create a scene with multiple cameras (that have set aspect ratios) which can be used to mimic a film set. The buttons activate the *Create a physical camera with real world parameters* tool, *Look through a camera created by 'create Camera'*, *Lock/Unlock the current camera*, *Show/Hide all cameras created by 'create Camera'*, *Show/Hide all camera frustrums*

lines, *Show/Hide all camera frustrum volumes*, and *Clear the aspect ratio bars and return to the default camera* tool.

Camera toolbar

The buttons on the *Camera* toolbar activate the *Orbit tool, Pan tool, Zoom tool, Zoom Window tool, Zoom Extents tool, Previous, Position Camera tool, Look Around tool* and *Walk* tool.

Classifier toolbar

Display a pop-up menu to choose a classification system to load into the model.

Construction toolbar

The buttons on the *Construction* toolbar activate the *Tape Measure tool, Dimensions tool, Protractor tool, Text tool, Axes tool,* and *3D Text* tool.

Drawing toolbar

Display the Drawing toolbar.

Dynamic Components toolbar

The buttons on the *Dynamic Components* toolbar activate the *Interact with Dynamic Compoments tool, Component Options* dialog box*,* and *Component Attributes* dialog box.

Edit toolbar

The *Edit* 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.

Getting Started toolbar

Display the default toolbar.

Large Tool Set toolbar

Located on the left side of the screen, the *Large Tool Set* toolbar displays a set of most commonly used tools.

Layers toolbar

The *Layers* toolbar provides quick access to several often used layer operations.

Location toolbar

The buttons on the *Location* toolbar activate the *Add Location tool,* the *Toggle Terrain* tool and the *Add Texture* tool.

Measurements toolbar

The *Measurements* toolbar contains a field where you can type values corresponding to the currently activated tool. For example, you can type in the specific length of a line while you are in the *Line* tool. The *Measurement* toolbar is used to create accurate geometry.

Principal toolbar

The buttons on the *Principal* toolbar activate the *Select tool, Make Component, Paint Bucket tool, and Eraser* tool.

Sandbox toolbar
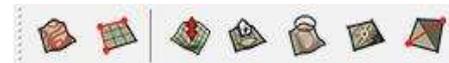
The buttons on the *Sandbox* toolbar activate SketchUp's Sandbox tools (*Sandbox From Contours, Sandbox From Scratch, Smoove tool, Stamp tool, Drape tool, Add Detail tool,* and *Flip Edge tool*).

Sections toolbar

The *Sections* toolbar allows you to conveniently execute common section operations. The controls on this toolbar include buttons for toggling the section cut effect and toggling the display of section planes.

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).

Solid Tools toolbar

The buttons on the *Solid Tools* toolbar activate the *Outer Shell tool, Intersect tool, Union tool, Subtract tool, Trim tool,* and *Split* tool.

Standard toolbar

The *Standard* toolbar contains a variety of menu items which help with file and drawing management and shortcuts to printing and help operations. These menu items are *New, Open, Save, Cut, Copy, Paste, Erase, Undo, Redo, Print,* and *Model Info*.

Styles toolbar

The buttons on the *Styles* toolbar activate SketchUp's face rendering styles (x-ray transparency, wireframe, hidden line, shaded, shaded with textures, and monochrome).

Trimble Connect toolbar

The *Trimble Connect* toolbar displays the *Trimble Connect* buttons. To know more about Trimble Connect, please visit the Trimble Connect Web site pages at https://connect.trimble.com.

Views toolbar

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

Warehouse toolbar
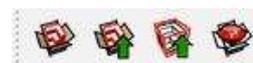
The buttons on the *Warehouse* toolbar are used to collaborate between SketchUp 3D Warehouse and Extension Warehouse. The toolbar contains the *Get Models..., Share Models...., Share Component...,* and *Extension Warehouse*. If you have an extension/plugin installed that has an update available, the *Extension Warehouse* icon will show a yellow circle with two circular arrows.

Reset Button

Allows a selected toolbar to be reset to its default buttons.

Reset All Button

Allows all toolbars to be reset to their default buttons.

New Button

Creates an empty toolbar in which any button can be moved to.

Rename Button

Changes the name of a newly created toolbar.

Delete Button

Deletes a newly created toolbar.

Close Button

Closes the toolbar control panel.

Options Tab
Show ScreenTips on toolbars

If you hover your mouse over a toolbar, a "ScreenTip" with more information about the tool will appear in a pop-up window and in the status line located in the bottom of the screen.

You can choose whether or not SketchUp shows that information in the pop-up window.

Click the *Options* tab then check the *Show ScreenTips on toolbars* checkbox to switch between showing the screen tips in a pop-up window or not.

Note: Screen tips are always displayed in the status line.

Large Icons

The toolbar can be displayed using small and large tool buttons. Click the *Options* tab then check the *Large Icons* checkbox to switch between small and large tool buttons.
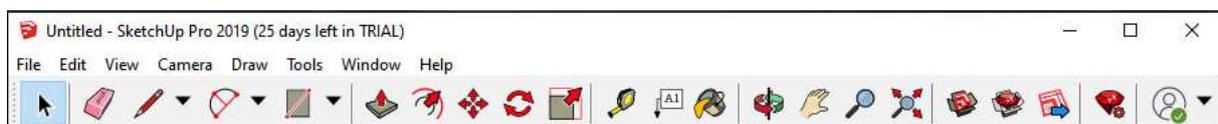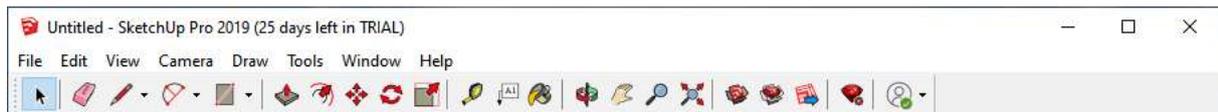
Menu commands

- File
- Edit
- View
- Camera
- Draw
- Tools
- Window
- Help

File
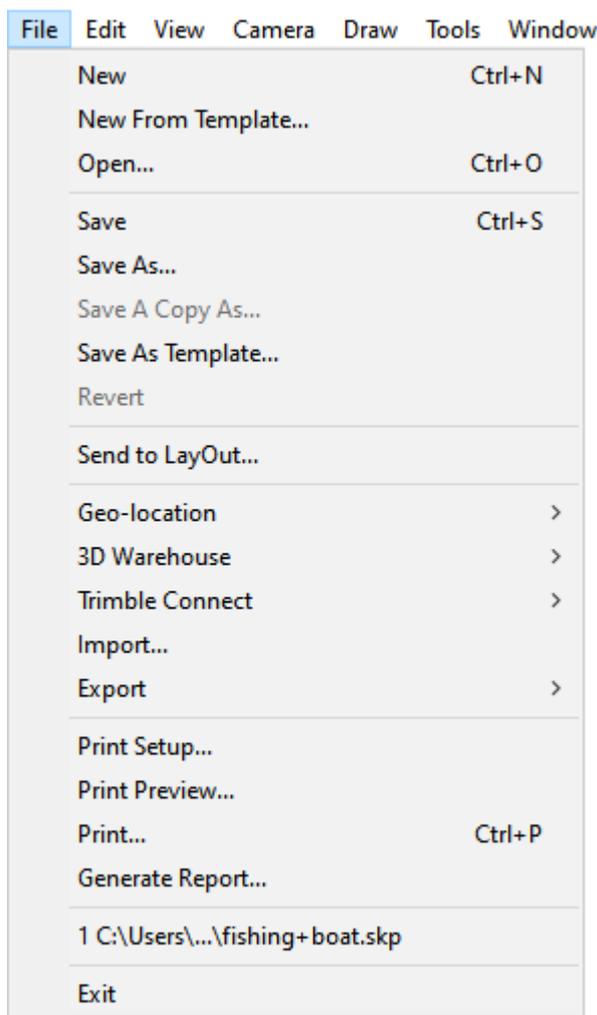
Edit
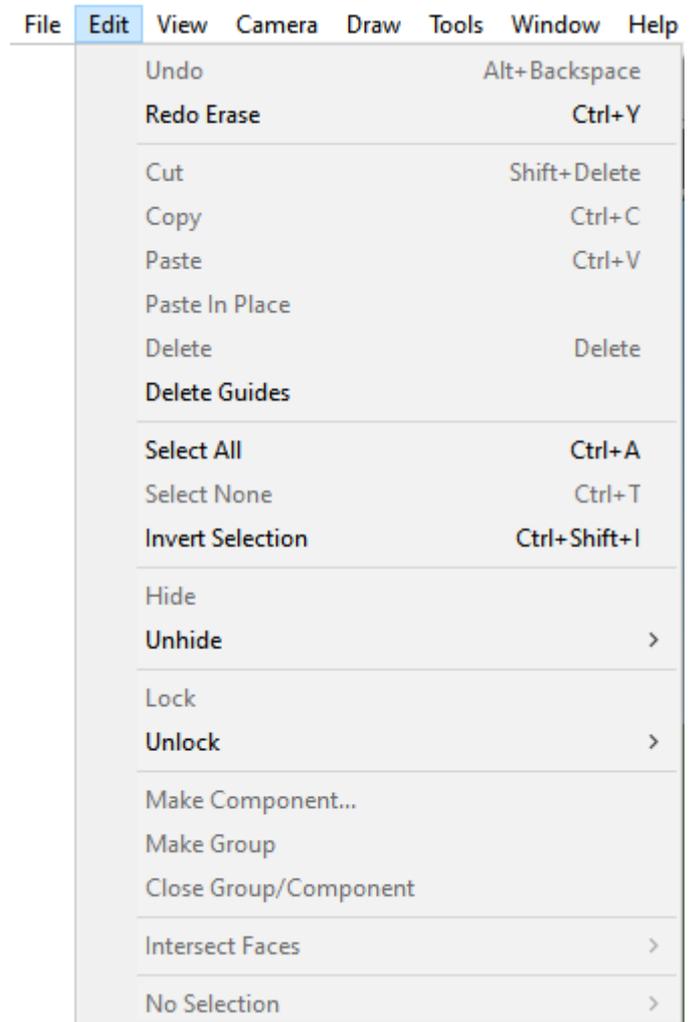
View
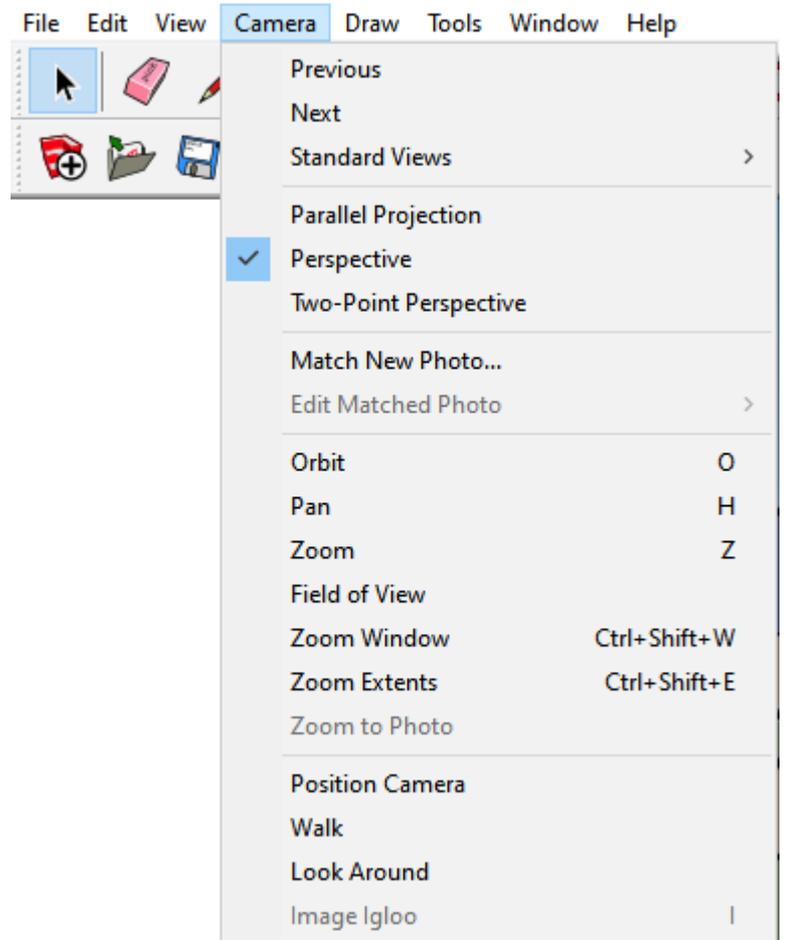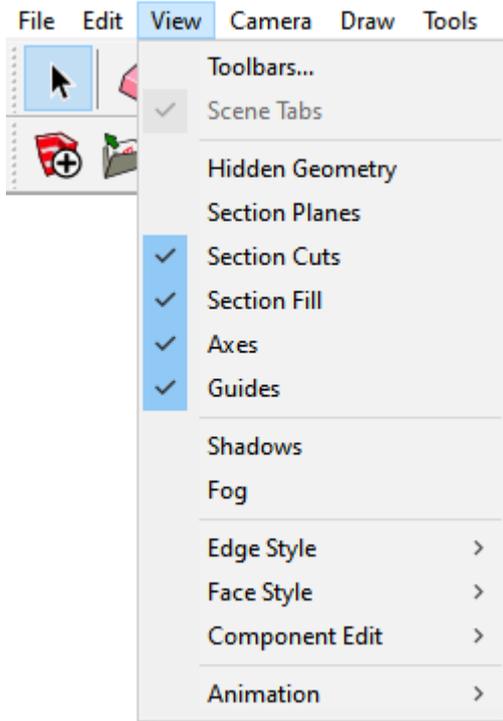
Camera
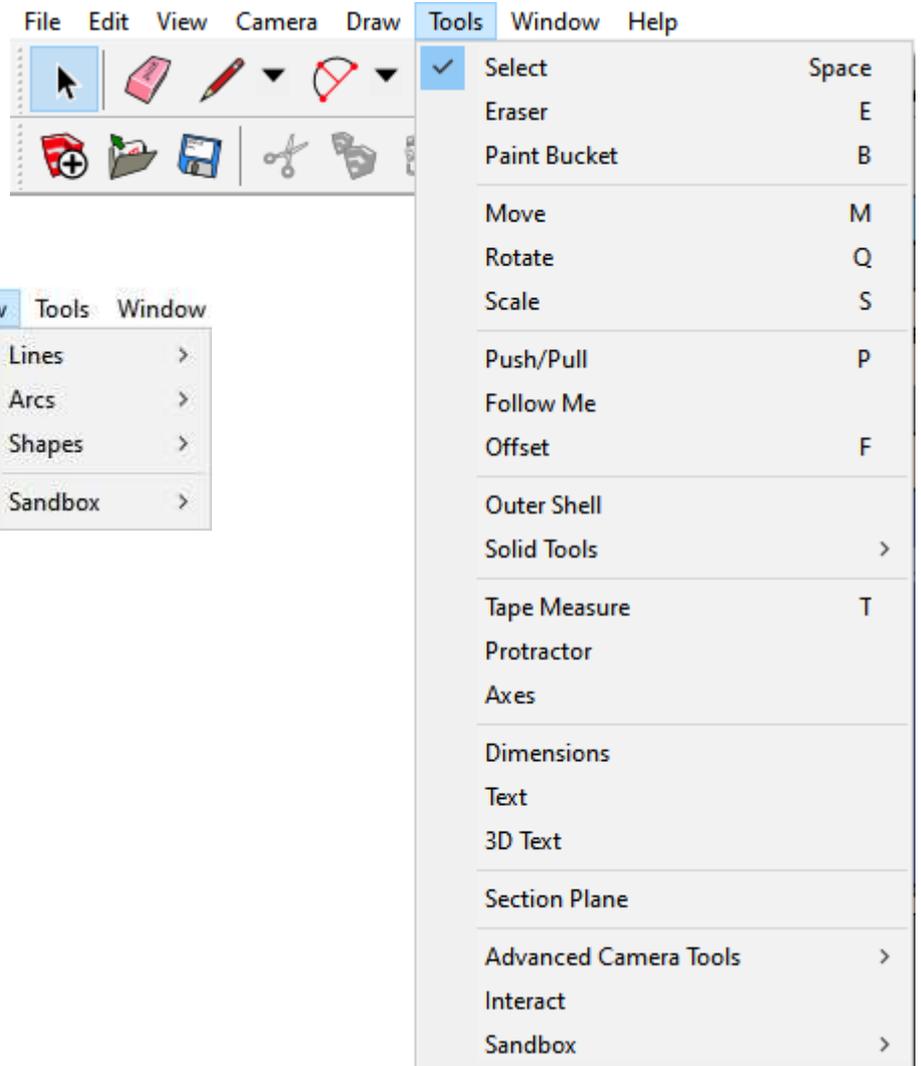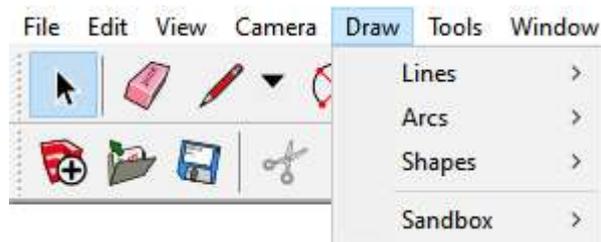
Draw

Tools

Window
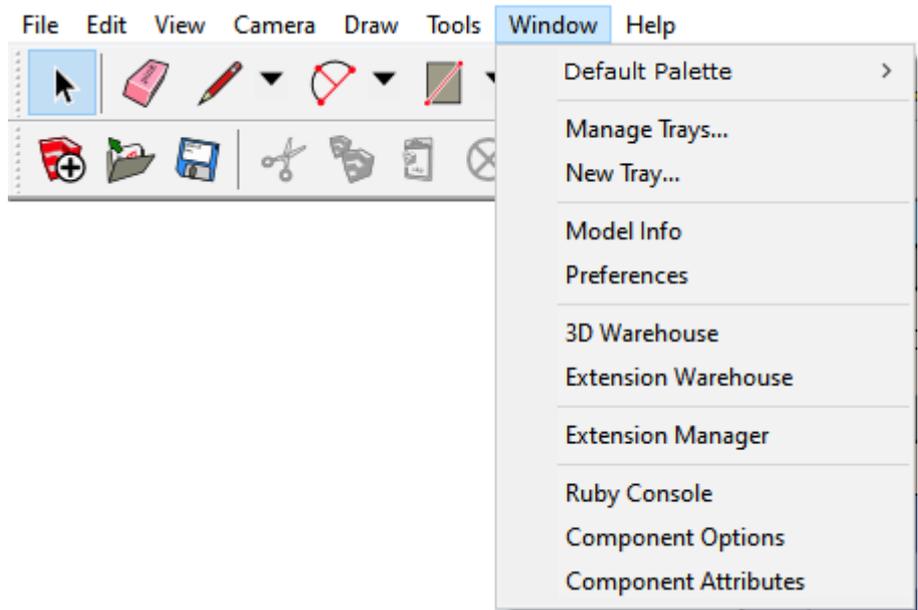
Help
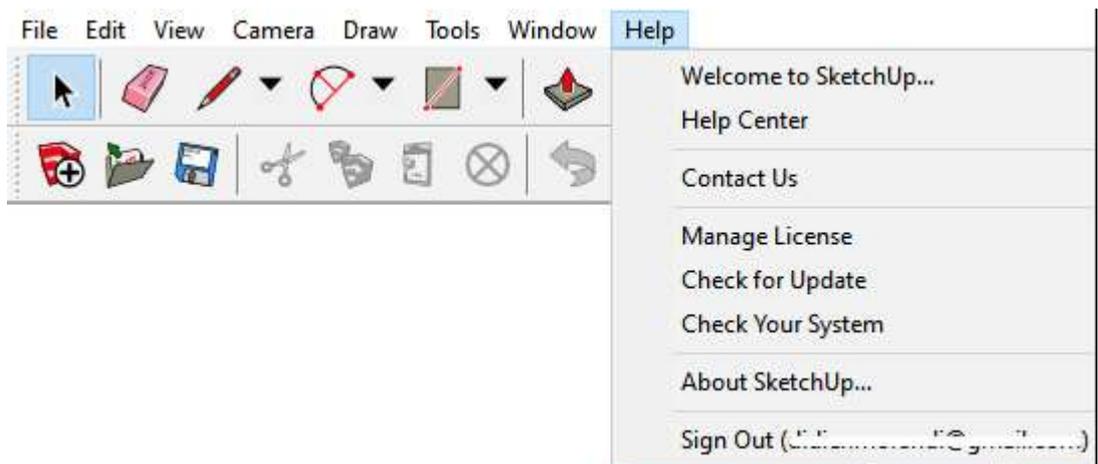
Drawing Axes

The Global axis lines in SketchUp show the origin of the X,Y and Z axis. They are essential to see the orientation of your model.

Determining which way is up, down, east, west, north, and south

Each axis has a solid line on one side of the origin and a dotted line on the other side of the origin. Following is an explanation for each line that make up the origin:

- The solid blue line leads up from the origin.
- The dotted blue line leads down from the origin.
- The solid red line leads east from the origin.
- The dotted red line leads west from the origin.
- The solid green line leads north from the origin.
- The dotted green line leads south from the origin.
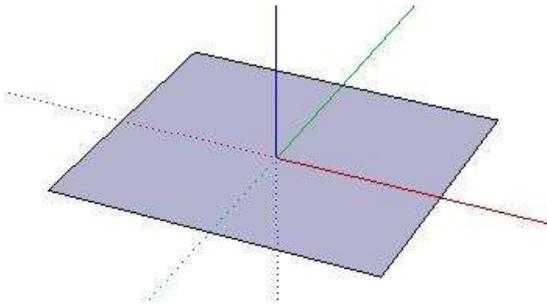
A Real-World 3D understanding of the axis is important when you are placing models in Google Earth[3] or casting realistic shadows.

Moving the Global Axes

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. Context-click on the drawing axes. The drawing axes context menu is displayed.
2. Select *Move* from the context menu. The *Move Sketching Context* (Microsoft Windows) or *Move Axes* (Apple macOS) dialog box is displayed.

   The following image shows the *Move Sketching Context* dialog box on Microsoft Windows:

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[3] See this video for more : https://help.sketchup.com/en/sketchup/viewing-your-model-google-earth
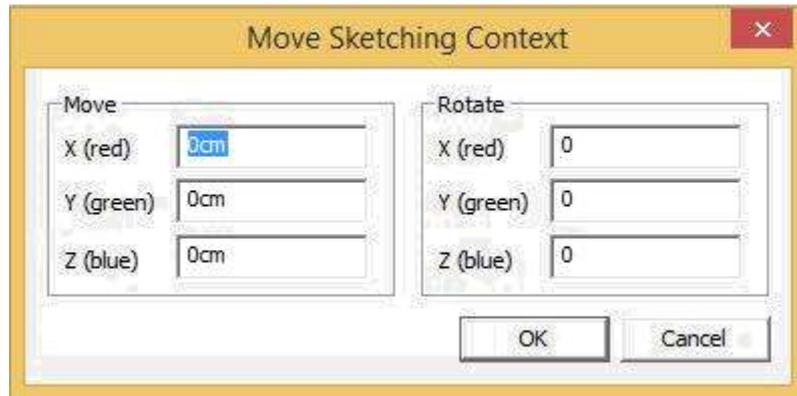
The following image shows the *Move Axes* dialog box on Apple macOS:
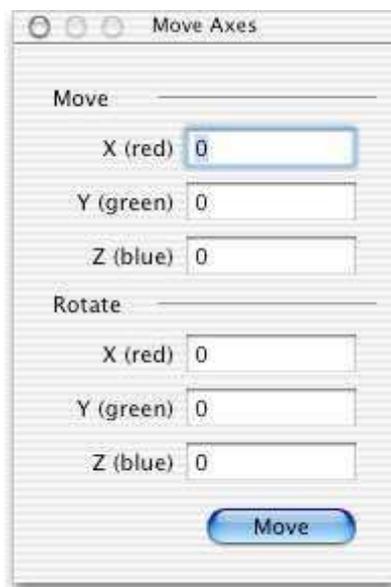
3. Specify displacement and rotation values in the units specified under the *Units* panel of the *Model Info* dialog box.
4. Click the **OK** button.

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 the *View menu*. You can

also hide the drawing axes from the drawing axes *Context menu* (when the drawing axes are visible).

Context Menu

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

The following image shows a context menu for a Face entity.
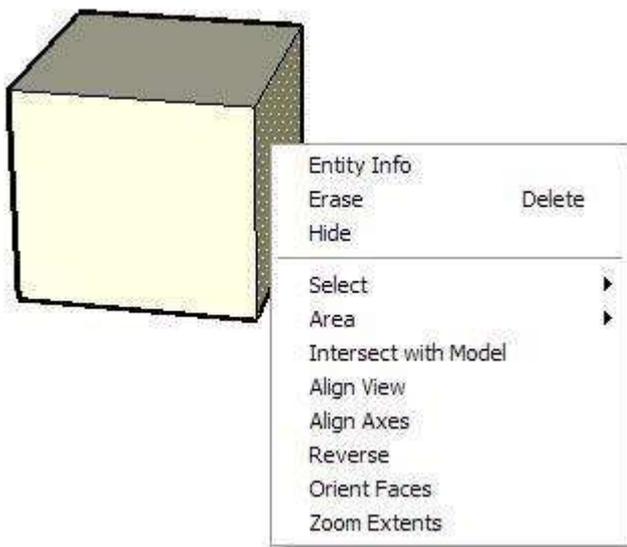
Context menus are invoked on entities by selecting one or more entities, holding down the Control key, and clicking the mouse button (single button mouse). Users with three-button or scroll-wheel mice can select one or more entities and click the right mouse button to invoke a context menu. This operation is referred to as a *context-click*.

Context menus are invoked on user interface components, such as a dialog box, by clicking the right mouse button while the cursor is on the component (three button mouse) or holding down the Control key and clicking the mouse button (single button mouse).

Inference

SketchUp features a 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.

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 pausing your mouse cursor over the particular location that you want SketchUp to infer from. When the visual cue appears, SketchUp will briefly prioritize that alignment as you continue drawing.

To encourage SketchUp to create a parallel line to match the length of 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 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 visual cue displays.
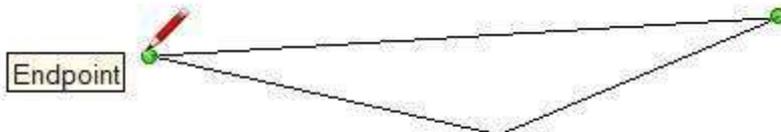
5. Move the Line tool to the approximate location where the end of the third line should appear. A dotted line, with the color of the corresponding axes (green) will follow the Line tool to indicate you

are directly inline with the start point of the first line.
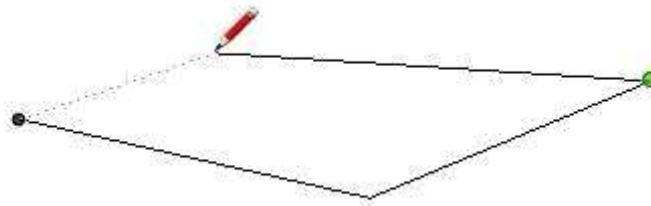
6. Click when the line you are drawing is indicated by an axes color (red) and is directly perpendicular to the green dotted line.

Note: Hover over any line to create a line parallel to that line. Refer to *Inference Types* for further information.

Locking an Inference

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, press hold 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.
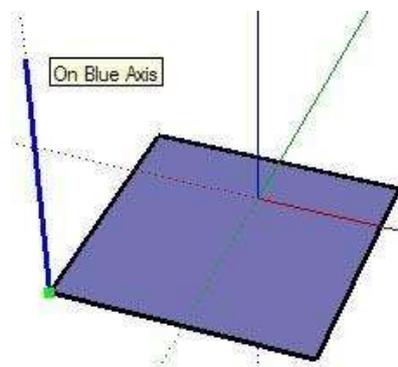
Any of the inference conditions may be locked; along an axis direction,

along an edge direction, on a face, from a point, parallel or perpendicular to an edge, and so on.

Forcing an Inference Direction

You can force SketchUp to infer parallel to a specific axis by pressing one of the following keys while using the *Line* tool, *Move* tool, or *Tape Measure* tool:

- Right arrow key = red axis
- Left arrow key = green axis
- Up arrow key = blue axis

Note: Down arrow locks to what direction has been focused to.

Dialog Boxes

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).

Resizing dialog boxes

Some SketchUp dialog boxes, such as the **Components 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.

Snapping dialog boxes to other user interface components

Some dialog boxes, such as the **Styles Browser** 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 > Styles** 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.

Principal Tools

- Select Tool
- Make Component Tool
- Paint Bucket Tool
- Eraser Tool

Select Tool

Select entities to modify when using other tools or commands.

Tool Operation

Click on an entity.

Modifier Keys

- Ctrl = Add an entity to set of selected entities
- Shift+Ctrl = Subtract an entity from set of selected entities
- Shift = Toggle whether an entity is within set of selected entities
- Ctrl+A = Select all visible entities in model

Select tool introduction

Use the *Select* tool 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 *Getting Started, Large Tool Set,* and *Principal* toolbars (Microsoft Windows), the *Tool Palette* (Apple macOS) or from the *Tools* menu.

Keyboard Shortcut: Spacebar

Selecting multiple entities

There are multiple ways to select multiple entities in SketchUp. These ways are:

- Selecting multiple entities with a selection box.
- Selecting connected entities using rapid mouse clicking.
- Selecting connected entities using the Select context-menu item.

Selecting multiple entities with a Selection box

A selection box is an expandable temporary box used to select multiple

entities. Selection box selections are useful when you want to perform a single operation on several connected or disconnected entities (the selection set).

To select multiple entities:

1. Choose 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.
3. Drag the mouse to expand the selection box over the elements you want to select.

- 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.
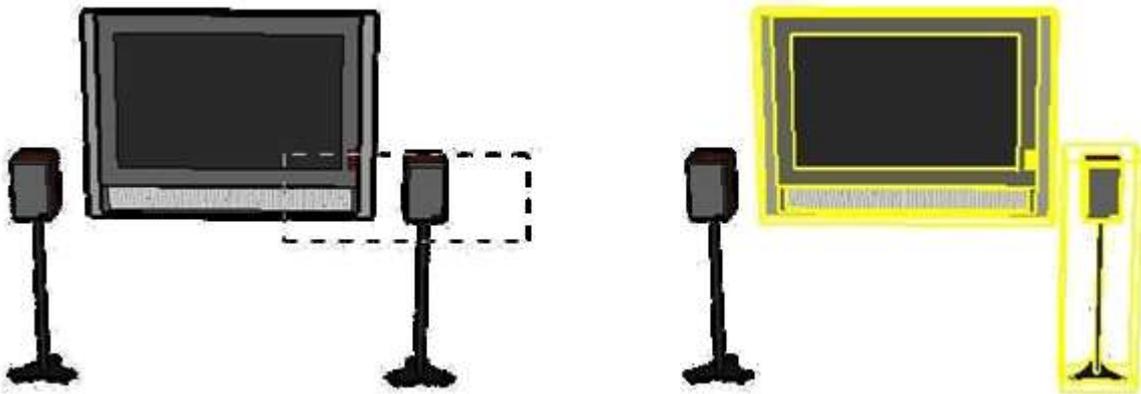
- Clicking to the left-side of the entities and dragging right, called a *window selection*, selects only those elements completely within the selection rectangle. The following image shows a left-to-right selection selecting one component because only one component (the left speaker) is completely within the bounds of the selection box.
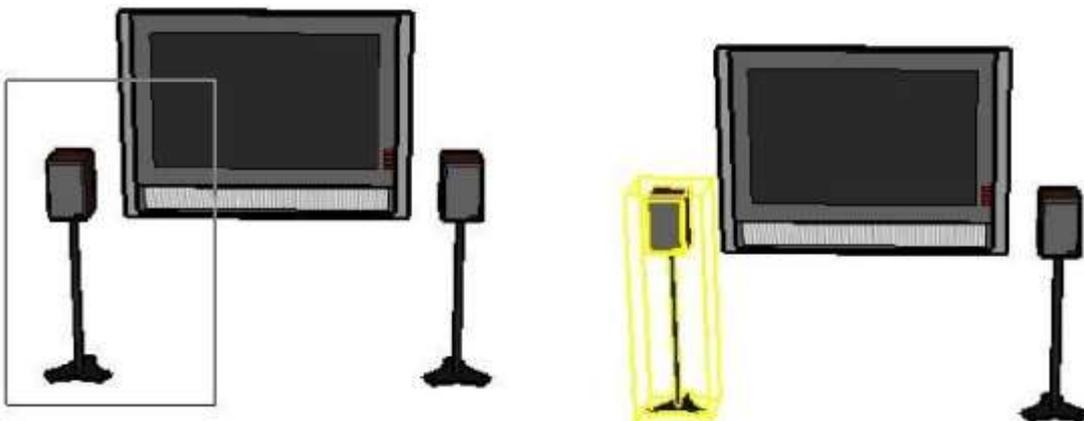
4. Release the mouse button when all of the elements are either partially included (right-to-left selection) or fully included (left-to-right selection) in the selection box.

Selecting connected entities using rapid mouse clicking

Rapidly clicking the mouse button will select one or more additional connected entities.

To Select a face and its bounding edges:

1. Choose the **Select** tool. The cursor will change to an arrow.
2. Double-click on a face to select the face and all of its bounding edges. The selected entities are highlighted.

To select just a face and an edge:

1. Choose the **Select** tool. The cursor will change to an arrow.
2. Double-click on an edge to select the connected face. The selected entities are highlighted.

To select all entities connected to a single entity:

1. Choose the **Select** tool. The cursor will change to an arrow.
2. Triple-click rapidly on any entity, in a set of connected entities, to select all of the connected entities. For example, if you triple-click a face in a cube, the entire cube is selected. Selected entities are highlighted.

Selecting connected entities using the Select context-menu item

Use the *Select* context-menu item to select entities based on their specific relationship to the currently selected entity. To use the Select context-menu item:

1. Select the **Select** tool ( ). The cursor will change to an arrow.
2. Context-click[4] on a single entity, such as an edge or face. The entity's context-menu appears.
3. Select the **Select** menu item. A sub-menu appears.
4. Select one of the **Select** sub-menu items:

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[4] aka *right-click*.

- Select **Bounding Edges** if you want to select all the bounding edges of a selected face.
- Select **Connected Faces** to select all of the faces connected to the selected entity.
- Select **All Connected** to select all entities connected to the selected entity (this is identical to performing a triple-click on the entity).
- Select **All on same Layer** to select all the entities on the same layer as the selected entity.
- Select **All with same Material** to select all the entities on with the same material as the selected entity.
- Select **Invert Selection** to deselect a selected element and select all elements in the model which were not selected.

Adding and subtracting from a 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 **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) 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 arrow with a plus and minus signs) 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 **Shift** and **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) 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 a 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 the selection set using the mouse

You can automatically add to a selection set by clicking the mouse button (while in 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.
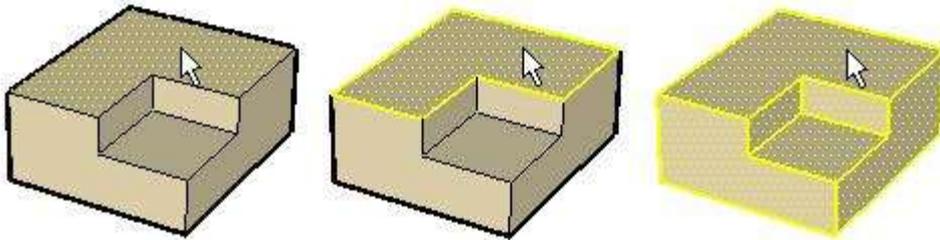
> Tip: Use a context-click to invoke the context menu for an entity. Many context menus have a Select submenu allowing you to expand a selection using one of the following commands: *Bounding Edges, Connected Faces, All Connected, All on same layer,* and *All with same material*.

Selecting 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.

> Tip: To select or unselect all geometry, choose **Edit > Deselect All**, press **Ctrl+T** (Microsoft Windows) or **Shift+Command+A** (Apple macOS). Alternatively, click on any empty space in the drawing area to de-select all currently selected entities.

Make Component

Make a component from the selected entities.

For more on Components read this[5].

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[5] https://help.sketchup.com/en/working-components-sketchup

Paint Bucket Tool

Assign materials and colors to entities.

Tool Operation

1. Select a materials library using drop down list in *Materials Browser*.
2. Select a material from *Materials Library*.
3. Click on faces to paint.

Modifier Keys

- Shift = Paint all faces with matching materials
- Ctrl = Paint all connected faces with matching materials
- Shift+Ctrl = Paint all faces on the same object with matching materials
- Alt = Sample material for painting

Paint Bucket tool introduction

Use the **Paint Bucket** tool 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 (Microsoft Windows), the *Tool Palette* (Apple macOS), or by choosing *Paint Bucket* from the *Tools* menu.

The **Paint Bucket** tool is separate from the **Apple Color Picker** used to select colors and materials (Apple macOS).

Keyboard Shortcut: B

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. In order to see the painted effect applied to edges, you'll need to display the edge color by material. To do so, open the *Styles Browser* (under the Window menu), choose the **Edit** tab, and select the **Edge Settings** button. Finally, choose **By material** from the Color menu.

Fill options

The **Paint Bucket** tool can be used with one or more keyboard modifiers to perform various painting operations.

Element fill

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.

> Note: Selecting a number of entities with the Select tool and painting causes just the entities within the selection set to be painted.

Adjacent fill

Press and hold the **Ctrl** (Microsoft Windows) or the **Option** (Apple macOS) 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.
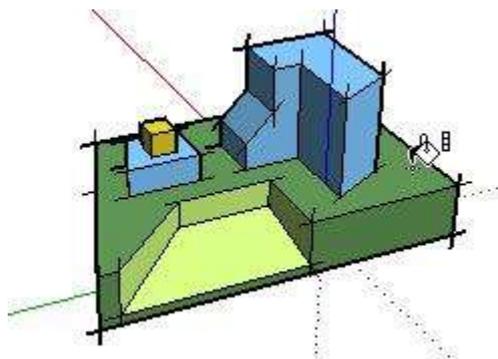
> Note: Selecting a number of entities with the Select tool and painting using the Ctrl (Microsoft Windows) or the Option (Apple macOS) modifier key causes just the entities within the selection set to be painted.

Replace

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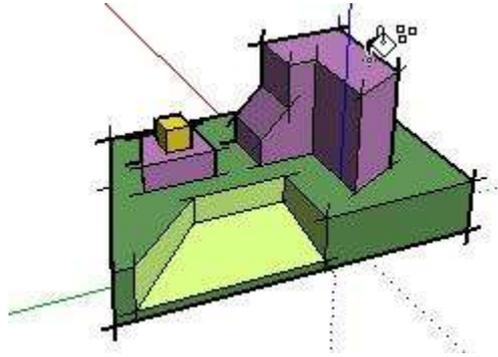
Note: Selecting a number of entities with the Select tool and painting using the Shift modifier key causes just the entities within the selection set to be painted.

Adjacent replace

Press and hold both the **Shift** and the **Ctrl** (Microsoft Windows) or the **Option** (Apple macOS) keys simultaneously while painting to only replace the material on the face within the confines of geometry that is physically connected to that face.

Sampling a material

Press and hold the **Alt** (Microsoft Windows) or **Command** (Apple macOS) 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** (Microsoft Windows) or **Command** (Apple macOS) key to return to the **Paint Bucket** tool. Paint the sampled material on a face.
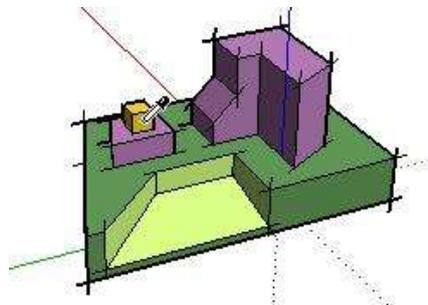
Note: The sampled material is placed in the Active Color Well of the Color Picker, where it can be painted on new entities, modified, or used as the basis of a new material (Apple macOS).

🖌 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 **Materials 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 Bucket** 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.
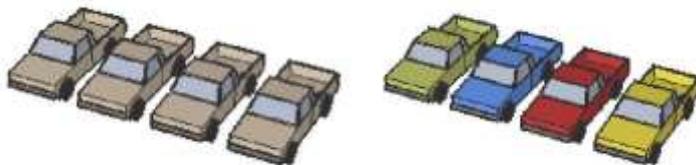
Note: Exploding a group or component assigns the object materials to any elements assigned the default material, thus making the material override permanent.

Eraser Tool

Erase, soften or smooth entities in the model.

Tool Operation

Click on entity to erase. Alternatively, hold down mouse button and drag over entities. All entities are erased when mouse button is released.

Modifier Keys

- Shift = Hide entities
- Ctrl = Soften and smooth entities
- Shift+Ctrl = Unsoften and unsmooth entities

Eraser tool introduction

Use the **Eraser** tool to delete entities. The **Eraser** tool can also be used to hide and soften edges. Activate the **Eraser** tool from the **Principal** toolbar (Microsoft Windows), the **Tool Palette** (Apple macOS), or by selecting **Eraser** from the Tools menu.

Keyboard Shortcut: E

Erasing entities

As mentioned previously, the **Eraser** tool is used to erase entities in the drawing area. Note, the **Eraser** tool does not allow you to erase faces (faces are erased after their bounding edges are erased). To erase entities:

1. Select the **Eraser** tool (   ). The cursor changes to an eraser with a small box.
2. Click on an entity to erase it. 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 lines

Press and hold the **Shift** key and use the **Eraser** tool to hide lines (instead of erasing lines).

 Softening or unsoftening edges

Press and hold the **Ctrl** (Microsoft Windows) or **Option** key (Apple macOS) key to soften/smooth edges (instead of erasing entities). Press and hold the **Shift** and **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) keys simultaneously to unsoften/unsmooth edges.

Drawing Tools

- Line Tool
- Freehand Tool
- Rectangle Tool
- Rotated Rectangle Tool
- Circle Tool
- Polygon Tool
- Arc Tool
- 2 Point Arc Tool
- 3 Point Arc Tool
- Pie Tool

Line Tool

Draw edges from point to point.

Tool Operation

1. Click at starting point of line.
2. Move cursor.
3. Click at ending point of line.
4. (optional) Move cursor.
5. (optional) Click to create connected line.
6. (optional) Repeat step 4 to create connected lines, or return to starting point of first line to create a face.

Modifier Keys

- Shift = Lock line to the current inference direction
- up arrow, left arrow, right arrow = Lock line to specific inference direction (up=blue, left=green, right=red)

Line tool introduction

Use the **Line** tool to draw edges or line entities. Line entities can be joined to form a face. The **Line** tool can also be used to divide faces or heal deleted faces. Activate the **Line** tool from the toolbar / **Tool Palette** or from the **Draw** menu.

Keyboard Shortcut: L

✏ Creating a face

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. To create a face using the **Pencil** tool:

1. Select the **Line** tool (✏). The cursor changes to a pencil.
2. Click to set the starting point of your line.
3. Move the cursor to the ending point of your line. As you draw a line, the length is displayed dynamically in the *Measurements* toolbar.
4. Click to draw your line. This ending point can also be the starting point of another line.

5. Move the cursor to an ending point for the second line.
6. Click to draw your line. This ending point can also be the starting point of another line.
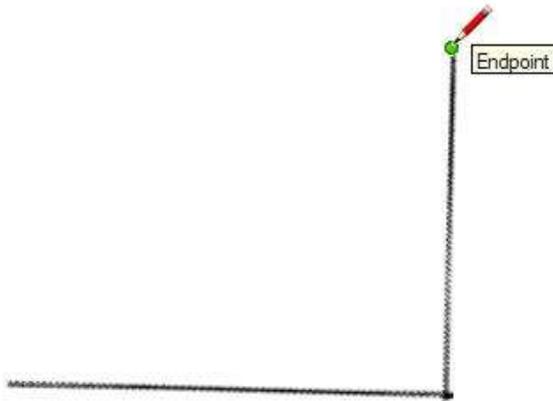
7. Move the cursor to the starting point of the first line. The tip of the Pencil cursor changes to a green circle and says "Endpoint."
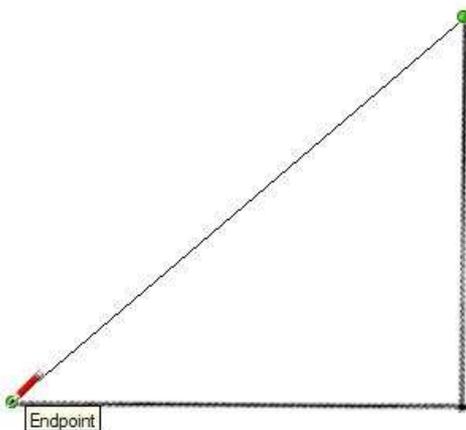
8. Click to draw your line. A face is created.
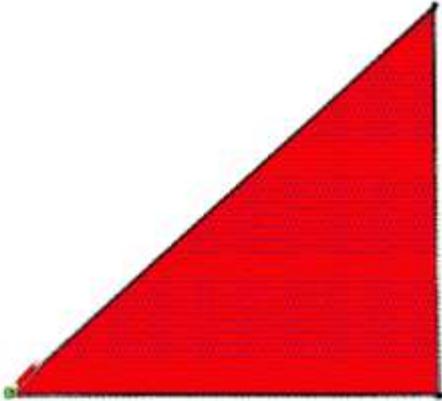
Tip: Set the display settings (**View > Face Style > Shaded**) to the "Shaded" or "Shaded With Textures" rendering style to clearly show new faces as they are created.

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.
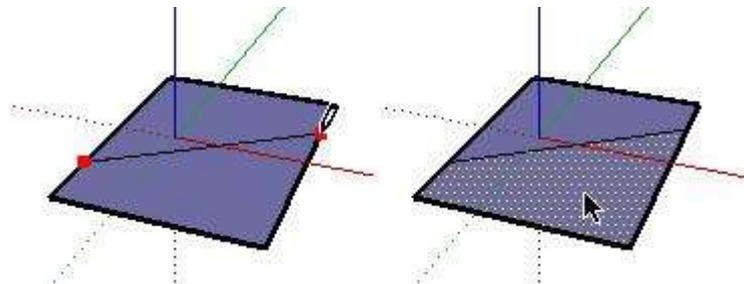
Splitting a line

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.
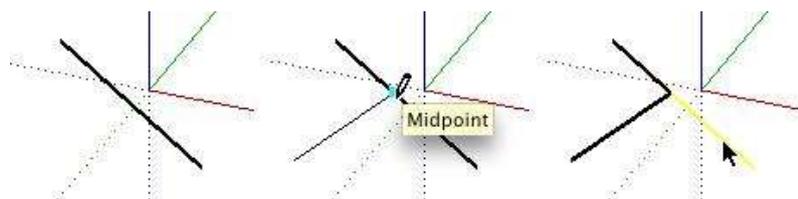
Select the original line to verify that the line has been split into two equal

segments.

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

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3. Move the cursor toward the center of the line to reduce the number of segments. Move cursor toward either end of the line to increase the number of segments.
4. Click on the line when the number of segments you would like is shown. The line will be divided into an equal number of joined line segments.

Creating precise lines

The *Measurements* toolbar displays the length of your line while you are drawing lines. You can also specify a line length value using the Measurements toolbar.

Entering a length value

The Measurements toolbar label indicates 'Length' after you place the starting point of a line. The following image shows the length value in the Measurements toolbar.

Type the length into the Measurements toolbar, after placing the starting point of the line, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) 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 Measurements toolbar.

Entering a 3D coordinate

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

Length [3',5',7']
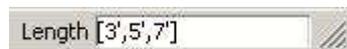
Entering an Absolute Coordinate30Type in the coordinates of a point in 3D space enclosed by brackets, such as [x, y, z], to get absolute coordinates relative to the current axes.

Length <1.5m,4m,2.75m>
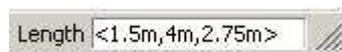
Entering a Relative Coordinate

Type the coordinate points enclosed by angle brackets, in the format , where x, y, and z values are relative distances from the starting point of your line.

Note: The exact format for Measurements toolbar 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.

Locking a line to the current inference direction

Press and hold the **Shift** key, while the line you are drawing is the specific

color of an axis, to lock drawing operation to that axis.

Locking a line to a specific inference direction

Press and hold either the up arrow, left arrow, or right arrow keys, where the up arrow equals blue, left arrow equals green, right arrow equals red, while drawing a line to lock the line to a specific axis.

Freehand Tool

Draw freehand lines by clicking and dragging.

Tool Operation

1. Click and hold at starting point of curve.
2. Drag the cursor to draw.
3. Release mouse button to stop.
4. (optional) End curve at starting point to draw closed shape.
5. Esc = Cancel operation.

Modifier Keys

Shift = Draw 3D Polyline entity

Freehand tool introduction

Use the Freehand tool to draw irregular hand-drawn 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 toolbar / Tool Palette or from the Draw menu.

Drawing 3D polyline entities

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. Press and hold the Shift key, before you begin drawing, to draw a 3D polyline.

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

Rectangle Tool

Draw rectangular faces from corner to corner.

Tool Operation

1. Click to set first corner.
2. Move cursor diagonally.
3. Click to set second corner.
4. Esc = Cancel Operation.

Modifier Keys

Shift = Lock rectangle to current inference direction

Rectangle tool introduction

Use the Rectangle tool to draw rectangular Face entities, specified by clicking at two opposite corners of the desired shape. Activate the Rectangle tool from the toolbar / Tool Palette or from the Draw menu.

Drawing a square

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

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. A diagonal dotted line will appear, along with a Square tool tip, 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.

4. Click to finish.

Tip: A dotted line and Golden Section tool tip 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.

Creating precise rectangles

A rectangle's dimensions dynamically appear in the Measurements toolbar as you draw. Specify exact length and width dimensions by typing them in the Measurements toolbar, and pressing **Enter** (Microsoft Windows) or **Return** (Apple macOS) either after the first corner is clicked or immediately after the rectangle is drawn.
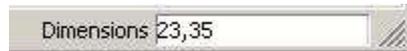
SketchUp will use the current document units setting if you only type in a numerical value. You can also specify either Imperial (such as 1'6") or Metric (such as 3.652m) units at any time, regardless the document units setting.

You can also type one dimension at a time in the Measurements toolbar. If you enter a value and a comma (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 value (,3'), only the second dimension will be changed.

Note: if you're using a non-English keyboard, you'll want to use a comma to indicate the decimal place and a semi-colon to separate the dimensions. For example, you might enter two sides of a rectangle as: 7,6m;4,3m.

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 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 the direction of an axis, a dotted inference line with a From Point tool tip will appear.
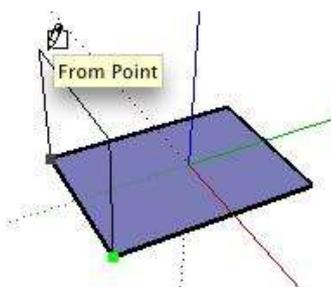
This tool tip indicates that you are aligned to that end point. You can also use a From Point inference to draw rectangles vertically or at non-orthogonal planes.

Rotated Rectangle Tool

Draw rectangular faces from three corners.

Tool Operation

1. Click to set first corner.
2. Move your cursor around the protractor to set the direction of the first edge.
3. Click to set second corner.
4. Move your cursor to set the length and angle of the second edge.
5. Click to set third and final corner.
6. Esc = Cancel Operation.

Modifier Keys

- Shift = Lock rectangle to current inference direction or protractor baseline
- Alt (after first click) = Lock drawing plane for first edge
- Alt (on a locked plane, after first click) = Set protractor baseline
- Alt (after second click) = Set protractor baseline

The *Rotated Rectangle Tool* can come in handy when you need to draw a rectangle that doesn't line up with the default red, green, or blue axes in SketchUp. Like the *Rectangle Tool*, you can create precise rectangles, squares, and also utilize inferencing. In this article, we'll cover the process of creating a rotated rectangle and creating a precise rotated rectangle.

Tip: You should be familiar with SketchUp's Rectangle Tool before working with the Rotated Rectangle tool.

Creating a rotated rectangle

1. Click the black arrow in the **Shape Tools** option in the toolbar. This will open the Shape Tools menu. Select the Rotated Rectangle tool ( ). Alternatively, you can click *Draw > Shapes > Rotated Rectangle*.
2. Select the **Rotated Rectangle** option.
3. Click once to set the first corner of your rectangle.
4. Move your cursor around the protractor to set the direction of the first edge.
5. Click to set second corner.
6. Move your cursor to set the length and angle of the second edge.

Experiment with this behavior a bit to get the hang of it. You can always Undo and try again.

7. Click to set third and final corner.

Creating a precise rotated rectangle

1. Drawing the first edge: After you click once to set the first corner of a rotated rectangle, move your mouse cursor around the screen. You will see the corner stay in place and an edge will remain attached to your mouse cursor. You can also click and drag the first click to set the baseline. See "Setting the baseline" below for more information. Note that you have precise control over the length and the direction of the first edge.
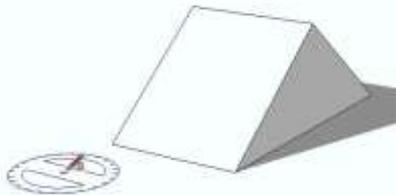
   a. Inferencing: Move your mouse cursor so that the first edge parallels one of the default axes directions: red, green, or blue; the edge will turn red, green, or blue. Press and hold the **Shift** key and the edge will turn thicker. The edge will now only draw along that default axes direction. Alternatively, you can also press the **Right Arrow** key to lock the edge to the red axes, the **Left Arrow** key to lock the green axes, and the **Up Arrow** key to lock the blue axes. Other *standard inferences*, such as parallel or perpendicular edges, are also accessible when drawing the first edge.

   b. Setting the baseline: Pressing the **Alt** key (Microsoft Windows) or the **Command** key (Apple macOS) initially does two things: lock the drawing plane and create a baseline that shows the direction of the first edge at the time you lock the baseline. If you keep the first edge on the baseline, you can press and hold the **Shift** key to lock the edge to the baseline. Press the **Alt** key (Microsoft Windows) or the **Command** key (Apple macOS) to reset the baseline. This is especially helpful if you need to lock the first edge to a specific direction.

   c. Setting the angle or length: *Before* you click a second time to complete the edge, you can type the exact angle, length, or both in the Measurements toolbar and press **Enter** (Microsoft Windows) or **Return** (Apple macOS).

> Note: Typing the Angle is only available after you set the baseline - initially you can only define the length.

For example to set the angle to 45 degrees, type "45, ". Notice the empty value after the comma.

> Note: If you type the angle in the Measurements toolbar, this will lock the direction of the rectangle. Tap the Shift key to unlock.

Likewise, to set the distance to ten feet, type ,10'. Again, notice the empty value before the comma. Finally, to set the angle and the length, enter the degrees first, then a comma, then the length of the edge. For example, 45,10' would create a 45 degree angle that is ten feet long.

2. Drawing the second edge: After you set the length and direction of the first edge of a rotated rectangle, you can draw a second edge which will draw the completed rectangle.
3. Setting the angle or width: Move your mouse around after you set the first edge. The outline of a rectangle will be connected to your mouse cursor with additional guides that indicate the drawing plane and angle. Look at the Measurements toolbar, and you will notice both an angle and a distance. To set a precise angle and distance, type the angle followed by a comma then type the length of the second edge. For example, typing 90, 5' will draw a rectangle that is perpendicular to the plane of the first edge and five feet long. Press **Enter** (Microsoft Windows) or **Return** (Apple macOS). Just like when you set the first edge, you can type just the angle or the width with empty values before/after the comma.

> Note: If you type the angle in the Measurements toolbar, this will lock the direction of the rectangle. Tap the Shift key to unlock.

- Setting the baseline: Just like when you draw the first edge, you can reset the baseline by pressing the **Alt** key (Microsoft Windows) or **Command** key (Apple macOS).
- Inferencing: Just like drawing the first edge, you can move your mouse cursor so that the second edge parallels one of the default axes directions: red, green, or blue; the edge will turn red, green, or blue. Press and hold the **Shift** key and the edge will turn thicker. The edge will now only draw along that default axes direction. Alternatively, you can also press the **Right Arrow** key to lock the edge to the red axes, the **Left Arrow** key to lock the green axes, and the **Up Arrow** key to lock the blue axes. Other *standard inferences*, such as parallel or perpendicular edges, are also

accessible when drawing the second edge.

Note: Both the Square and Golden Section inferences are also available to rotated rectangles.

Note: if you're using a non-English keyboard, you'll want to use a comma to indicate the decimal place and a semi-colon to separate the values in Measurements toolbar. For example, you might enter the angle and length of the second edge as 43,2;8,2m to get an angle that's 43,2 degrees and 8,2 meters long.

Circle Tool

Draw circles from center point to radius.

Tool Operation

1. Click to place center point.
2. Move cursor away from center point to define radius.
3. Click to finish circle.
4. Esc = Cancel operation.

Modifier Keys

Shift (before first click) = Lock circle to current orientation

Circle tool introduction

Use the Circle tool to draw Circle entities. Activate the Circle tool from the toolbar / Tool Palette or from the Draw menu.

Keyboard Shortcut: C

Locking a circle to its current orientation

Press and hold the **Shift** key, before you begin drawing a circle, to lock drawing operation to that orientation.

Creating Precise Circles

The Measurements toolbar displays the radius after setting the center point of a circle. Use the Measurements toolbar to enter an exact radius and number of segments.

Specifying a Radius

The Measurements toolbar's label indicates "Radius" after you place the center point of the circle. Type the radius size in the Measurements toolbar, after placing the center point, and press the **Enter** (Apple macOS) or **Return** (Apple macOS) key. You can perform this action either during or immediately following the creation of the circle. For example: 24r or 3'6"r or 5mr.

Note: The Circle Tool will snap to any previously entered radius within the Measurements toolbar.

Specifying the Number of Sides

The Measurements toolbar's label indicates "Sides" when the Circle Tool is initially activated. Specify the number of sides in the Measurements toolbar, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key, before clicking to set the center point of the circle. For example: *100*.

You can also specify the number of sides in a circle immediately after the creation of the circle. Type the number of sides in the Measurements toolbar, followed by the letter s, and press the38**Enter** (Microsoft Windows) or **Return** (Apple macOS) key. For example: 20s. This number will be applied to any future circles.

Note: The default number of segments for a circle is 24.

Polygon Tool

Draw N-sided polygons from center point to radius.

Tool Operation

1. Click to place center point.
2. Move cursor away from center point to define radius.
3. Click to finish polygon.
4. Esc = Cancel operation.

Modifier Keys

- Shift (before first click) = Lock polygon to its current orientation
- Ctrl (before second click) = Toggle between inscribed and circumscribed radii of polygon

Polygon tool introduction

Use the Polygon tool to draw regular Polygon entities. Activate the Polygon tool from the toolbar / Tool Palette or from the Draw menu.

Locking a polygon to its current orientation

Press and hold the Shift key, before you begin drawing a polygon, to lock drawing operation to that orientation.

Drawing precise polygons

The Measurements toolbar displays the radius after setting the center point of a polygon. Use the Measurements toolbar to enter an exact radius and number of segments.

Specifying a Radius

The Measurements toolbar's label indicates 'Radius' after you place the center point of the polygon. Type the radius size in the Measurements toolbar, after placing the center point, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. You can perform this action either during or immediately following the creation of the polygon. For example: 24r or 3'6"r or 5mr.

Tip: Tap the **Ctrl** key (Microsoft Windows) or **Option** key (Apple macOS) to toggle between an *inscribed* or *circumscribed* polygon.

Note: The Polygon tool will snap to any previously entered radius within the Measurements toolbar.

Specifying the Number of Sides

The Measurements toolbar's label indicates 'Sides' when the Polygon tool is initially activated. Specify the number of sides in the Measurements toolbar, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key, before clicking to set the center point of the polygon. For example: 10.

You can also specify the number of sides in a polygon immediately after the creation of the polygon. Type the number of sides in the Measurements toolbar, followed by the letter s, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. For example: 6s. This number will be applied to any future polygons.

Arc Tool

Draw arcs from center and two points.

Tool Operation

1. Click to define arc center.
2. Move cursor to define first arc point or enter radius.
3. Click to set first arc point.
4. Move cursor around the protractor guide or enter angle.
5. Click to set second arc point.
6. Esc = Cancel operation.
7. Tip: Click Window > Model Info > Units > Angle Units to change snapping angle.

Modifier Keys

- Shift (before first click) = Lock arc to current orientation
- Shift (after first click) = Lock tool to current inference direction

Arc tools introduction

Use the Arc tools to draw Arc entities: arcs comprised of multiple connected line segments (which can be edited as a single arc). Activate any of the Arc tools from the toolbar / Tool Palette or from the Draw menu. There are four Arc Tools; *Arc, 2-Point Arc, 3-Point Arc, and Pie* tools which all draw arcs but in different ways.

Keyboard Shortcut for 2-Point Arc: A

Drawing a half-circle using the 2-Point Arc or 3-Point Arc tools
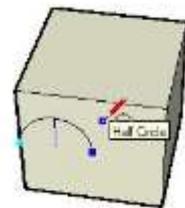
A 2-Point arc temporarily snaps to a half-circle as you pull out a bulge distance. Similarly, when drawing a 3-Point arc, the arc will snap to a half circle before setting your third point. Watch for the half-circle inference tool tip indicating when your arc is a half-circle.

Drawing an open half-circle using the Arc tool

Open half-circle entities consist of three parts: the center, the starting point and the ending point. To draw an open arc:

1. Select the Arc tool (icon). The cursor changes to a pencil with an

open arc and a half-circle protractor is drawn to show the orientation of a drawing plane.

2. Click to place the center of your arc. You have locked the orientation of the drawing plane and a full-circle protractor is drawn.
3. Move the cursor to the starting point of your arc.
4. Click to place the starting point of your arc. A straight dotted line is created which represents the radius of your arc.
5. Move the cursor to the ending point of your arc, for a half-circle use the inference locking to stop at 180 degrees.
6. Click to place the ending point of your arc. An open half-circle is created.

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

Drawing a closed half-circle using the Pie tool

Closed half-circle entities consist of five parts: the center, the starting point and the ending point and two edges (from center to starting and ending points).

To draw a closed arc:

1. Select the Pie tool ( ). The cursor changes to a pencil with a closed arc and a half-circle protractor is drawn to show the orientation of a drawing plane.
2. Click to place the center of your arc. You have locked the orientation of the drawing plane and a full-circle protractor is drawn.
3. Move the cursor to the starting point of your arc.
4. Click to place the starting point of your arc. A straight solid line is created (you just established a radius for your arc and created the first edge of the pie).
5. Move the cursor to the ending point of your arc, for a half-circle use the inference locking to stop at 180 degrees.
6. Click to place the ending point of your arc. An closed arc is created with two edges.

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

The arc Radius value can be specified using the Measurements toolbar after you place the center point of your arc. Angle and number of segments can be specified using the Measurements toolbar immediately after an arc is drawn. See *Creating Precise Arcs* for more information on manipulating arcs using the Measurements toolbar. When you draw an

arc, by default, the arc will be drawn with a fixed number of line segments that will grow or shrink in length depending on the completeness of the arc. You can change the number of segments in the arc by typing a new value for "Sides:" in the Measurements toolbar. For example, typing 10s will create an arc with 10 line segments.

Arc tools allow you to set the number of **Circle Sides** as well, which tells SketchUp how many line segments you'd like to see if you draw a full circle. When you specify the circle sides, the line segments stay roughly the same length and the number of line segments depends on the completeness of your arc.

The following image shows a 1/4 Arc (Red), 1/2 Arc (Green), 3/4 Arc (Blue), and a Full Circle (Black). The top shape was created using the Pie tool and set to have six Circle Sides.
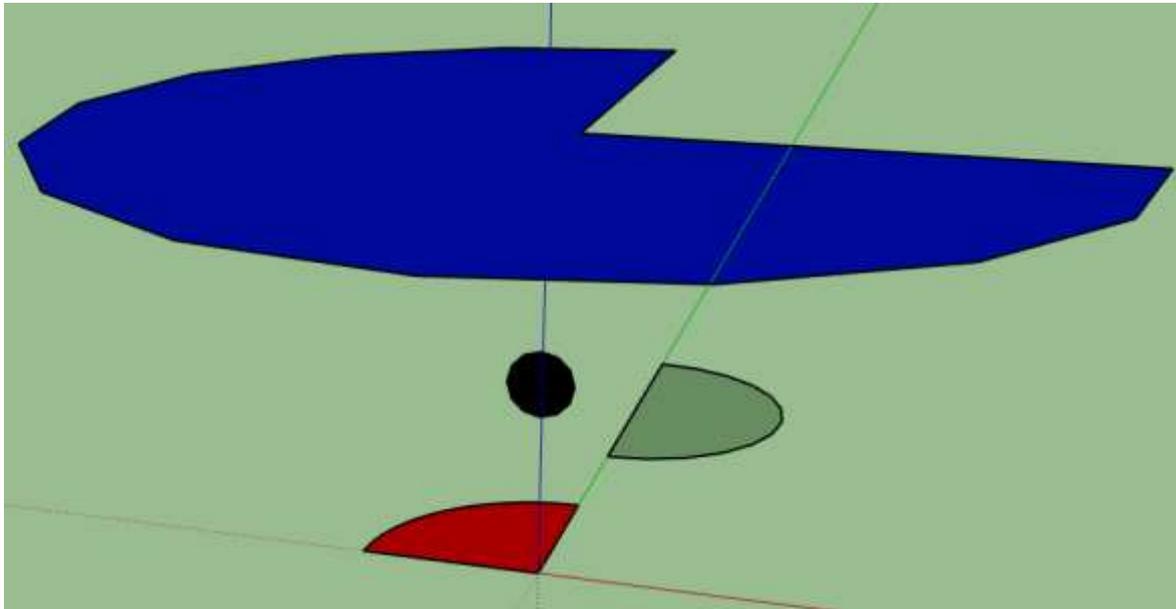
To specify the number of Circle Sides:

1. Select the Arc tool.
2. Without clicking anywhere else, type the number of Circle Sides in the Measurements toolbar, followed by the letter "c". For example: 20c
3. Press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key.

If you want to return to drawing a set number of line segments (vs Circle Sides), type the number of sides in the Measurements toolbar followed by the letter "s". For example: 20s

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Note that you can also perform this action either during or immediately following the creation of the arc.

⌒Creating precise arcs

Use the Measurements toolbar to enter exact lengths for the chord length, bulge distance, radius, number of segments, or angle.

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

For 2-Point Arc tool: The Measurements toolbar's label indicates 'Length' after you place the starting point of an arc. Type the chord length into the Measurements toolbar, after placing the starting point of the arc, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. Specify a negative value, such as *-6.5"*, to indicate that you want the length to apply in an opposite direction to the current drawing direction.

For the 3-Point Arc tool: Typing a Length value in the Measurements toolbar specifies the *linear* length between the first point and the second point.

Specifying a bulge distance

For 2-Point Arc tool: The Measurements toolbar's label indicates 'Bulge' after you place the ending point of an arc. Type the bulge length in the Measurements toolbar, after placing an ending point, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. You can also enter the bulge distance after the creation of the arc as long as 'Bulge' is displayed as the Measurements toolbar label. Negative bulge values can also be used to create an arc in an opposite direction to the current drawing direction.

After drawing the arc, select the Move tool, then click and drag the mid-point of the arc to adjust the Bulge. You can type a specific value in the Measurements toolbar.

Specifying a radius

For 2-Point Arc: You can specify an arc radius instead of a bulge distance. Type the desired radius in the Measurements toolbar, followed by the letter r and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) Key. You can perform this action either during or immediately following the creation of the arc. For example: 24r or 3'6"r or 5mr. 44*For Arc and Pie tools:* You can specify an arc radius in the Measurements toolbar after your first click. You don't need to include the

letter r.

After drawing the arc, select the arc, then context click on the arc and select **Entity Info.** You can adjust the Radius field in Entity Info as needed.

Specifying the number of segments

You can also specify the number of segments in an arc. Type the number of segments in the Measurements toolbar, followed by the letter s, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. You can perform this action either during or immediately following the creation of the arc. For example: 20s.

Specifying the dynamic number of segments

You can also specify the dynamic number of segments (Circle Sides) in an arc. When specifying the dynamic number of segments, the number of segments in an arc will be proportionate to the arc central angle. Type the number of segments in the Measurements toolbar, followed by the letter c, and press the Enter (Microsoft Windows) or Return (Apple macOS) key. You can perform this action either during or immediately following the creation of the arc. For example: 20c With this setting a 90 degree arc will have 5 (20 * 90/360) segments.

Specifying an angle (for 3-Point Arc, Arc, and Pie tools only)

The Measurements toolbar's label indicates 'Angle' after your second click when drawing the arc. Type the angle value (in degrees) in the Measurements toolbar after placing a starting point, and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. You can also enter the angle value after the creation of the arc as long as 'Angle' is displayed as the Measurements toolbar label. Negative angle values can also be used to create an arc in an opposite direction to the current drawing direction.
Specifying the axis of rotation (for Arc and Pie tools only)

Just like the Rotate and Protractor tool, you can click and drag the first point to define the axis of rotation. This is especially helpful if you need to rotate on an axis that isn't on the red, green, or blue planes.

2-Point Arc Tool

Draw arcs from point to point with bulge.

Tool Operation

1. Click to set start point of arc.
2. Move cursor.
3. Click at ending point of arc or enter value.
4. Move cursor perpendicular to line to set bulge distance or enter value.
5. Click to finish arc.
6. Esc = Cancel operation.

Modifier Keys

Shift = Lock tool to current inference direction

Drawing tangent arcs and fillets

The 2-Point Arc tool and 3-Point Arc tool will display a cyan tangent arc when you set your first point on an unconnected endpoint or on an edge. If you trace the second point of the arc along another edge, the arc will turn magenta when it is tangent to both edges. Double-click to finish the arc and create a fillet.

Tip 1: After creating your first fillet, you can double-click at a point on another face near the corner of two edges. An arc/fillet of the last used radius will be drawn on the face at that corner.
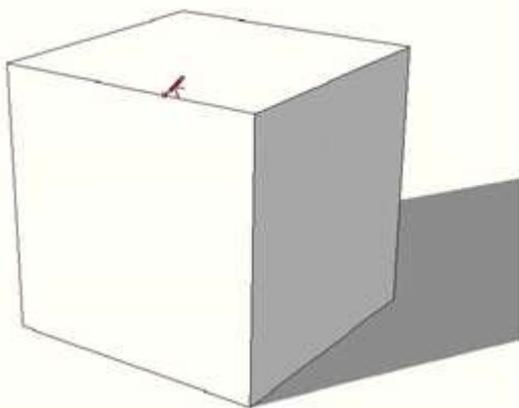
Tip 2: Similar to Tip 1, after creating your first fillet, you can double-click at a point on a 2D free-standing face (i.e. a face with only two connected edges) near the corner of two edges; the geometry outside the arc will be removed and make a rounded corner on the face. This also works on the very first fillet, too, by double-clicking the second endpoint after you see

the magenta arc. To prevent SketchUp from removing the geometry, press the Alt key (Microsoft Windows) or the Command key (Apple macOS) when you double-click on the face.

3-Point Arc Tool

Draw arcs through three points on the arc circumference.

Tool Operation

1. Click to set start point of arc.
2. Move cursor away from start point.
3. Click to set second point. The arc will always go through this point.
4. Move cursor to the end point. An angle will appear in the Measurements box, and you can type a precise value.
5. Click to finish arc.
6. Esc = Cancel operation.

Modifier Keys

Shift = Lock tool to current inference direction for any point.

Pie Tool

Draw closed arcs from center and two points.

Tool Operation

1. Click to define pie's center.
2. Move cursor to define first arc point or enter radius.
3. Click to set first arc point.
4. Move cursor around the protractor guide or enter angle.
5. Click to set second arc point.
6. Esc = Cancel operation.
7. Tip: Click Window > Model Info > Units > Angle Units to change snapping angle.

Modifier Keys

- Shift (before first click) = Lock pie to current orientation
- Shift (after first click) = Lock tool to current inference direction

Modification Tools

- Move Tool
- Push/Pull Tool
- Rotate Tool
- Scale Tool
- Offset Tool
- Follow Me Tool
- Intersect With Model Tool
- Position Texture Tool

Move Tool

Move, stretch, or copy entities.

Tool Operation

1. Click on entity. Alternatively, pre-select multiple entities with Select tool.
2. Move cursor to new location.
3. Click to finish move operation.

Modifier Keys

- Shift = Lock move to the current inference direction
- Ctrl = Toggles copy of selection
- Alt = Toggles auto fold of selection
- up arrow, left arrow, right arrow = Lock line to specific inference direction (up=blue, left=green, right=red)

Move tool introduction

Use the Move tool 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 (Microsoft Windows), the tool Palette (Apple macOS)or the Tools menu.

Keyboard Shortcut: M

Moving by inference

The Move tool uses SketchUp's sophisticated geometric inference engine

to help you place entities 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 move operation and the geometry of your model. Refer to the inference engine topic for additional information.

Locking a move to the current inference direction

Press and hold the **Shift** key, while the move you are performing is the specific color of an axis, to lock move operation to that axis.

Locking a line to a specific inference direction

Press and hold either the up arrow, left arrow, or right arrow keys, where up arrow equals blue, left arrow equals green, and right arrow equals red, while moving to lock the move to a specific axis.

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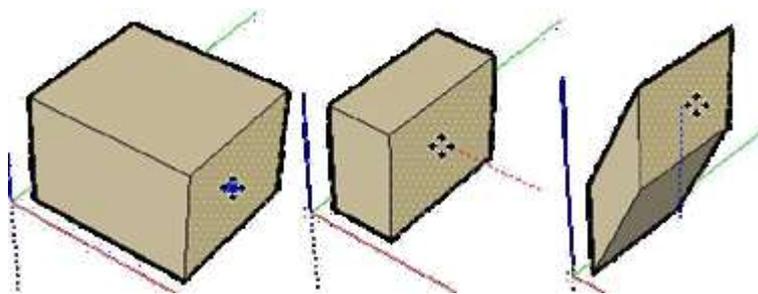
You can also move single line segments to stretch an object. In the following example, a line is selected and moved up in the blue direction to form a sloped roof.
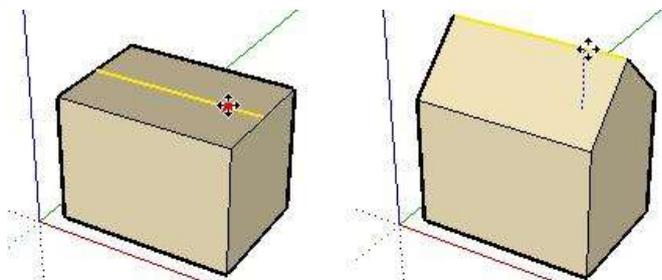
Moving or stretching with Autofold

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.
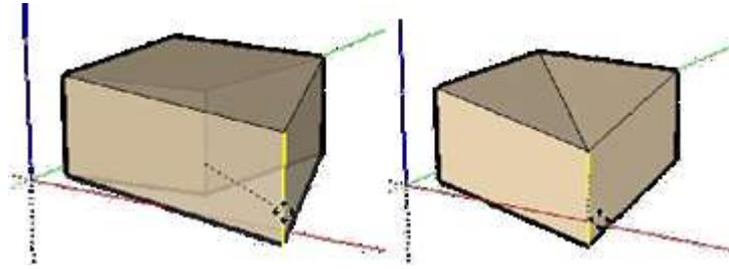
Forcing Autofold Behavior

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 pressing and releasing the Alt (Microsoft Windows) or Command (Apple macOS) key before performing the move operation. This key sequence enables Autofold allowing geometry to move freely in any direction.
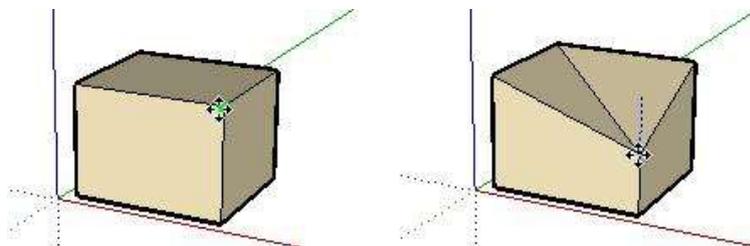
 Making copies

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 ( ). The cursor will change to a four-way arrow.
4. Press and release the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key on your keyboard. The cursor will change to a four-way arrow with a plus sign. This action informs Sketchup that you want to duplicate the selected entities.

5. Click on the selected entities to copy.
6. Move the cursor 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.

Note: You can press and release the **Command** (Microsoft Windows) or **Option** (Apple macOS) key on your keyboard at any time during a move operation to perform a copy (not just at the start).

 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 ( ). The cursor will change to a four-way arrow.
4. Press and release the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key on your keyboard. The cursor will change to an arrow with a plus sign. 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.

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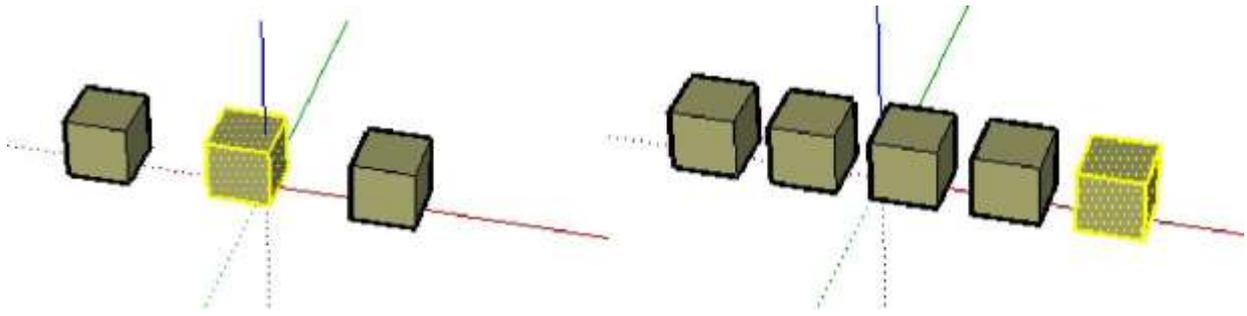
This feature is particularly useful in creating models of items such as fences, bridges, and decks, where you might want several posts or beams an equal distance apart.

Moving precisely

The Measurements toolbar 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 Measurements toolbar.

Type the positive or negative displacement value (such as 20' or -35mm) in the Measurements toolbar and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key.

Note: You can type values in the Measurements toolbar 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 Measurements toolbar.
6. Type the exact or relative coordinate.

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

Length [3',5',7']
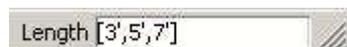
Relative Coordinates: relative to the start point:

Length <1.5m,4m,2.75m>
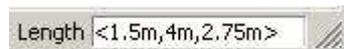
Note: You can define only one or two values as part of your 3D coordinate. For example, to move geometry to 2 feet in the z or blue direction enter the following in the Measurements toolbar: [,,2']

Note: The exact format for values typed in the Measurements toolbar 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]

Push/Pull Tool

Push and pull Face entities to add or subtract volume from your 3d models.

Tool Operation

1. Hover cursor to select a face.
2. Move cursor to push or pull face into 3D form.
3. Click to finish push/pull operation.
4. Esc = Cancel operation.

Pre-Pick Tool Operation

1. Use Select tool to select a face.
2. Activate Push/Pull tool.
3. Click once to set start point of Push/Pull.
4. Click to finish push/pull.
5. Esc = Cancel operation and clear selection.

Modifier Keys

- Ctrl = Toggles creating new starting face
- Alt = Push/Pull while stretching attached faces

Push/Pull tool introduction

Use the Push/Pull tool to push and pull Face entities to add volume to or subtract volume from your 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 the Tool Palette (Apple macOS), the Modification toolbar (Microsoft Windows) or the Tools menu.

Keyboard Shortcut: P

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

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.

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.
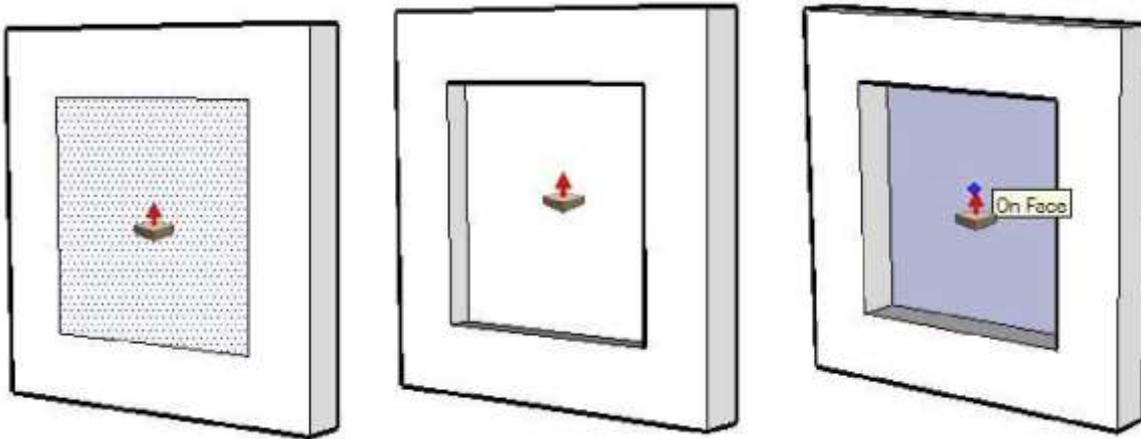
Note: This operation only works when the front and back faces are parallel. Such as when you have two parallel walls in a house and you want to create a void for a door or window.

Creating a new Push/Pull starting face

Push/pull a face (click on the face, move, and then click again) and then press and release **Control** (Microsoft Windows) or **Option** (Apple macOS) (the cursor will contain a plus sign) 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 and released Ctrl (Microsoft Windows) or Option (Apple macOS)and pulled again (middle) and then the user pressed and released Ctrl (Microsoft Windows) or Option (Apple macOS) and pulled again (right).
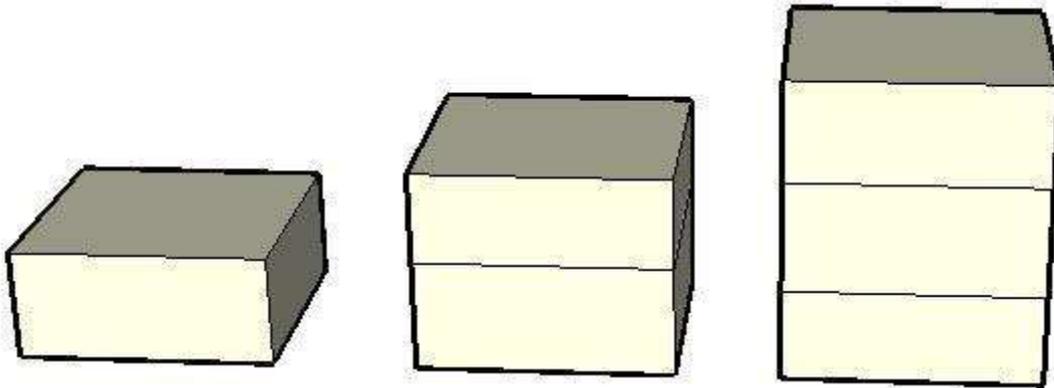
This operation is particularly useful for creating quick space planning diagrams (such as for an office building). Simply use a combination of push/pull and push/pull with Ctrl to create offices, halls, break rooms, meeting rooms and so on (with walls created when Ctrl is pressed).

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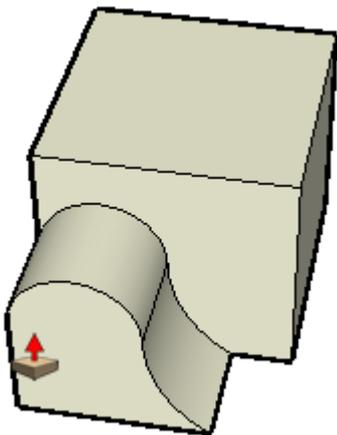
Tip: Select View > Hidden Geometry to view and manipulate the individual faces in the surface.

Pushing and pulling precisely

The displacement of a push/pull operation is displayed in the Measurements toolbar. 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.

Rotate Tool

Rotate, stretch, distort, or copy entities along a rounded path.

Tool Operation

1. Click on entity.
2. Move cursor in circle until it is at staring point of rotation.
3. Click to set starting point of rotation.
4. Move cursor in circle until it is at ending point of rotation.
5. Click to complete rotation.

Modifier Keys

- Shift (before first click) = Lock tool to current orientation
- Ctrl = Toggles rotated copy of selection

Rotate tool introduction

Use the Rotate tool to rotate, stretch, distort, or copy entities along a rounded path. Activate the Rotate tool from the Modification toolbar (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

Keyboard Shortcut: Q

Making rotated copies

The Rotate tool can be used to make rotated copies of entities within your model. To make copies of an entity using the Rotate Tool:

1. Select the **Rotate** tool (   ). The cursor will change to a protractor with a circular arrow.
2. Click on the entity to rotate.
3. Press and release the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key on your keyboard. The cursor will change to a protractor with a plus sign. This action informs Sketchup that you want to duplicate the entity.
4. Move the cursor in a circle until it is at the starting point of the rotation.
5. Click to set the starting point of the rotation. Use the inference tool tips to help you to find the center of the rotation.
6. Move the cursor until it as at the ending point of the rotation. A copy of the entity appears and is rotated about the starting point. If the

'Enable angle snapping' checkbox is checked in the Units Panel of the Model Info dialog box, movements close to the protractor result in angle snaps, while those further away from the protractor allow free rotation.
7. Click to complete the rotation.

> Note: Press and hold the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key at any time during a move operation to perform a copy (not just at the start).

Creating multiple rotated 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 **Rotate** tool ( ). The cursor will change to a protractor with a circular arrow.
2. Click on the entity to rotate.
3. Press and release the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key on your keyboard. The cursor will change to a protractor with a plus sign. This action informs Sketchup that you want to duplicate the entity.
4. Move the cursor in a circle until it is at the starting point of the rotation.
5. Click to set the starting point of the rotation. Use the inference tool tips to help you to find the center of the rotation.
6. Move the cursor until it as at the ending point of the rotation. A copy of the entity appears and is rotated about the starting point. If the 'Enable angle snapping' checkbox is checked in the Units Panel of the Model Info dialog box, movements close to the protractor result in angle snaps, while those further away from the protractor allow free rotation.
7. Click to complete the rotation.
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.
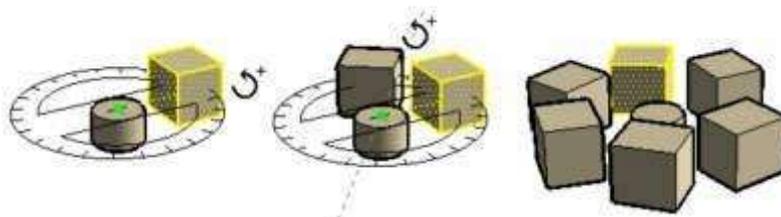
> Note: Press and hold the **Ctrl** (Microsoft Windows) or **Option** (Apple macOS) key at any time during a rotate operation to perform a copy (not just at the start).

Creating copies at an equal distance apart

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

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.
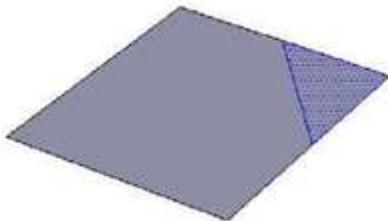
3. Select the **Rotate** tool ( ). The cursor will change to a protractor with a circular arrow.
4. Click and hold on one end of the fold line or edge where the fold will appear in your geometry.
5. Drag the cursor along the fold line to align the protractor to the fold line (the bottom of the triangle).
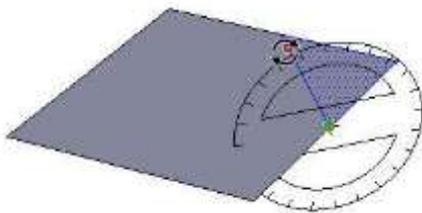
6. Release the mouse button to set the rotation point or the point upon which the geometry will rotate.

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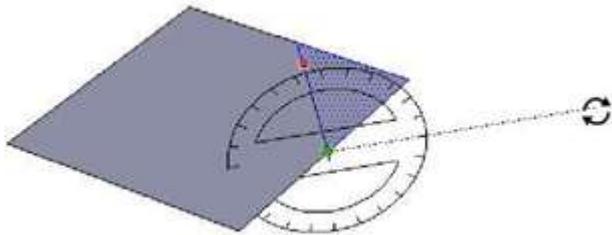
8. Move the mouse to rotate. If angle snaps are active under preferences, you'll notice that as you move the mouse, movements close to the protractor will result in angle snaps, while those further away from the protractor will allow free rotation.
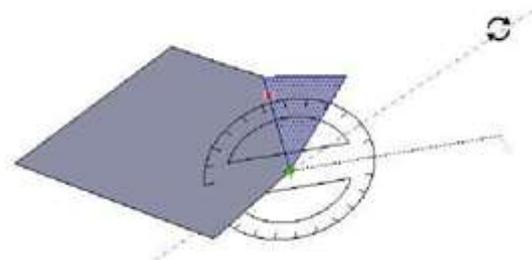

9. Click a third time at the ending point of the rotation (to complete the rotation).
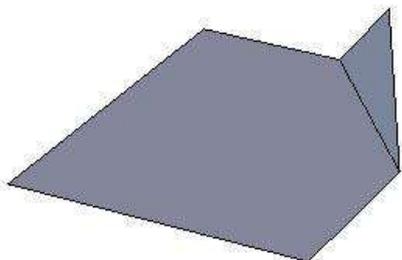
Rotating precisely

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

Entering an angular rotation value

To specify an exact angle in degrees, type a decimal value into the Measurements toolbar 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 Measurements toolbar, 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

Resize and stretch portions of geometry relative to other entities in model.

Tool Operation

1. Click on an entity (lines cannot be scaled).
2. Click on a scaling grip.
3. Move cursor to scale entity.
4. Click to finish offset operation.

Modifier Keys

- Shift = Scale uniformly
- Ctrl = Scale about the geometry's center

Scale tool introduction

Use the Scale tool to resize and stretch portions of geometry relative to other entities in your model. Activate the Scale tool from the Modification toolbar, the Tool Palette or the Tools menu.

Keyboard Shortcut: S

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 2D surface or image entities

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 Command (Microsoft Windows) or Option (Apple macOS) 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 are scaled to match (all instances of the same Line entity in all component instances).

⬛ Scaling about the geometry center

The Scale tool allows you to scale outward from geometry's center point. Press and hold the Control (Microsoft Windows) or Option (Apple macOS) 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.

⬛ Scaling uniformly

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 (Microsoft Windows) or Option (Apple macOS) and 'Shift' keys 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.
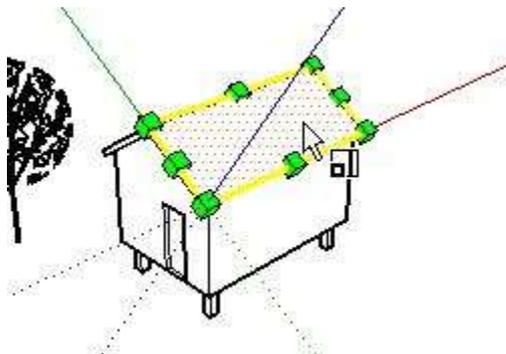
Scaling precisely

The Measurements toolbar 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 Measurements toolbar 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 a 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 Measurements toolbar 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 Measurements toolbar and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key.

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 Measurements toolbar 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 Measurements toolbar 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.)

Offset Tool

Create copies of lines at a uniform distance from originals.

Tool Operation

1. Click on a face.
2. Move cursor.
3. Click to finish offset operation.
4. Esc = Cancel operation.

Modifier Keys

None.

Offset tool introduction

Use the Offset tool to create copies of lines and faces at 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 tool Palette (Apple macOS), the Modification toolbar (Microsoft Windows) or from the Tools menu.

Keyboard Shortcut: F

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.
3. Select the **Offset** tool (   ). The cursor will change to two offset corners.
4. Click on one of your selected line segments. The cursor will automatically snap to the nearest line segment.
5. Move the cursor to define the offset dimension
6. Click to finish the offset operation.
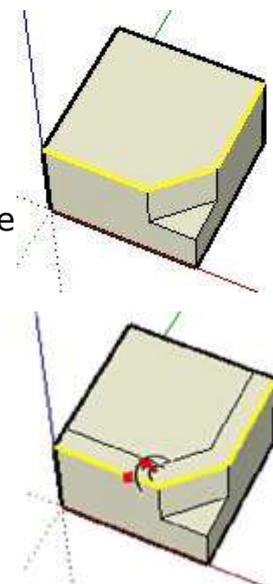
Tip: You can click once on the selected line segments, drag to set the offset while holding down the mouse button, and release the button to accept.

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.

Offsetting precisely

The Measurements toolbar 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 Measurements toolbar 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 an offset value during an 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. Use the Command (Microsoft Windows) or **Option** (Apple macOS) and/or Shift key to change your selection.
3. Select the **Offset** tool (   ). The cursor will change to two offset corners.
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 Measurements toolbar and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS).

Follow Me Tool

Duplicate a face along a path.

Tool Operation

1. Identify edge of geometry to modify. This edge is the path.
2. Draw a face perpendicular to path.
3. Click on Follow Me tool.
4. Click on face.
5. Drag cursor to end of path.
6. Click to complete Follow Me operation.
7. Esc = Cancel operation.

Modifier Keys

Alt = Use perimeter of surface as the path

Follow Me introduction

Use the Follow Me tool to duplicate the profile of a face along a path. This tool is especially useful when trying to add details to a model, such as a crown molding, because you can draw the profile of the molding 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, the Modification toolbar (Microsoft Windows), or the tool Palette (Apple macOS).

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

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 an 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 Tools > Follow Me.
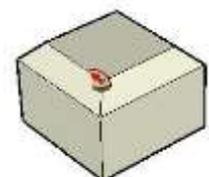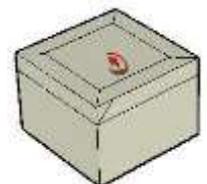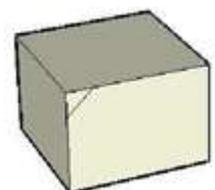
4. Press and hold the Alt (Microsoft Windows) or Command (Apple macOS) Key.
5. Click on the profile that you created.
6. Move the cursor off the profile surface onto the surface around which you wish to sweep. The path will automatically close.

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.

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7. Click to commit the follow-me operation.

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.
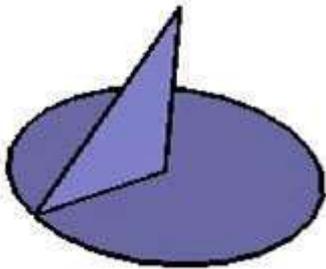
3. Select **Follow** Me tool ( ) The cursor will change to a slanted box with an arrow.
4. Follow the edge of the circle with the face using one of the methods above.
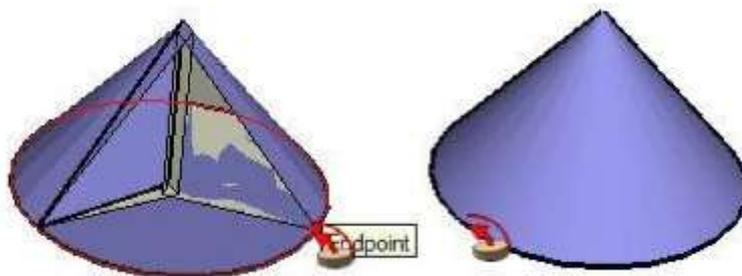
Intersect Tool

Intersect all selected solids but keep only their intersection in the model.

Intersect introduction

Use *Intersect* tool to create complex geometry.

This option allows you 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 *Intersect* from either context menu, the Edit menu or the Solid Tools toolbar.

Position Texture Tool

🔴🖌 Position Texture tool introduction

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.

Use the Position Texture tool 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.
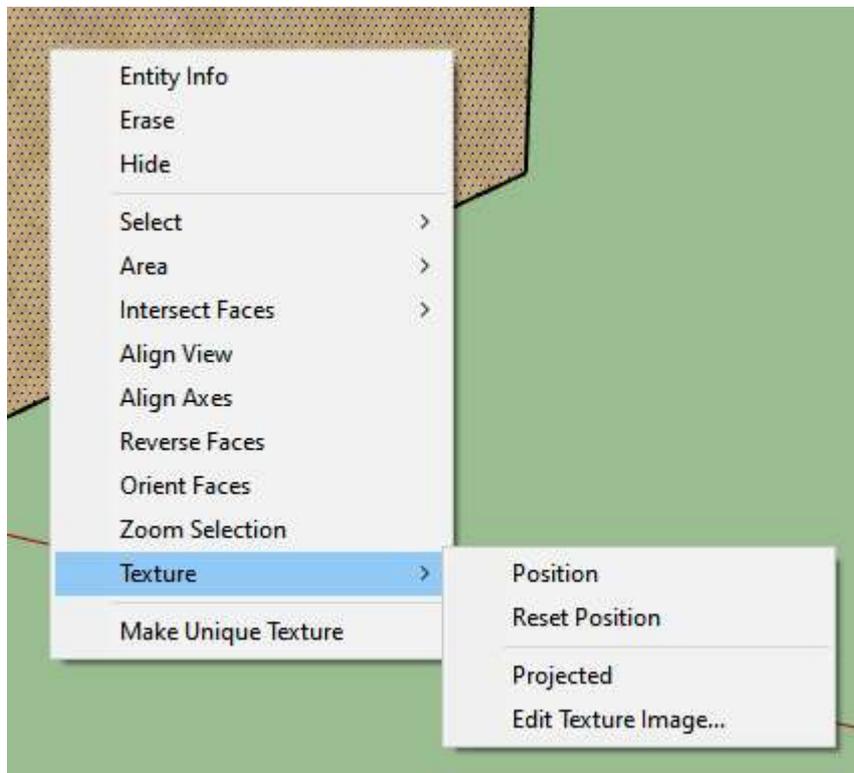
Note: The Position Texture tool can only be used to modify textures applied to flat surfaces.

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.

⊞🖌 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 **Texture > Position**. A matrix of dotted lines is displayed on the material to indicate the material's individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Drag the 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, context-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 to 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 context-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.

⊞🖌 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 **Texture > Position**. A matrix of dotted lines is displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Context-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.
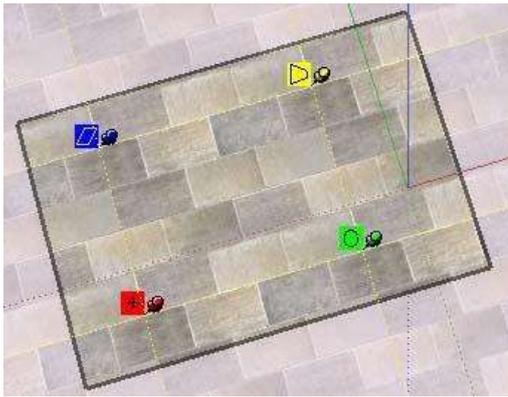
5. Manipulate the material by clicking, holding, and dragging on one of the pins. See Fixed Pin Mode Options in this topic for additional information.

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.

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6. When you are finished modifying the texture, context-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, context-click and select **Done**, or just click outside the texture to close it. Or you can simply press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key when finished.

 Scale/Rotate Icon and Pin: Drag (click and hold) the Scale/Rotate icon or pin to increase the scale of the texture or to rotate it about the Move pin's axis. When you are finished modifying the texture, context-click and select **Done**, or just click outside the texture to close it. Or you can simply press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key when finished. 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 **Texture > Position.** A matrix of dotted lines is displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
3. Context-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, context-click and select **Done**, or just click outside the texture to exit the Position Texture tool.

⊞🎨 Manipulating an 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.   Select the File > Import to insert an image, such as the image of a real door. The Open dialog box is displayed.
3.   Select an image format from the Files of type drop-down list.
4.   Click on the 'Use as texture' radio button.
5.   Select the image file.
6.   Click on the Open button. The cursor changes to the Paint Bucket tool with the image.
7.   Click on one corner of the face to place the starting point of the texture.
8.   Drag the cursor away from the starting point so that the texture is resized over the rectangular face.
9.   Click again to place the texture on the rectangular face.
10.   Context-click on the material to display its context menu.
11.   Select Texture > Position. A matrix of dotted lines is displayed on the material to indicate the materials individual tiles. The cursor also changes to a hand and four pins are displayed.
12.   Context-click on the texture.
13.   Select the Fixed Pins menu item if there is a check mark next to the item.
14.   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.

15.   When you are finished modifying the texture, context-click and select **Done**, or just click outside the texture to exit the Position Texture tool.

⊞🎨 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.   Select **File > Import** to insert an image.

3. Select an image format from the Files of type drop-down list.
4. Select the image file.
5. Click the **Open** button. The cursor changes to the Select tool with the image.
6. Click in the drawing area to place the starting point of the image.
7. Drag the cursor away from the starting point so that the image is resized.
8. Click again to place the image.
9. Context-click on the image. The Image entity's context menu appears.
10. Select **'Use as Material.** The image appears in the In Model (Microsoft Windows) or Colors in Model (Apple macOS)materials library within the Materials Browser.
11. Select the **Paint Bucket** tool ( ). The cursor will change to a paint bucket and the Materials Browser is activated.
12. Click and hold the **Alt** (Microsoft Windows) or **Command** (Apple macOS) key while using the Paint Bucket tool to change to the eye dropper.
13. Click on the thumbnail of your image in the In Model (Microsoft Windows) or Colors in Model (Apple macOS) material library within the Materials Browser.
14. Release the **Alt** (Microsoft Windows) or **Command** (Apple macOS) key.
15. Click on a face of your model and paint the material.
16. Context-click on the material to display its context menu.
17. Select **Texture > Position.** Don't position anything!
18. Context-click again.
19. Select Position Texture tool > Done.
20. Click and hold the **Alt** (Microsoft Windows) or **Command** (Apple macOS) key while using the Paint Bucket tool to change to the eye dropper.
21. Click on the painted material using the eye dropper to sample the material.
22. Release the **Alt** (Microsoft Windows) or **Command** (Apple macOS) key.
23. Paint the sampled texture on the remainder of the model. The texture is wrapped around the 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. Select **File > Import** to insert an image.
3. Select an image format from the Files of type drop-down list.
4. Select the image file.
5. Click the **Open** button. The cursor changes to the Select tool with the image.
6. Click in the drawing area to place the starting point of the image.
7. Drag the cursor away from the starting point so that the image is resized.
8. Click again to place the image.
9. Context-click on the image. The Image entity's context menu appears.
10. Select **Use as Material**. The image appears in the In Model (Microsoft Windows) or Colors in Model (Apple macOS) material library within the Materials Browser.
11. Click on the material in the Materials Browser. The cursor changes to the Paint Bucket tool.
12. 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. Select the File > Import menu item to insert an image.
3. Select an image format from the Files of type drop-down list.
4. Select the image file.
5. Click the Open button. The cursor changes to the Select tool with the image.
6. Click in the drawing area to place the starting point of the image.
7. Drag the cursor away from the starting point so that the image is resized.
8. Click again to place the image.
9. Context-click on the image. The Image entity's context menu appears.

10. Select Use as Material. The image appears in the Colors In Model material library within the Materials Browser.
11. Click on the material in the Materials Browser. The cursor changes to the Paint Bucket tool.
12. Paint the Material on the cylinder. The material will automatically wrap around the cylinder, repeating itself as necessary to wrap the entire model.
13. Select the Display > Hidden Geometry menu item.
14. Select one of the faces of the cylinder.
15. Context-click on the selected face. The Face entity's context menu appears.
16. Select the Texture > Position menu item.
17. Reposition the texture on the face.
18. Sample the repositioned texture using the eyedropper button on the material browser, or using the Alt key with the Paint Bucket tool.
19. Click on Display > Hidden Geometry to turn off Hidden Geometry.
20. 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. Select **File > Import.** 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. Context-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' 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 (Microsoft Windows).

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.

Reorienting materials

You can reorient materials (skew, rotate, resize, and so on) using the Position Texture tool. See Position Texture tool for more information.

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.

> 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.

Make Unique Texture
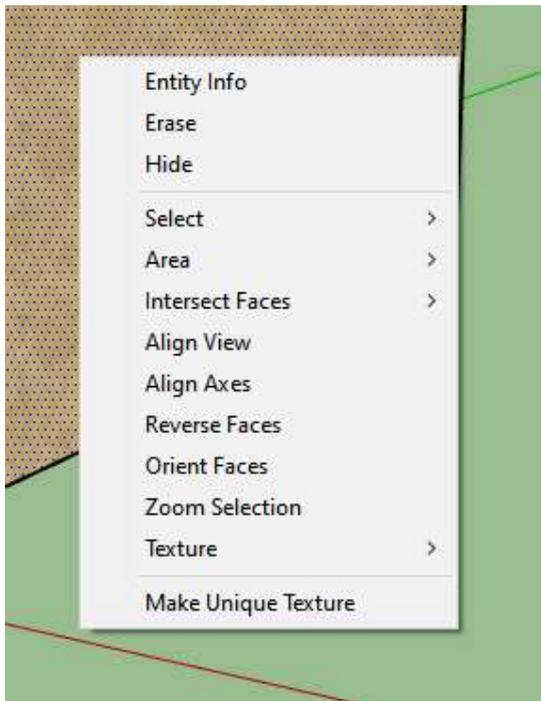
The Make Unique Texture menu item is used to create a texture which is unique to that face. This menu item is not available if SketchUp already considers the texture to be 'unique' and applies to the side of the face which is visible at the time you select the menu.

The Make Unique Texture menu item is also available for a face which doesn't already have a texture applied. The Texture Size dialog box is displayed when you select a face without a texture and then select the Make Unique Texture menu item. This dialog box prompts you to enter a texture size.

This menu item is available for the Face entity.

> Note: The Make Unique Texture menu item will not be available again for a face unless you change the size of the face, edit the material size, or reposition the texture on the face.

Construction Tools

- Tape Measure Tool
- Dimension Tool
- Protractor Tool
- Text Tool
- Axes Tool
- 3d Text Tool
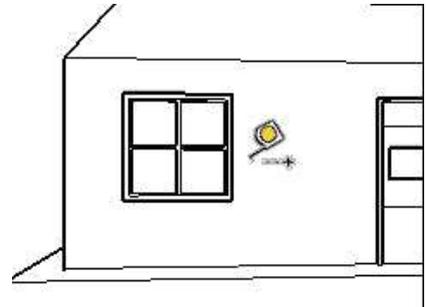
Tape Measure Tool

Measure distances, create guide lines or points, or scale a model.

Tool Operation

1. Click at starting point of measurement.
2. Move cursor.
3. Click at ending point of measurement.

Modifier Keys

- Ctrl = Toggles create guide lines or guide points
- Shift = Lock tape measure to current inference direction
- Up arrow, left arrow, right arrow = Lock line to specific inference direction (up = blue, left = green, right = red)

Tape Measure tool introduction

Use the Tape Measure tool to measure distances, create guide lines or points, or scale a model. Activate the Tape Measure tool from the Construction toolbar (Microsoft Windows), the tool palette (Apple macOS) or the Tools menu.

Keyboard Shortcut: T

Creating Guide Lines and Guide Points

Guide Line entities and Guide Point entities are useful for drawing precisely. To create an infinite parallel guide line using the Tape Measure tool:

1. Select the **Tape Measure** tool ( ). The cursor changes to a tape measure.

2. Click on a line that will be parallel to the guide line, to set the starting point of your measurement. You must click on an On Edge or Midpoint 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 guide line. Start from an 'Endpoint' to create a finite guide line with a guide point at the end.

3. Press and release the Ctrl (Microsoft Windows) or Option (Apple macOS) key.

4. Move the cursor in the direction you want to measure. A temporary measuring tape line and a guide line will stretch out from your starting point.

5. Click again at the point where you want to set your guide line. The final distance is displayed in the Measurements toolbar.
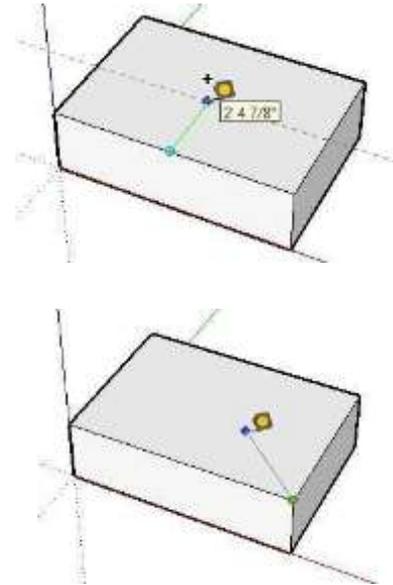
Tip: Starting from an endpoint or midpoint results in a Guide Point.

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. Measure the distance between two points on your model:

    a. Click one end of a line segment to set the starting point of a measurement. Use the inference tool tip to make sure you click on the exact point.
    b. 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.
    c. Click again at the other end of the line segment. The final

distance is displayed in the Measurements toolbar.
3.   Enter a new size for the line in the Measurements toolbar and
      press the Enter (Microsoft Windows) or Return (Apple macOS)
      key. This size will be used as the basis for a proportional rescale of
      your model. The following dialog box appears.
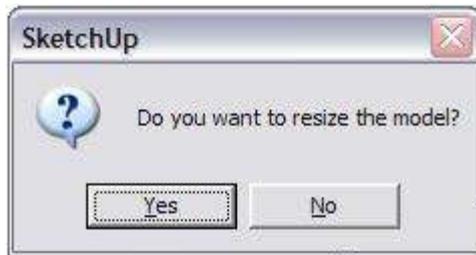
4.   Click the Yes button. The model will be rescaled proportionally.

Note: Only components that are created within the current model (not
dragged from the Components browser and loaded from an external
component file) can be resized. To resize an inserted component you can
double-click the component to be resized and then follow the steps above
to resize that component, you'll need to do this for each inserted
component.

Placing precise guide lines and guide points

The Measurements toolbar displays the distance that a guide line is from
the starting point. Specify a different distance simply by typing it in the
Measurements toolbar. Specify a negative length to draw the line in the
direction opposite the one indicated.

Dimension Tool

Place Dimension entities.
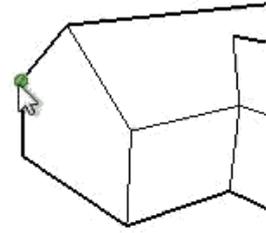
Tool Operation

1.   Click at starting point of dimension.
2.   Move cursor.
3.   Click at ending point of dimension.
4.   Move cursor perpendicular to dimension.
5.   Click to fix position of dimension string.
6.   Esc = Cancel operation.

Modifier Keys

None.

Dimension tool introduction

Use the Dimension tool to place Dimension entities in your model. Activate the Dimensions tool from the Guide toolbar (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

Placing radius dimensions

To place a radius dimension on an Arc entity:

1.   Activate the **Dimension** tool ( ). The cursor changes to an arrow.
2.   Click on an Arc entity.
3.   Move the cursor to pull a dimension string out from the model.
4.   Click the mouse again to fix the 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 an arrow.
2.   Click on a Circle entity.
3.   Move the cursor to pull a dimension string out from the model.
4.   Click the mouse again to fix the position of the dimension string.

Toggling dimension type

Context-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).

Protractor Tool

Measure angles and create angled Construction Line entities.
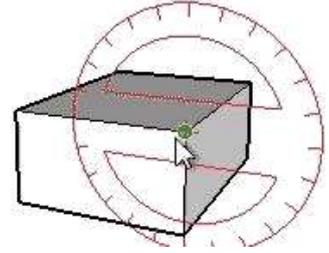
Tool Operation

1. Place protractor's center at vertex (where two lines meet) of angle.
2. Click to set vertex.
3. Move cursor in circle until touching start of angle (one of the lines).
4. Click to set start of angle.
5. Move cursor in circle until touching end of angle (other line).
6. Click to measure angle.
7. Esc = Cancel operation.

Modifier Keys

*   Ctrl = Toggles create construction line
*   Shift = Lock inference

Protractor tool introduction

Use the Protractor tool to measure angles and create angled guide lines. Activate the Protractor tool from the Guide toolbar (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

Locking the Protractor tool to its current orientation

Press and hold the Shift key, before you click on an entity, to lock the operation to that orientation.

Creating precise angles

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

Entering an angular rotation value

To specify an exact angle in degrees, type a decimal value into the

Measurements toolbar 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 Measurements toolbar, 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.

Text Tool

Geodesic sphere

Place Text entities.
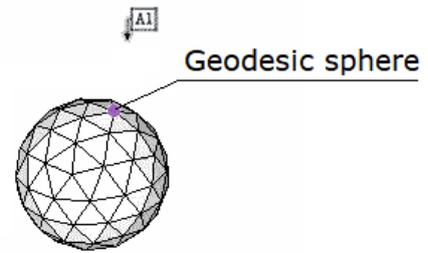
Tool Operation

1.   Click on an entity to indicate ending point of leader line (location where leader will point).
2.   Move cursor to position text.
3.   Click to place text.
4.   (optional) Click in text box.
5.   (optional) Enter text in text box.
6.   Click outside text box to complete operation.
7.   Esc = Cancel operation.

Modifier Keys

None.

Text tool introduction

Use the Text tool to insert text entities into your model. Activate the Text tool from either the Construction toolbar (Microsoft Windows), the tool palette (Apple macOS), or the Draw menu. There are two types of text in SketchUp: Leader text and Screen text.

Creating and placing screen text

Screen text contains characters and is not associated with an entity and is fixed to the screen regardless of how you manipulate or orbit the model.

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** (Microsoft Windows) or **Return** (Apple macOS) key twice, to complete text entry. Screen text will stay fixed on the screen regardless of how you manipulate and orbit the model.

Tip: Type \n followed by a space to start a new line of text (Microsoft Windows).

Editing text

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

Configuring 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.

Axes Tool

Move or reorient drawing axes.
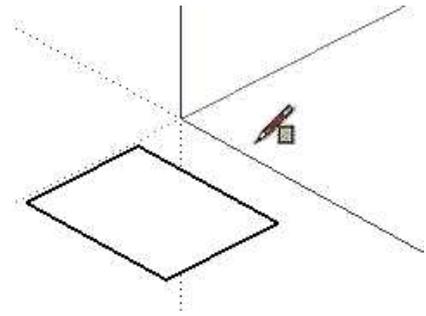
Tool Operation

1. Move cursor to point in drawing area for new origin.
2. Click to establish origin.
3. Move cursor away from origin to set direction for red axis.
4. Click to accept direction.
5. Move cursor away from origin to set direction for green axis.
6. Click to accept direction.
7. Esc = Cancel operation.

Modifier Keys

None.

Axes tool introduction

Use the Axes tool 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 (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

Resetting the drawing axes

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

Note: When context-clicking on the axes, there must be empty model space behind the axes. If there are entities behind the axes, the context menu for the entities will appear.

3D Text Tool

Create 3 dimensional text using any font.
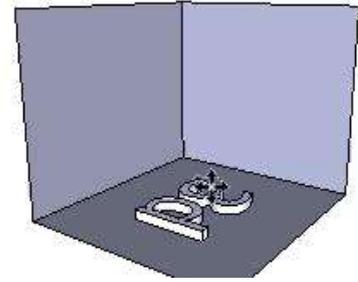
Tool Operation

1.   Type text in text box.
2.   Press Fonts... button and select font, font style, points and height.
3.   Check Filled checkbox.
4.   Check Extruded checkbox.
5.   Press Place button.
6.   Move cursor to position text.
7.   Click to finish position operation.

Modifier Keys

None

3D Text tool introduction

Use the 3D Text tool to create 3 dimensional geometry from text. Activate the 3D Text tool from either the Construction toolbar (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

Place 3D Text dialog box

Use the options in the Place 3D Text dialog box to enter and configure 3D text.

Font

Select a Font from the drop-down list to change font. Select Regular or Bold from the drop-down menu to create regular (non-bold) or bold text, respectively.

Height

Type the height, in current modeling units, in the Height text entry box.

Align

Select Left, Center, or Right, from the drop-down list to align two or more lines text to the left, center, or right respectively.

Filled

The 3D Text dialog box allows you to create 2D text with just outlines (edges) or faces, or 3D extruded text. Check the Filled checkbox to create faces for 3D text. Uncheck the Filled checkbox to create 2D text outlines (just edges).

Note: The Filled checkbox must be checked to create 3D text.

Extruded

Check the Extruded checkbox to create extruded (push/pull) 3D text. Uncheck the Extruded checkbox to create 2D text.

Note: The extruded checkbox must be checked to create 3D text.

Section Tools

- Section Plane Tool
- Display Section Planes
- Display Section Cuts
- Display Section Fill

Section Plane Tool

Create section cuts effects enabling you to view geometry within a model.

Tool Operation

Click on a face to cut create a Section Plane entity and resulting section cut effect.

Modifier Keys

Shift (before first click) = Lock tool to current orientation

Section Plane tool introduction

Use the Section Plane tool to create section cuts enabling you to view geometry within your model. Activate the Section Plane tool from the Guide toolbar (Microsoft Windows), the Tool Palette (Apple macOS) or the Tools menu.

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 context-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, is selected.

There are two ways to activate a section plane: double-click on the section plane while in the Select tool or context-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

To create grouped section slices, context-click on a Section Plane entity, then select 'Create Group from Slice' from the context menu. new edges, encapsulated within a group, are generated (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.

Using sections with scenes

Active Section Plane may be saved to a scene. Section cut effects will animate during animations.

Aligning your 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.

Display Section Planes

Toggle Section Planes on and off.

Display Section Cuts

Toggle Section Cuts on and off.

Display Section Fill

Toggle Section Fill on and off.

Camera Tools

- Orbit Tool
- Pan Tool
- Zoom Tool
- Zoom Window Tool
- Zoom Extents Tool
- Previous Tool
- Position Camera Tool
- Look Around Tool
- Walk Tool

Orbit Tool

Rotate camera about model.

Tool Operation

1. Click anywhere in drawing area.
2. Move cursor in any direction to rotate around center of drawing area.

Modifier Keys

- Shift = Pan Tool
- Ctrl = Suspend gravity (do not try to keep vertical edges up and down)

Orbit tool introduction

Use the Orbit tool to rotate the camera about the model. The Orbit tool is useful when viewing geometry from the outside. Activate the Orbit tool from either the Camera toolbar (Microsoft Windows), Tool Palette (Apple macOS), or the Camera menu.

Keyboard Shortcut: O

Orbiting with a three button mouse

The Orbit tool is used heavily when creating and editing models. SketchUp contains a few 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 (scroll wheel) on a three-button mouse to temporarily activate the Orbit tool while in any other tool (except the Walk tool).

Note: Click and hold the Control and Command keys simultaneously while clicking and holding the left mouse button to temporarily activate the Orbit tool (if you have only one mouse button) (OS X).

Note: 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 (Mac = Opt) key during orbit to suspend this gravity setting and to roll the camera on its side.

Pan Tool

Move camera (your view) vertically or horizontally.

Tool Operation

1. Click anywhere in the drawing area.
2. Move cursor in any direction to pan.

Modifier Keys

Esc = Enable previously selected tool

Pan tool introduction

Use the Pan tool to move the camera (your view) vertically and horizontally. Activate the Pan tool from either the Camera toolbar (Microsoft Windows), or Tool Palette (Apple macOS) or the Camera menu.

Keyboard Shortcut: H

Panning while in the Orbit tool (Three 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 (scroll wheel) to activate the Pan tool.

Zoom Tool

Move camera (your view) in or out.

Tool Operation

1. Click and hold anywhere in drawing area.
2. Drag cursor up to zoom in (closer to model) and down to zoom out (farther from model).

Modifier Keys

- Shift = Change field of view degrees
- Esc = Enable previously selected tool

Zoom tool introduction

Use the Zoom tool to move the camera (your view) in or out. Activate the Zoom tool from either the Camera toolbar (Microsoft Windows), or Tool Palette (Apple macOS), or the Camera menu.

Keyboard Shortcut: Z

Zooming with a scroll wheel mouse

Rolling 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 the model in the drawing area.

Changing field of view

Field of view, expressed in degrees, refers to the amount you can see of a model. A narrow field of view only allows you to see a small portion of a model, while a wider field of view allows you to see more of a model. Wider fields of view are useful when working inside a room where you might want to see more of the room while you draw. To change field of

view:

1. Select the **Zoom** tool. The cursor changes to a magnifying glass.
2. Hold the shift key while moving the cursor up or down. The field of view increases when the cursor goes up. The field of view decreases when the cursor goes down.

Note: The Measurements toolbar will display field of view in degrees when using the Zoom tool in the field of view mode.

Note: Alternatively, you can select Camera > Field Of View to change field of view.

Zoom Window Tool

Use Zoom Window Tool to zoom in to a specific area of screen.

Tool Operation

1. Click and hold a short distance away from entities you want to appear in zoom window.
2. Move cursor diagonally.
3. Release when all entities are enclosed within zoom window.

Modifier Keys

Esc = Enable previously selected tool.

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 the 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 a short distance away from the entities you want to appear in the zoom window. This is the zoom window starting point.
3. Move cursor diagonally.
4. Release the mouse button when all of the entities are enclosed within the zoom window. These entities will fill the screen.

Zoom Extents Tool

Use the Zoom Extents tool to zoom your view to a distance which makes the whole model visible and centered in the drawing area. Activate the Zoom Extents tool from either the Camera toolbar (Microsoft Windows), or Tool Palette (Apple macOS), or the Camera menu.

Keyboard Shortcut: Shift+Z

Previous Tool

Undo to the previous camera view.

Position Camera Tool

Position camera (your view) at a specific eye height to check line of sight or walk through model.

Tool Operation

Click on a point in your model. Camera is placed at average eye-height over point. You are placed in Look Around Tool.

Modifier Keys

None.

Position Camera tool introduction

Use the Position Camera tool to position the camera (your view) at a specific eye height so that you can check the line of sight of a model or walk through a model. Activate the Position Camera tool from either the Walkthrough toolbar (Microsoft Windows), Tool Palette (Apple macOS) or the Camera menu.

Positioning the camera

The camera is positioned using one of two methods. The first method places the camera at eye-level above a specific point (5' 6" above that point by default). The second method placing the camera at a specific point, facing a specific direction.

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 (👤). The cursor changes to a small person with a red X. Notice that the Measurements toolbar 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 on a point in your model. SketchUp
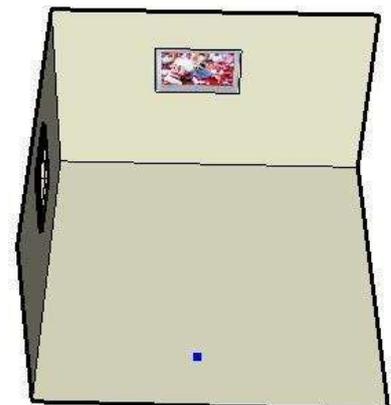
places the camera's point of view at an average eye-height over the point you click on. You are also placed in the Look Around tool. The following image shows a point in the middle of a room. The camera will be positioned directly above this point (5' 6") facing the television set if you click at this point.

Note: The viewing direction defaults to the top of screen, which is due north, if you place the camera from a plan view.

Positioning the Camera Using Specific Target Points

The second position camera method allows you to position the camera at a specific point, facing a specific direction.

1. Select the **Position Camera** tool (🧍). The cursor changes to a small person with a red X. Notice that the Measurements toolbar 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 Measurements toolbar to drag parallel construction lines off of edges as a method to provide accurate camera placements.

2. Click and hold mouse button on a point in your model.
3. Drag the 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.
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.
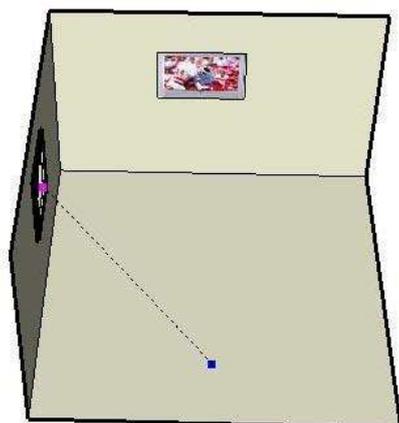
5. (optional) Type a new eye height into the Measurements toolbar to reposition the camera at eye height above the point selected in step 2.

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.

Look Around Tool

Pivot camera (your view) from a stationary
point.

Tool Operation

1. Click to start pivot.
2. Move cursor up or down to tilt, right or left to pan.

Modifier Keys

None.

Look Around tool introduction

Use the Look Around tool to pivot the camera (your view) around a
stationary point. 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
(Microsoft Windows, Tool Palette (Apple macOS) or the Camera menu.

Specifying an eye height

Type the eye height for the camera in the Measurements toolbar and
press the Enter (Microsoft Windows) or Return (Apple macOS) key to
change the height above the ground plane for the camera.

Activating the Look Around tool while in the Walk tool

Click the middle mouse button to activate the Look Around tool while in
the Walk tool.

Walk Tool

Walk through (tour) a model.

Tool Operation

1. Click and hold anywhere in drawing area. A small plus sign (cross hair) is placed at that location.
2. Move cursor up (forward), down (backward), left (turn left) or right (turn right) to walk. The further from cross hair, the faster you walk.

Modifier Keys

- Shift = Move up or down instead of forward or backward
- Ctrl = Run instead of walk
- Alt = Walk through entities

Walk tool introduction

Use the Walk tool 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 the Walkthrough toolbar (Microsoft Windows), Tool Palette (Apple macOS) or the Camera menu.

Looking around while in the Walk tool

Click and hold the middle mouse button, while using the Walk tool, to use the Look Around tool.

Ascending and descending

The Walk tool automatically walks up and down inclines or steps while maintaining eye height.

Toggling collision detection

Press the Alt (Microsoft Windows) or Command (Apple macOS) 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).

Groups and Components

- Groups
- Components

Groups

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:

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.
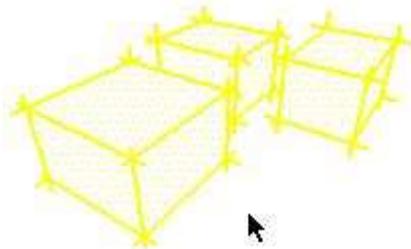
5. Select the **Edit > Make Group.** Alternatively, context-click on the currently selected entities and select **Make Group** from the context menu. The geometry you selected appears grouped within a highlighted bounding box.

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.

Note: You can ungroup grouped entities by context-clicking on the group and then selecting the **Explode** menu item.

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.
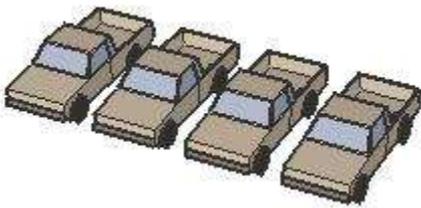
Each entire Group entity was then painted after being edited, turning each of the faces with the default material to the color applied on the group (green, blue, red, and yellow from left to right):
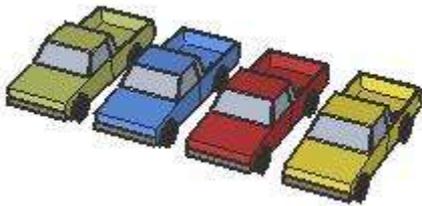
Editing a group

Groups can be opened for editing, placing you in the Group's context.

To edit a group:

1.   Select the **Edit > Group > Edit Group** 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. Double-click on the group to edit the group.
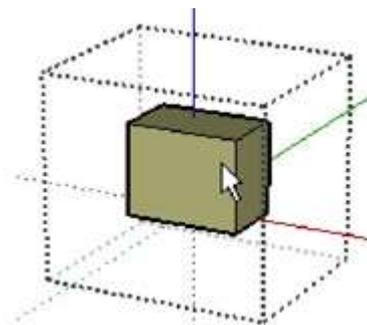
2. Make changes to entities within the group. Any changes while in the group's context only affects the Group entity. You can, however, perform inference alignments to geometry outside of the group while you are editing the group.
3. Select the **Edit > Close Group / Component** to end the edit session. Alternatively, context-click on the currently selected group's bounding box and select **Close Group** from the context menu. Click outside of the group to close the Group.

Components

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 Components 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. Draw your component in the orientation where it will be used. For
   example, if you are creating a couch component, draw it on the
   ground plane. If you are creating a window or door, draw it on a
   wall that is vertically aligned to the blue axis.
2. Select the **Select** tool. The will change to an arrow.
3. Click and hold the mouse button a short distance away from the
   entities you want to select to start a selection box.
4. Drag the mouse to the opposite corner of the selection starting
   point.
5. 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.
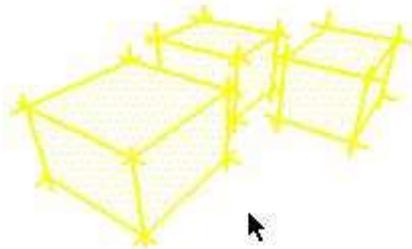
6. Select **Edit > Make Component**. Alternatively, context-click on
   the currently selected entities and select **Make Component** from
   the context menu. The Create Component dialog box is displayed.
   The following image shows the Create Component dialog box on
   Microsoft Windows:
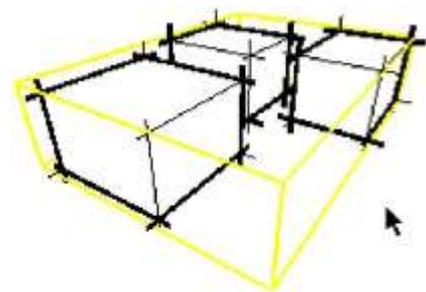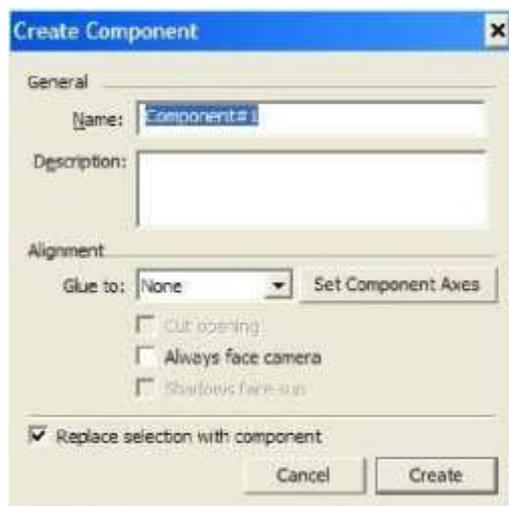
Warning: Components that are created in their appropriate 'glue to' orientation are created with the blue and green axis swapped. This is only of importance when creating dynamic components as the LENZ will have a value for length along the green axis and the LENY will have a value for the length along the blue axis.

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.

Setting the gluing and cutting plane of a component

The Set gluing plane button 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.   Select **Show Component Axes** in the Components panel of the Model Info dialog box to see the existing axes on a component before using the 'Set gluing plane' button to move the axes.
2.   Draw your component in the orientation where it will be used. For example, if you are creating a couch component, draw it on the ground plane. If you are creating a window or door, draw it on a wall that is vertically aligned to the blue axis.
3.   Select the **Select** tool. The will change to an arrow.
4.   Click and hold the mouse button a short distance away from the entities you want to select to start a selection box.
5.   Drag the mouse to the opposite corner of the selection starting point.
6.   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.
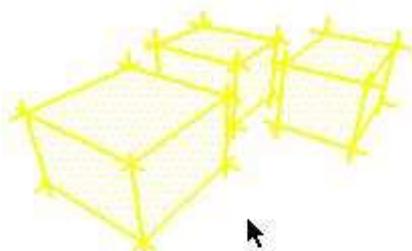
7.   Select **Edit > Make Component**. Alternatively, context-click on the currently selected entities and select **Make Component** from the context menu. The Create Component dialog box is displayed. The following image shows the Create Component dialog box on Microsoft Windows:
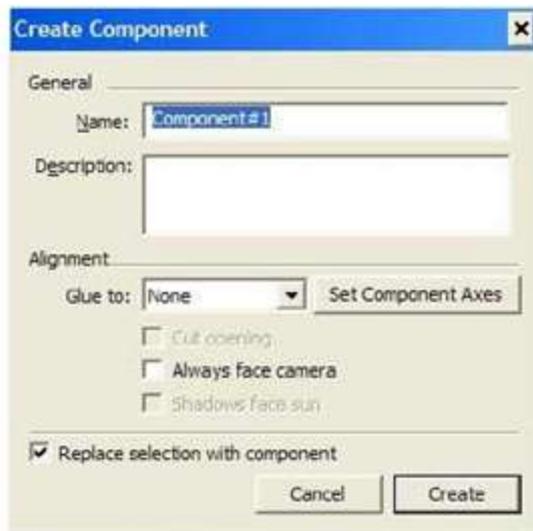
Note: When you change the axes you change the orientation of the component when dragged from the Components browser and also the component's cutting plane. By default, the component's axes are placed as though the component will be inserted from the Components browser in the exact orientation as the 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 reorient the component axes.

8. (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.
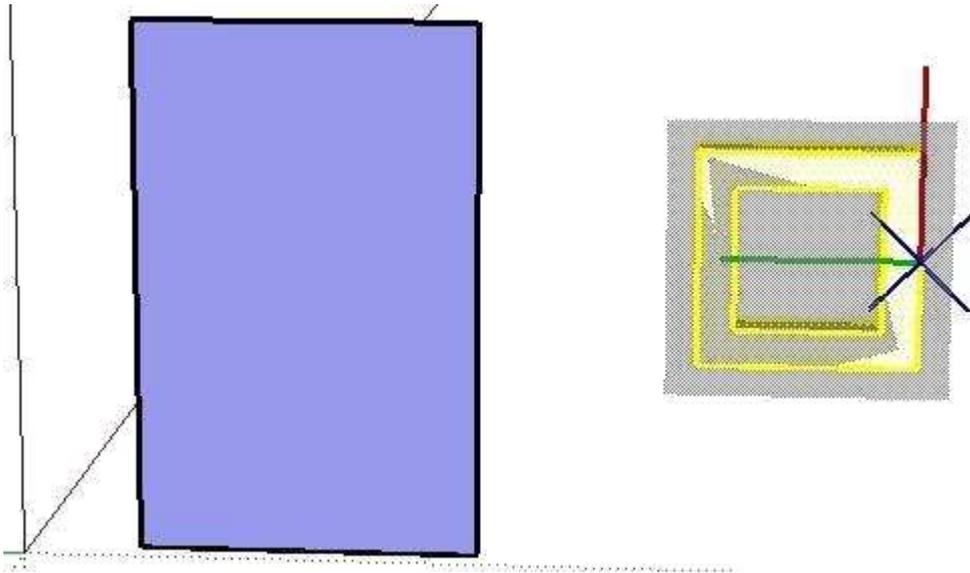
9. (optional) Click to set the new orientation.

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 **Edit > Component Instance > Edit Component** 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 Edit > Close Group / Component** 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.

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 **Edit > Component Instance > Explode**. 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.

Inserting Components

There are a variety of different ways to insert component instances in SketchUp.

Inserting components from Components browser

Pre-defined components are most often inserted from the Components browser into a model. See the 'Components 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** menu item. The Open file dialog box appears.

2. Click the **Open** button. 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.

3. Move the mouse to the location in the drawing area where you want to place the component.
4. Click the mouse button again to release the component.

Insert a component from the File Explorer (Microsoft Windows) or the Finder (Apple macOS).

To insert a component from the File Explorer (Microsoft Windows) or the Finder (Apple macOS):

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.

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.

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4. Release the mouse button to place the component into your model.

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.
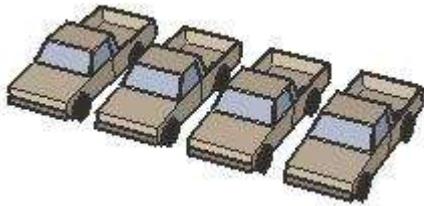
Each entire component instance was then painted after being edited, turning each of the faces with the default material to the color applied on the group (green, blue, red, and yellow from left to right):
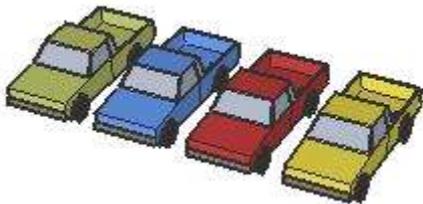
Moving the insertion point

The Components 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

Context-click on the component in the Components browser and select **Reset insert point** to reset the insertion point to the origin of the component's axes.

Inserting a Component

To insert a component in your drawing area:

1. Navigate to a component in the Components browser.
2. Click on the component.
3. 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 1: 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.

Note 2: 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 Components 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 tool.

4. 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 (Microsoft Windows) or the Finder (Apple macOS) into your model. See the 'Component entity' for further information.

Create component dialog box

The *Create Component* dialog box is displayed when you attempt to create a component using the *Make Component* menu item from the *Edit* menu.

General
Definition
The *Definition* field (formerly named *Name*) can contain the name of the component. 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 Components browser. For example, a standard door might only be glued to faces in the vertical (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.
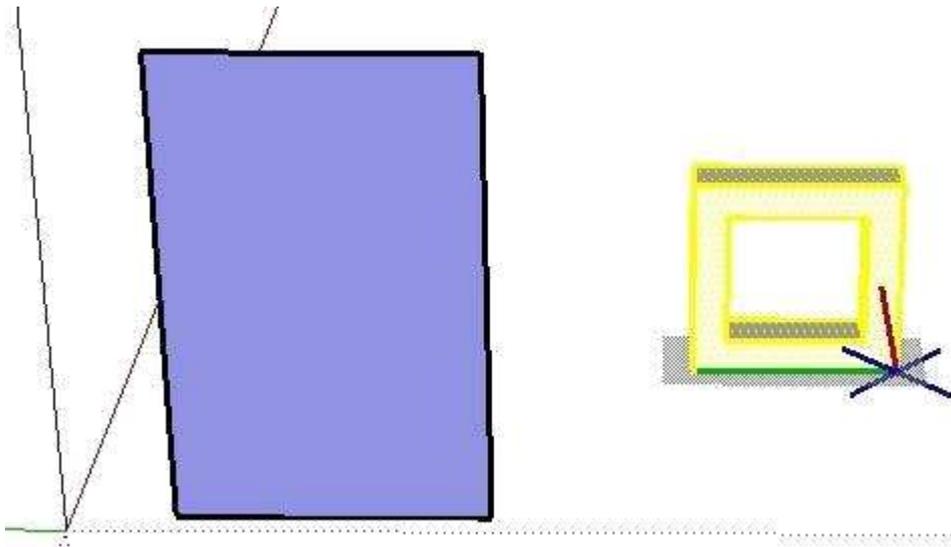
The following image shows what occurs when the previously mentioned window component is placed, from the Components browser, against a vertical surface. Notice that the window meets the vertical face at the window's bottom because it was created with a gluing plane that is parallel to the bottom of the window.
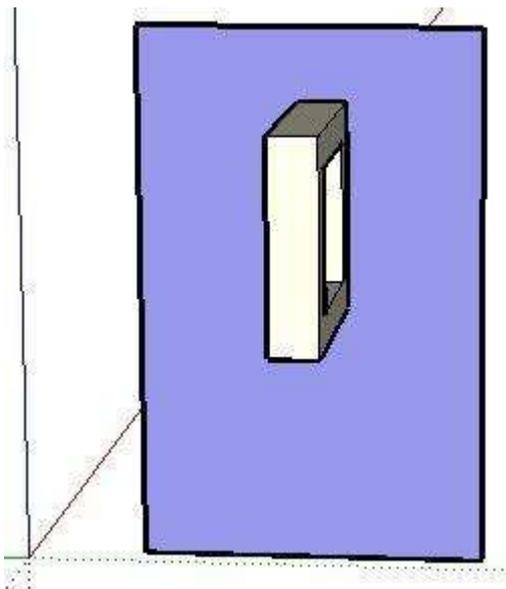
Usually you want windows and doors to have a gluing plane that is parallel to the front or back of the window or door (not the bottom). You might need to reorient the component's axes at creation, using the 'Set Plane' button, to properly set the gluing or cutting plane.

Set Component Axes

The *Set Component Axes* (formerly *Axis*) defines how the component inserts and aligns to other geometry or to the camera. The component axes also defines the cutting plane by the orientation of the red/green plane. The Set component axes button is used to specify a different origin for the component and to modify the orientation of the component when it

is placed.

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.

Shadows face sun

This option is only available when the 'Always face camera' option is enabled. The shadows face sun option causes the shadow to be cast from the component's current position as though the component were facing the sun. The shadow shape does not change as the component rotates to face the camera. This option works best with components that have short bases (such as trees). This option does not work well with components that have wide bases (such as people in mid stride).

> Note: Ensure the component's axis is positioned at the bottom center of the component for best results.

Uncheck this option to cause the shadow to be cast from the component's current position. The size of the shadow changes based on view point.

Advanced attributes

Design is so much more that defining what something looks like. With the Advanced Attributes section in the Create Component dialog box, you can attach information to components. This information helps teams make design decisions and supports the eventual construction process.

For example, if your component is a door and you add a price, size, and

type to your component, you can then generate a report that can help you estimate how much using that door in your project would cost. To see how using a different type of door might impact the cost, you can swap one door component for another using SketchUp's Replace Selected component feature and see an updated report with new cost estimates.

You can set the following attributes in the Create Component dialog box:

- **Price**: To do simple price calculations based on content you create, enter the cost of your component in the text box. Note that the Price attribute does not support different currencies that this time. Simply enter a numerical value.
- **Size**: Enter a simple indication of size. For example, you might enter 30x80 to indicate the size of a door. Note that scaling does not update the string-based attributes. To add that kind of logic, you need to create a dynamic component.
- **URL**: Enter a web page address that is relevant to the component, such as the page where you can purchase a door or window and find other technical specifications from the manufacturer.
- **Type**: Select an option from the Type drop-down list if you're using IFC classification data. See Classifying Objects for details.

Tip: After you create a component, you can also add attributes for the status and owner on the Entity Info panel. With the component selected and the Entity Info panel open, click the Show Advanced Attributes icon. You see a list of advanced attributes you added in the Create Component dialog box as well as options to enter details about the component status or owner. Simply type your desired information in the appropriate text box. Keep in mind that the instance values apply only to an instance of the component. If you change an instance value (status or owner), the change applies only to that specific component instance, not all components with that definition. In the Entity Info panel, you can also edit values for the Price, Size, URL, and Type attributes, which are part of the component definition. Changing a definition value changes all instances of the component.

Note: When you use LayOut to create construction documents, any component attributes that you specify in SketchUp flow into LayOut. First, in SketchUp, generate a report as a .csv file. Then, in LayOut, you can import the .csv file data into a table. This compatibility is useful not only for creating tables, but also labeling items automatically.

Replace selection with component

The Replace selection with component option causes the currently selected entities to be turned into a component instance. Uncheck this

option to create a component definition in the Components browser without creating a component instance from selection set.

Layers and Outliner

- Layers
- Outliner

Layers

The Layers Manager is used to apply and manage layers in your model. Activate the Layers Manager from the Window menu. The following image shows the Layers Manager on Microsoft Windows:
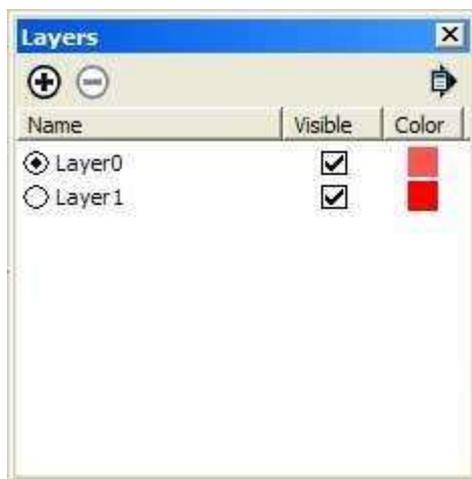
The Layers Manager displays all the layers and their associated visibility in the model. Every model has one layer, called Layer0, which will be visible the first time you activate the Layers dialog box.

> Note: You cannot delete Layer0. Entities that reside on Layer0 always inherit their visibility from the layer of the components/groups that contain them. This makes layer 0 work like the default drawing layer. 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.

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.

Add

Use the Add button to create additional layers. Enter the layer name (or press **Enter** on Microsoft Windows or **Return** on Apple macOS to accept

the default name). Each new layer has a different color to help you distinguish layers.

Delete

Select a layer and use the Delete button 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.

Use 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 press the Ctrl key and click on layer names to selectively pick layers or press the Shift key and click to select a series of contiguous layers.

Visible

Use Visible checkbox to toggle the visibility of a layer. 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. Click on a layer and select a new color to change the color of the layer. Click on the Color column header to sort layers by color. Click again on the Color column heading to reverse the order.

Using the layers toolbar (PC)
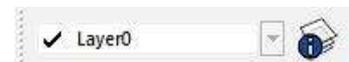
The Layers toolbar provides quick access to several often used layer operations.

Display the current layer

When nothing is selected, the current layer name is displayed in the toolbar and has a check mark next to it. Any new entities you draw will be assigned to the current layer.

Change the current layer

Select a layer from the Layers toolbar, when nothing is selected, to change the current layer.

View entity's layer

Select an entity. The layer for the entity will be displayed, with a check mark, in the Layers toolbar.

Change the layer assignment of entities

Select an entity and then a layer from the Layer toolbar to change the layer of an entity.

Display the Layers Manager

The button on the right hand side displays the Layers Manager allowing you to create new layers, control their visibility, and more.

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.

Note: Assigning a different layer to a group or component doesn't affect the layer assignments of the individual entities within the group or component. If you want to assign new layers to the entities within a group or component, you must edit the group or component, select specific entities within the group or component, and then assign a different layer to the selected entities.

Working With Layers

Layers are important in determining how to lay out your content in a document. Layers are a mechanism for toggling the display of certain items, such as when displaying a presentation. A special layer, called a shared layer, also allows you to replicate the same content across all pages in your document. You should always be aware of the layer you are

on as you create or insert content. To identify the layers in a document:

1. Select **Window > Layers.** The Layers dialog box is displayed. This dialog box contains all of the layers in the document. The Letter Landscape template contains 2 layers: 'Default' and 'On Every Page.' The red pencil indicates that the Default layer is the current drawing layer. Anything drawn on the page will be placed on this layer. The Default layer is used for anything you draw or insert that you only want to appear on the page where its drawn or inserted as indicated by the shared layer icon to the right of the layer name. If this icon has one page (☐), the layer contains items that appear on one page. If this icon has four pages, the layer contains items that appear across all pages (shared).
2. Click on the **Add new layer (+)** button. A new layer, titled 'Layer 3' is added.
3. Double-click on 'Layer 3.' The 'name' field is opened for edit.
4. Type 'Text' and press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key. This new layer will be used to hold all text entities in your document. We will use this layer later in this getting started guide.
5. Click on the 'On Every Page' layer. The On Every Page layer is a shared layer as indicated by the shared layer icon with multiple pages

Outliner

The 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 Outliner dialog box from the Window menu. The following image shows a screenshot of the Outliner on Microsoft Windows:
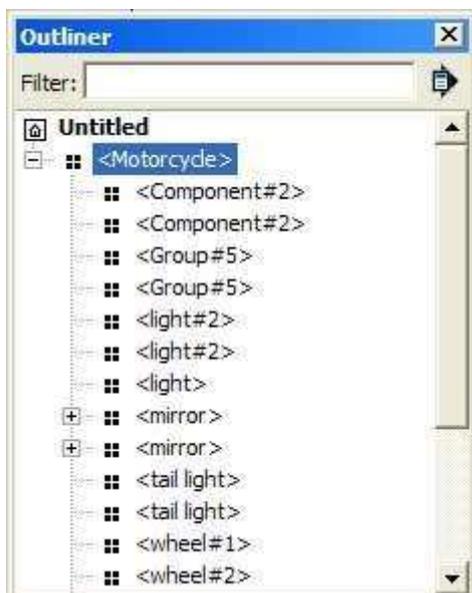
Identifying entries in the outliner hierarchy

The 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 outliner

Use the 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. Select **Window > Outliner.** The 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 124groups when additional levels in the hierarchy exist (additional groups or components within the top-level groups or components).
2. Click on the + 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 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 Outliner:

1. Select **Window > Outliner**. The 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 when additional levels in the hierarchy exist (additional groups or components within the top-level groups or components).

2. Type a string in the 'Filter' field.

3. Click on the + 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.

Naming groups and components within the outliner

The 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. 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 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 the **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 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 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 Outliner.
4. Press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key 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.

Note: 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 Outliner. To rename a group:

1. Context-click on the group in the 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 Outliner.
4. Press the **Enter** (Microsoft Windows) or **Return** (Apple macOS) key to save your change.

Traversing component and group hierarchies

The outliner contains a hierarchy view that reflects your hierarchy of components and groups in your model. To navigate a component or group hierarchy using the hierarchical view:

1. Select **Window > Component**. The 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 when additional levels in the hierarchy exist (additional groups or components within the top-level groups or components).
2. Click on the + icon to list groups or components within other groups or components. Or, optionally, click on the 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.

Scenes, Shadows, Styles and Fog

Scenes

You can use Scenes to save camera views, as well as several additional properties.

For example, open a new model, change the view-move the camera by orbiting, panning, and zooming-and then click "Window" > "Scenes." Click the "+" button to add a new scene. Change the view by orbiting a little bit. Click the "Scene 1" scene (the "Scene 1" tab at the top). The view is returned to the position and zoom level determined by "Scene 1."

You can update the properties that are saved on a scene. You can also add new Scenes, and then update each scene with different properties. As you move from one scene to the next, the view changes smoothly from the one set for the first scene to the one set for the second scene, and the different properties you saved for each scene are also enabled. You can use this to add a nice animation quality to the display of your model.

To add a new Scene, right-click the "Scene 1" scene, and then click "Add."

You can store several properties with each scene:

•   The view (camera position, zoom, and field of view).
•   Shadow settings.
•   Whether entities are hidden or displayed.
•   Section planes.
•   Drawing axes.
•   Style Display Settings (display styles and edge effects).
•   Layers.

Scenes only store properties, not geometry. There's only one instance of the geometry in a model, and all Scenes are simply views of that geometry. If you have a scene selected and draw some new geometry, you'll see the new geometry on every scene. The only things you can change from scene to scene are the properties that are stored.

The easiest way to control scene properties is via the Scenes dialog box (open the "Window" menu, and then click "Scenes").

When you add a new scene, it inherits all of the saved properties of the current scene. But if you're on existing scene and make change to any of the scene properties, you must update the scene in order to save the changes. You can also update other Scenes at the same time to save the

same property changes to those Scenes as well.

> *An important note:* Components don't have Scenes. If you create a model with Scenes and share it in the 3D Warehouse, and then you download that model directly into a SketchUp model, it comes in as a component. To be able to see and access Scenes, you must open the model in a new instance of SketchUp, so it opens as a full model rather than as a component in a model:
>
> In the 3D Warehouse, click "Download to SketchUp."
>
> A message asks, "Load this directly into your SketchUp model?" Click "No."
>
> The next message asks, "Do you want to open or save this file?" Click "Open."

Creating an animation

Animations are a series of scenes that are displayed in succession to give a hands-free tour of a model. To create an animation:

1. Select **Window > Scenes.** The Scenes Manager displays.
2. (optional) Type the name of the scene in the 'Name' field. For example, type "Front View" if the scene represents the front view of a building.
3. (optional) Press the **Tab** key. The cursor advances to the Description field.
4. (optional) Type a description of the scene in the 'Description' field. The description could be the address of a building, some detail shown in the scene, and so on.
5. (optional) Deselect any of the **Properties to save** that you do not want saved with the scene. Refer to Scene Manager Options for further information.
6. Click the **Add Scene** button (⊕) to add the scene. A scene tab is added above the Drawing Window. The scene tab has the name of the scene you assigned in step 2.
7. Move to a new location of the model using either the Camera tools or Walkthrough tools.
8. (optional) Repeat steps 2-7 to add additional scenes.
9. Context-click on the first scene tab above the Drawing Window. A context menu appears.
10. Select **Play Animation.** The animation cycles through each scene.

Running an animation

SketchUp contains controls for starting, stopping, and pausing an animation. To run an animation:

1. Select the **View > Animation > Play** menu item. The animation controls are displayed and the animation starts to cycle through your scenes.
2. Press the **Pause** button to pause the animation.

3. Press the **Pause** button to stop the animation.

Note: You can also context-click on a scene tab and select **Play Animation** to begin an animation

Adding a scene

The Add Scene button (⊕) allows you to add a new scene to the current file. To add a scene:

1. Select **Window > Scenes.** The Scenes Manager displays.
2. (optional) Type the name of the scene in the 'Name' field. For example, type "Front View" if the scene represents the front view of a building.
3. (optional) Press the **Tab** key. The cursor advances to the Description field.
4. (optional) Type a description of the scene in the 'Description' field. The description could be the address of a building, some detail shown in the scene, and so on.
5. (optional) Deselect any of the **Properties to save** that you do not want saved with the scene. Refer to Scene Manager Options for further information.
6. Click the **Add Scene** button (⊕) to add the scene. A scene tab is added above the Drawing Window. The scene tab has the name of the scene you assigned in step 2.

Updating a scene

Use the Update Scene button (↻) to update a scene if you have made changes to the scene.

To update a scene:

1. Select the scene you want to update.
2. Click on the properties, within Properties to save, to store with the

scene.

> Warning: Each scene has a series of properties that you can store with that scene (Properties to save). Additionally, when you update a scene, you can choose those properties, of the properties that are stored with the scene, 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 scene (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 scene checked prior to updating a scene.

3. Click on the **Update Scene** button (🔄). An Scene 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 scene.
5. Click on the Update button.

Assigning styles to scenes

You can assign individual styles to each scene in your SketchUp file. To assign a style to a scene:

1. Select the **Window > Styles** menu item. The Styles Manager is displayed.
2. Select a styles collection from the drop-down list, such as 'Assorted Styles.'
3. Click on a style in the collection. The style is selected and applied to your current model.
4. Select **Window > Scenes.** The Scenes Manager is displayed.
5. Click the **Add Scene** button (⊕). A scene is added with the current style.
6. Repeat steps 2 to 5 for additional styles and scenes.

Modifying a style assigned to a scene

To modify a style already assigned to a scene:

1. Select **Window > Styles.** The Styles Manger is displayed.
2. Select the **In Model** styles from the drop-down list. The In Model styles are displayed.
3. Click on one of the In Model styles to modify.
4. Select the **Edit** tab. The Edit panel is displayed.
5. Adjust the style as necessary.
6. Click the **Update Style with changes** button. The style is updated

in all scenes.

Click on the arrow in the upper-right portion of the Scene Manager dialog box to show or hide additional options.

Scenes Manager

The Scenes Manager is used to control the various features of SketchUp scenes. Activate the Scenes dialog box from the Window menu or from the scene tabs.

The Scenes Manger contains a list of all of the scenes for the model. Scenes in this list are displayed in the order in which they will be displayed when running an animation. The following image shows the Scenes Manager on Microsoft Windows:
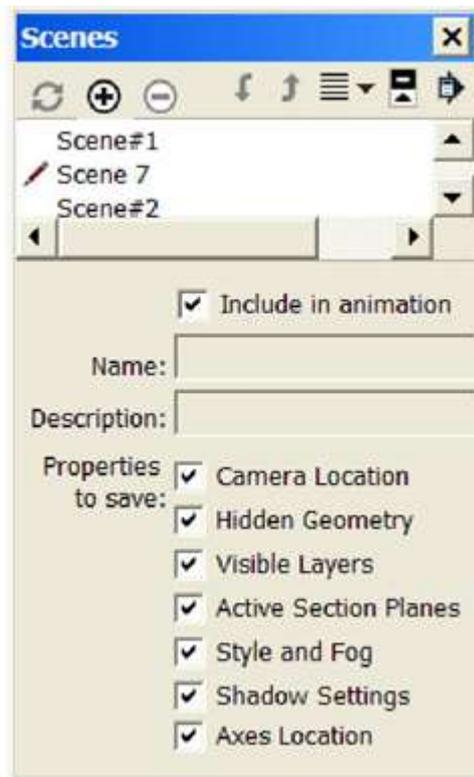
Note: Each scene has a series of properties that you can store with that scene (Properties to save). Additionally, when you update a scene, you can choose those properties, of the properties that are stored with the scene, to update (the dialog box that appears when the Update button is clicked). If you choose to update a property that you are not storing with a scene (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 scene selected prior to updating a scene.

Scenes manager options
Update Scene

Use the Update Scene button (⟳) to update a scene if you have made changes to the scene.

Add Scene button

The Add Scene button (⊕) allows you to add a new scene to the current file.

Delete Scene button

Use Delete Scene button (⊖) to delete a scene from the current file. Select the scene in the list of scenes and click the Delete Scene button to delete the scene.

Move Scene Down button

Use the Move Scene Down button (↓) to move the currently selected scene down in the list.

Move Scene Up button

Use the Move Scene Up button (↑) to move the currently selected scene up in the list.

View options drop-down list

Click on the View options drop-down list to list several options for viewing your scenes in Scenes browser.

Small thumbnails

Click the Small thumbnails menu item (⊞▾) to display small thumbnail representations of your scenes.

Large thumbnails

Click the Large thumbnails menu item (⊞▾) to display large thumbnail representations of your scenes.

Details

Click on the Details menu item (⊟▾) to display large thumbnails

representations of your scenes along with the scene name, author, and description.

List

Click on the List menu item (≡ ▾) to display a list of all scene names.

Refresh

Click on the Refresh menu item to refresh the current list of scenes.

Show/Hide Details button

Use the Show/Hide Details button (▣) to show or hide additional details.

Include in animation

Use the Include in animation checkbox to indicate whether a scene should be used in the Use the Active Section Planes checkbox to store the active section animation. Select a scene and select the Include in Animation option to include the scene in animation. Disable this option for scenes you want to work on but do not want in a animation.

Name

Click in the Name field to name the currently active scene.

Description

Click in the Description field to provide a short description or note for the currently active scene.

Properties to save

The Properties to Save options are used to control the different properties that can be stored with each scene. Properties that are not selected in the Properties to save section cannot be updated with the Update button.

Camera Location: Selecting the Camera Location checkbox stores the point of view, including the zoom distance and field of view, with the scene.

Hidden Geometry: Use the Hidden Geometry checkbox to store the hidden geometry visibility with the scene.

Visible Layers: Use the Visible Layers checkbox to store the visible layers

with the scene.

Active Section Planes: Use the Active Section Planes checkbox to store the active section planes with a scene.

Note: Use different section cuts in successive scenes to create exciting animations with different cross sections of your model.

Style and Fog: Use the Style and Fog checkbox to store the drawing style settings, such as edge rendering and fog, with the scene.

Shadows Settings: Use the Shadows Settings checkbox to store all shadow-related information, including type, time, date, and so on, with the scene.

Axes Location: Use the Axes Location checkbox to store display and position of SketchUp's drawing axes with the scene.

Details menu

Use the Details arrow ( ) to display the Details menu. The following lists all options found on the details menu in both Microsoft Windows and Apple macOS versions of SketchUp.

Add Scene

The Add Scene menu item allows you to add a new scene to the current file.

Update Scene

Use the Update Scene menu item to update a scene if you have made changes to the scene.

## Delete Scene

Use Delete Scene menu item to delete a scene from the current file. Select the scene in the list of scenes and click the **Delete Scene** button to delete the scene.

Use Scene Thumbnails

Select the Use Scene Thumbnails to create a thumbnail image from the current scene (the menu item will have a check next to it). Select the User Scene Thumbnails again to use background Match Photo image for thumbnails (the menu item will not have a check next to it). By default, thumbnails are only created for scenes that contain background Match

Photo images.

Update Scene Thumbnails

Use the Update Scene Thumbnails to update thumbnail images for the currently selected scene. These images are displayed when either the Small thumbnails or Large thumbnails option is selected in the View Options drop-down list.

Add Scene with Matched Photo

Use the Add Scene with Matched Photo to match a new photo. Refer to Match Photo: Introduction for further information.

Show/Hide Details

Use the Show/Hide Details menu to show or hide additional details.

Shadows

The SketchUp Shadows feature is designed to give you a general idea of how the sun and shadows relate to your model during the course of a day and throughout the year. The calculations are based on the location (latitude and longitude, directional orientation of the model, and an associated time zone). The time is not adjusted for Daylight Saving Time. You can enable shadows based on location data captured from the Add Location dialog box or by entering data directly into the Geo-location panel of the Model Info dialog box.

Enabling shadows using location data from the Add Location dialog box

Use the Add Location dialog box to set location and directional orientation for models that represent real-world structures. To enable shadows using the Add Location dialog box:

1. Add the location where you want your 3D model to appear. Refer to Adding a Location for further information. SketchUp is calibrated to the same latitude, longitude, and time zone as the location you choose in the Add Location dialog box. These coordinates appear in SketchUp's Geo-location panel of the Model Info dialog box (Model Info > Geo-location).
2. Note: The time zone in SketchUp is set based on the coordinates of the image. Because some time zones lines zigzag rather dramatically, the time zone for some locations may inaccurate by up to one hour (sometimes longer).
3. Select View > Shadows.
4. To manipulate shadows using the controls in the Shadows Settings dialog box at Window > Shadows.

Enabling shadows using manually input data

Use the Location section of the Model Info dialog box to manually input location information. To enable shadows using manual input data:

1. Select **Window > Model Info.** The Model Info dialog box appears.
2. Select **Geo-location** on the left-hand side of the Model Info dialog box. The Geo-location panel appears.
3. Click on the **Set Manual Location** button.
4. Enter a country in the 'Country' field.
5. Enter a location (city) in the 'Location' field.
6. Enter the latitude in the 'Latitude' field.
7. Enter the longitude in the 'Longitude' field.

8. Click the **OK** button.
9. Select View > Shadows.
10. To manipulate shadows using the controls in the Shadows Settings dialog box at Window > Shadows.

Note: You cannot change the color of shadows.

Shadows settings dialog box

The Shadows 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. The following image contains a screen shot of the Shadows Settings dialog box on Microsoft Windows:
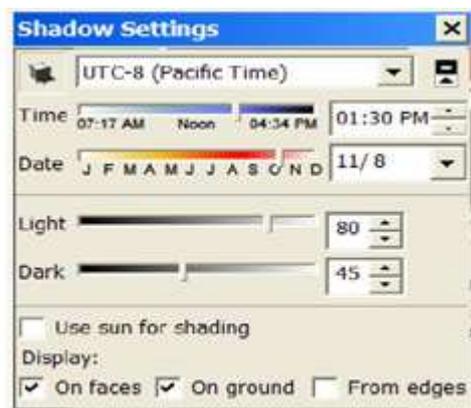
Note: Ensure you have set the proper location for your model set through the Location panel of the Model Info dialog box for accurate shadow casting.

Note: Daylight savings time is not factored into shadow calculations.

Shadow options

SketchUp contains a variety Shadow setting options allowing you to manipulate the use of shadows within your model.

Display shadows button

Use the Display shadows button ( ) to toggle between displaying and not displaying shadows within your model.

Time Zone drop-down list

Select a time zone form the Time Zone drop-down list to identify your

location for accurate shadows.

Time slider

Use Time slider 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 slider

Use the Date slider 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.137Type a date into the date text field to set a precise day. You can specify the date in numerical form (11/8).

Light slider

Use the Light slider to control the intensity of the light in the model. (it lightens or darkens light). This option effectively lightens and darkens illuminated surfaces.

Dark slider

Use the Dark slider 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.
Use sun for shading check box

Click the Use sun for shading check box to use the sun to shade parts of the model while not actually casting shows.

On faces checkbox

Use the On faces checkbox to enable casting of Face shadows. This feature makes intensive use of your 3D graphics hardware and can cause performance degradation.

On Ground checkbox

Use the On ground checkbox to enable the casting of shadows on the ground plane (the red/green plane).

From edges checkbox

Use the From edges checkbox to enable the casting of shadows from

edges that are not associated with a face.

Controlling shadow settings in SketchUp

The SketchUp Shadow tools are designed to give you a general idea of how the sun and shadows affect your model during the course of a day and throughout the year. The calculations are based on the location (latitude and longitude) and directional orientation of the model, as well as the associated time zone. The time is not adjusted for Daylight Saving Time.

Using SketchUp, you can set the location and directional orientation by importing an image from Google Earth:

1. In SketchUp, click Add Location. The Add Location dialog box opens.
2. Search for or zoom in to your desired location.
3. Clicking Select Region will bring up selection pins.
4. Drag and drop the pins to enclose your region.
5. Click Grab, and SketchUp imports that area.
6. The SketchUp model is now set to the same latitude and longitude as the location you chose. The imported terrain image is aligned with the axes in SketchUp (the solid green axis points north and the solid red axis points east). The time zone in SketchUp is approximated based on the coordinates. Because some actual time zones lines zigzag rather dramatically, the automatically calculated time zone for some locations may be off by an hour. In a few cases, the variance can be even longer.

You can also manually set the location, time zone, and solar orientation:

1. Open the **Window** menu.
2. Click Model Info.
3. In the left pane, click **Geo-location**.
4. In the right pane, you can use the **Add Location** option listed above, or you can click **Set Manual location** to set a precise latitude and longitude.

Note: To use the Shadow tools, enable the Shadows toolbar (open the **View** menu, point to **toolbars**, and then click **Shadows**). You can fine tune the shadows in the Shadow Settings dialog box (**Window** > **Shadows**).

Styles

The Styles Browser contains options used to alter how your model and the drawing area is rendered (its edge type face type, background colors, watermarks, and so on). Activate the Styles Browser from the Window menu. The following image contains a screen shot of the Styles Browser on Microsoft Windows:

Applying styles

Select one of many pre-defined styles from the Styles Browser and apply it to your model. Each pre-defined style represents a collection of specific settings that can be applied to your model and the drawing area. To select and apply a pre-defined style:

1. Select **Window > Styles**. The Styles Browser is displayed.
2. Click on the **Select** tab.
3. Click on the 'Styles Collections' drop-down list.
4. Select a **Styles** collection or In Model styles. All of the pre-defined styles in the collection are displayed in thumbnail form.
5. Click on one of the pre-defined styles. The style is applied to your model and the drawing area.

Editing Styles

Styles are comprised of several different settings found under the Edit tab. To Edit a style:

1. Select **Window > Styles.** The Styles Browser is displayed.
2. Click on the **Select** tab.
3. Click on the 'Styles collections' drop-down list.
4. Select a **Styles** collection or In Model styles. All of the pre-defined styles in the collection are displayed in thumbnail form.
5. Click on one of the pre-defined styles. The style is applied to your model and the drawing area. A copy of the style appears in the In Model styles.
6. Click on the **Edit** tab. One of five panels is displayed (Edge, Face, Background, Watermark, and Modeling).
7. Modify settings in any of the edit panels. Refer to Edge Panel, Face Panel, Background Panel, Watermark Panel, or Modeling Panel for further information.
8. Click on the **Update Style** button. The copy of the style in the In Model styles is updated with your changes.
9. Type a name in the 'File name' field and click the **Save** button. The file is saved. Refer to the Open or create a collection context-menu

item for information on how to retrieve this style for use in other SketchUp files.

Sharing styles

Styles are shared in .style files that can be opened by any copy of SketchUp 6 or above. You must create a .style file to share a style with others. To share a style:

1. Create a new folder/directory on your computer to contain your shared styles.
2. Select **Window > Styles**. The Styles Browser is displayed.
3. Select the **In Model** styles from the drop-down list. The In Model styles are displayed.
4. Context-click on the style name you would like to share. The style context menu is displayed.
5. Select the **Save As** menu item. The Save As dialog box appears.
6. Navigate to the folder/directory you created in step 1.
7. Type the name of the style in the 'File name' field.
8. Click the **Save** button. The style file is saved to your newly created folder/directory.

You can now share this style file with others.

Adding a watermark to the drawing area

To add a watermark to the drawing area:

1. Select **Window > Styles**. The Styles Browser is displayed.
2. Click on the **Edit** tab.
3. Click on the **Watermark** panel button. The Watermark panel is displayed.
4. Click on the **Add** button. The Choose Watermark dialog box is displayed.
5. Navigate to an image you will use as a watermark.
6. Click the **Open** button. The image will appear as a watermark in the background of the drawing area. The Create Watermark dialog box is displayed.
7. Click either the **Background** button to have the image appear behind your model as a background watermark, or the **Overlay** button to have the image appear in front of your model as an overlay.
8. Click on the **Next** button.
9. (optional)Select the 'Create Mask' checkbox. Refer to Watermark Settings for further information.

10. (optional) Move the **Blend** slider to the left to fade the watermark. Or, move the **Blend** slider to the right to fade the model.
11. Click on the Next button.
12. Select the button that reflects where you want the watermark to appear on the screen. Refer to Watermark Settings for further information.
13. Click on the **Finish** button.

Styles Browser controls

Use the Styles Browser controls to navigate among styles in your styles collections.

Style thumbnail

The style thumbnail displays the style currently applied to your model.

Style name field

The style name field displays the name of the currently active style.

Style description field

The style description field displays a description of the currently active style.

Display the secondary selection pane button

The Display the secondary selection pane button displays a second select panel below the primary select panel. This feature allows you to display both your styles collections and the In Model materials at the same time. This feature is useful when you want to drag styles between collections and the In Model styles.

Note: You can only apply one style at a time, but you might want multiple styles in the In Model styles. For example, you might have one style that you apply when you are working on the model (because performance is better when you use the style) and another style you use for presenting the model. Styles in the In Model styles are also saved with your SketchUp file.

Create new style button

Use the Create new style button to create a copy of the currently active style in the In Model styles.

Update style button

Use the Update style button to update the copy of the currently active style in the In Model styles.

Select tab

Click on the Select tab to display the select panel. Use the select panel to navigate between style collections.

Back arrow

Use the Back arrow to navigate to the previous location in the hierarchy of collections.

Forward arrow

Use the Forward arrow to navigate to the next location in the hierarchy of collections.

In Model button

Use the In Model button to navigate to the In Model styles collection. The In Model styles collection contains all of the styles currently included with your model file.

Note: You can have styles in your In Model style collection that are not being used by your model.

Collections drop-down list

Select the collections drop-down list to list all of the styles collections in SketchUp. Click on a collection to select the collection.

Details menu

Click on the Details arrow to display the Details menu. See Details menu for further information.

Edit Tab

The Edit tab contains five separate panels: The edge rendering panel, face rendering panel, drawing area styles panel, watermark panel, and other styles panel.

Mix tab

The Mix tab contains five wells corresponding to the five categories of

styles in the styles browser (edge, face, background, watermark, and modeling styles). Click on this tab to display the five wells and a secondary style pane. Select a style from the secondary style pane and click on one or more wells to sample the corresponding settings from that style. For example, click on a style in the secondary style pane and click on the Edge Settings well to sample the edge settings from that style. The style in the style thumbnail changes to reflect the newly sampled settings.

The Styles Browser Face panel

SketchUp contains a variety of face rendering styles allowing you to manipulate the amount of material and textures displayed on the screen.

Front color button

Use the Front color button to set the default color for all front sides of faces. Materials assigned to faces override this setting.

Back color button

Use the Back color button to set the default color for the back sides of faces. Materials assigned to faces override this setting.

Display in wireframe button

Use the Display in wireframe button to display the model as a collection of lines. Faces are not displayed in Wireframe mode.
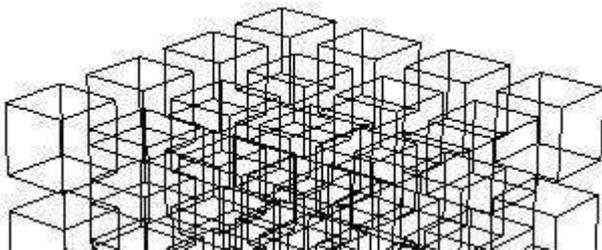
Note: You cannot use face modification tools, such as the Push/Pull tool, on a wireframe rendered model

Display in hidden line mode button

Use the Display in hidden line mode button to display faces in the model without any shading or textures.

Note: 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.
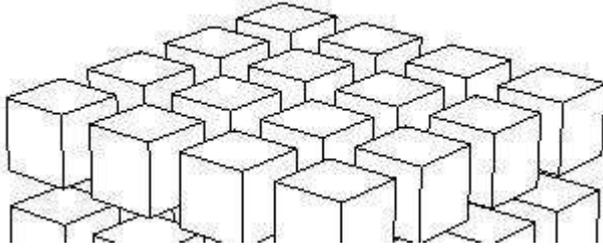
Display in shaded mode button

Use the Display in shaded mode button 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 for front and back face are displayed when no color has been applied to a face.
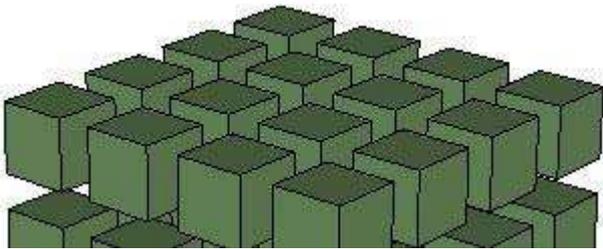
Display in shaded using textures button

Use the Display in shaded using textures button to display the faces In Model with the texture images that have been applied to a face.

> Note: 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.

Display in shaded using all the same button

Use the Display in shaded using all the same button to display the faces with the default front and back face color.

Display X-Ray mode button

Use the Display in X-Ray mode button 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.
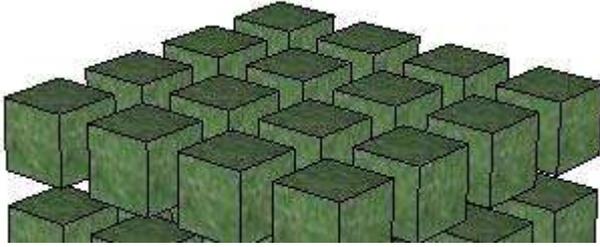
Remember, however, that it is not possible to select and infer faces that would otherwise be hidden.

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.
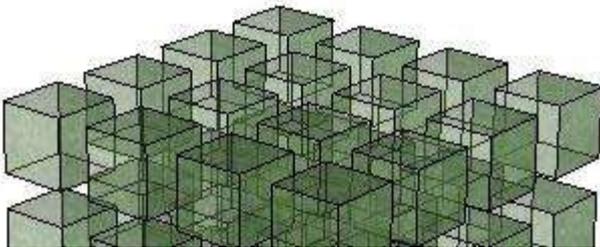
Enable transparency checkbox

Selecting the Enable transparency checkbox enables or disable global material transparency.

Transparency quality

The Transparency quality options allow you 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.

The Styles Browser Background panel

Styles also contain options for configuring the drawing area background, sky, and ground colors.

Background button

Use the Background button to select a background color for SketchUp's drawing area.

Sky checkbox

Use the Sky checkbox to have a sky color that is different from the background color. Use the Sky button to select a sky color.

Ground checkbox

Selecting the Ground checkbox selects a ground color that is different from the background color.

Transparency slider

Slide the Transparency slider 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 checkbox

Selecting the Show ground from below checkbox toggles the display of the ground plane from viewpoints below the horizon.

Styles Browser details menu

Selecting the right arrow next to the styles collections drop-down list opens the details pop-up menu. This menu allows you to perform additional styles-related functions.

Create style

Use the Create style menu item to create a copy of the currently active style in the In Model styles.

Open or create a collection (Microsoft Windows)

Use the Open or create a collection menu item to open an existing collection (or file) or create a new styles collection (folder).

Note: The open or create a collection menu item displays a Browse for Folder dialog box. This dialog box does not display files, only folders. Select the folder or location containing the file and click the **OK** button. Use the Make New Folder button to create a new collection folder.

Open an existing collection (Apple macOS)

Use the Open an existing collection menu item to open an existing collection.

Note: The Open an existing collection item displays a Open dialog box. This dialog box does not display files, only folders. Select the folder or location containing the file and click the **Open** button.

Create a collection (Apple macOS)

Use the Create a collection menu item to create a new styles collection.

Save collection as

Use the **Save collection as** menu item to save a copy of a collection with a new name.

Add collection to favorites

Use the **Add collection to favorites** menu item to add an existing collection to the list of favorite collections (appearing at the bottom of the collections drop-down list).

Remove collection from favorites

Use the Remove collection from favorites menu item to remove a collection from your list of favorites (appearing at the bottom of the collections drop-down list).

Purge unused

Use the Purge Unused menu item to remove all styles in the In Model styles that are not actually used in your model.

Small, Medium, Large and Extra Large Thumbnails

Use the Small Thumbnails, Medium Thumbnails, Large Thumbnails, or Extra Large Thumbnails menu item to change the size of the thumbnails in the Select panel.

List View

Use the List View menu item to display the styles in list form.

Refresh

Use the Refresh menu item to update the styles appearing in the styles browser.

The Styles Browser Edge panel

SketchUp contains a variety of edge rendering styles allowing you to manipulate the edge appearance on the screen.

Note: Endpoints and Jitter edge rendering styles are not available for NPR Edges.

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.

Edges checkbox

Selecting the Edges checkbox activates the display of edges in your model. The following images shows a series of cubes without and then with edges.
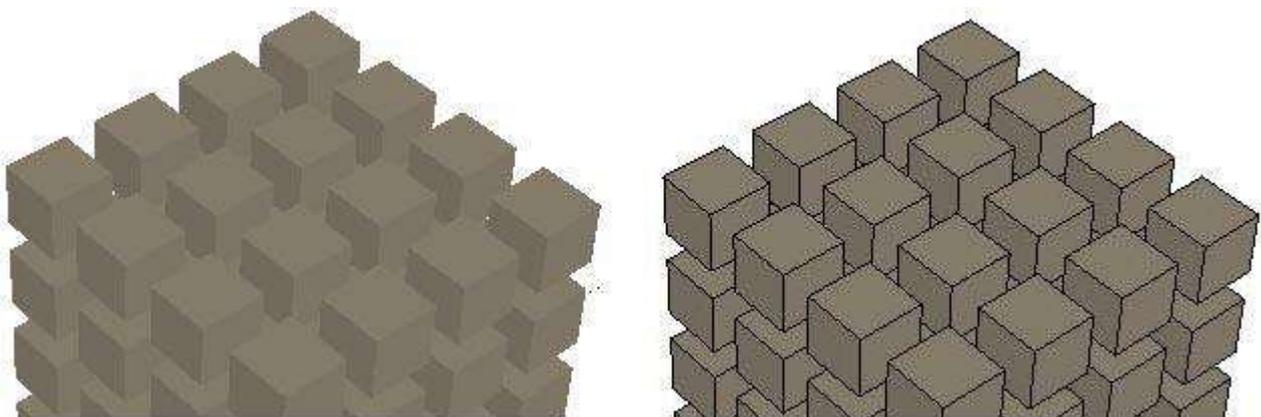
Back Edges checkbox

Selecting the Back Edges checkbox activates the display of edges

obscured by other edges in your model. Obscured edges appear as dashed lines. The following images show a cube without back edges and then a cube with back edges. Note that enabling back edges will disable the X-Ray face style.
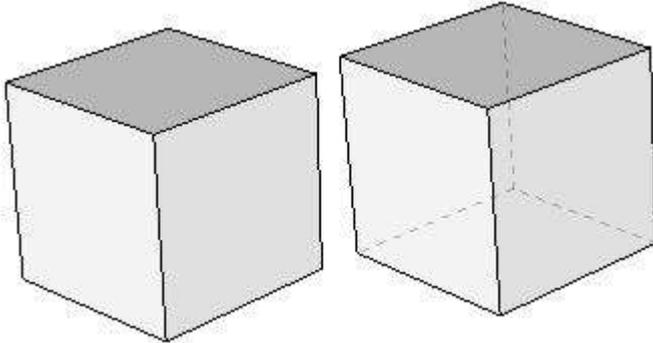
Profiles checkbox

Selecting the Profiles checkbox emphasizes 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.
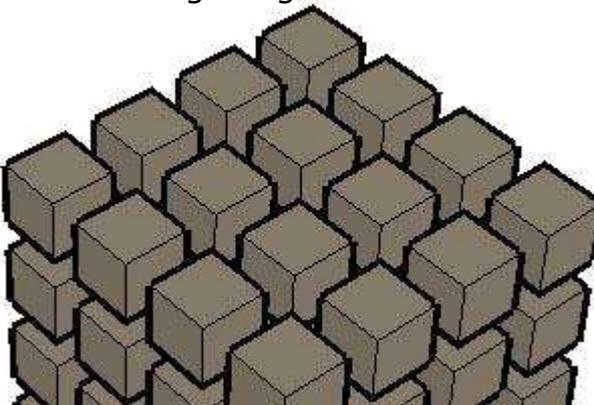
Depth cue checkbox

Selecting the Depth cue checkbox emphasizes 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.
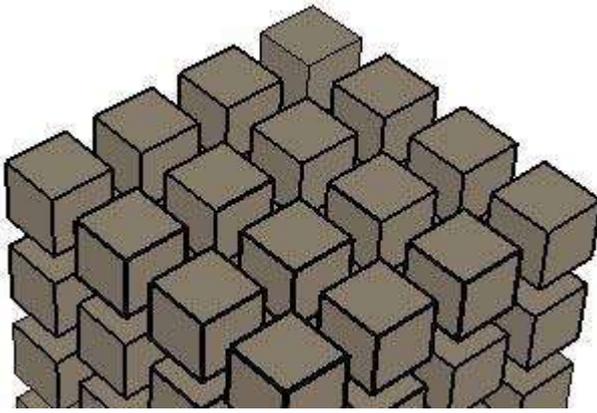
Extension checkbox

Selecting the Extension checkbox extends 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.
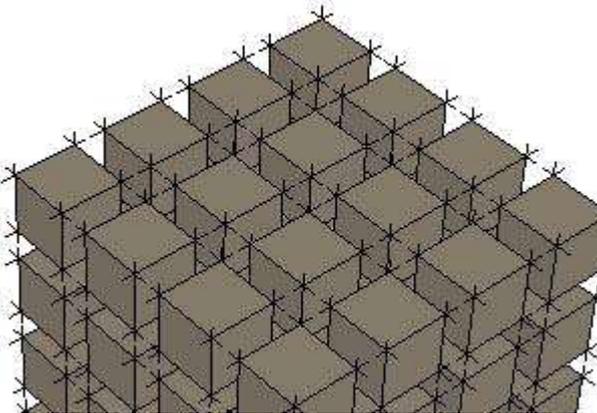
Endpoints checkbox

Selecting the Endpoints checkbox 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.
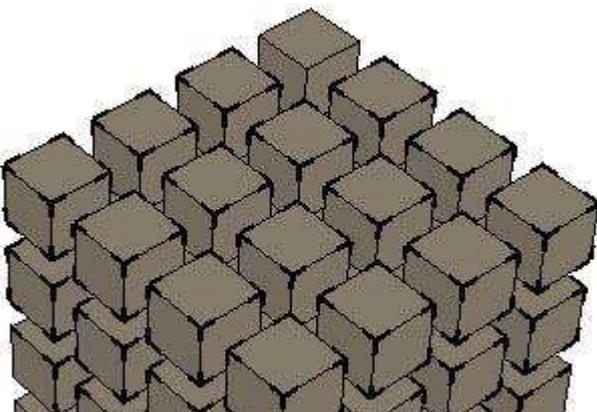
Jitter checkbox

Selecting the Jitter checkbox renders 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.
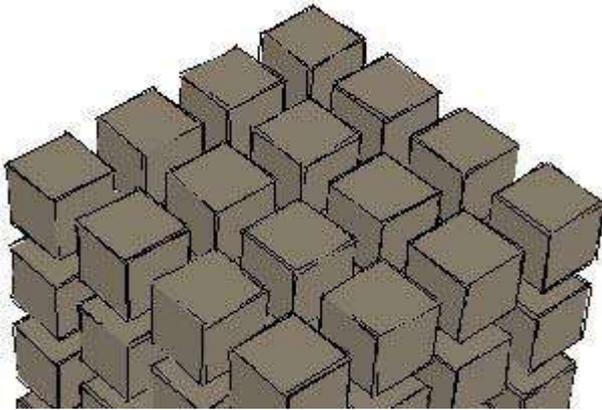
Color

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

Use the All same drop-down list item to display all edges using the Edges color as defined in color swatch.

This option does not actually change any edge color assignments you may have made, but preserves them if you choose to view them again.
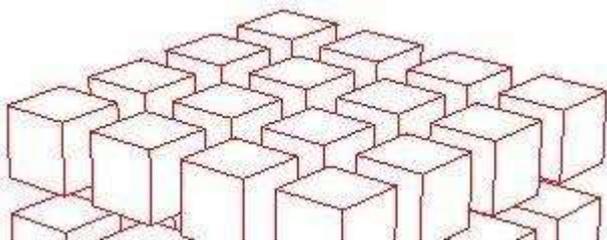
By material

Use the By material drop-down list item to display edges using an assigned material color.

The following image contains edges painted with a grey material.
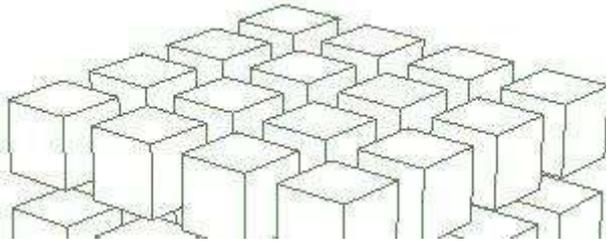
By axis

Use the By axis drop-down list item 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.
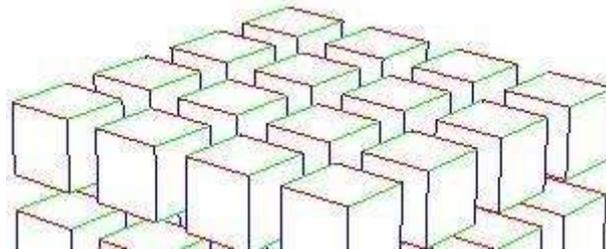
The Styles Browser Modeling panel

The Modeling panel contains styles for how a variety of entities are displayed on the screen.

Selected button

Use the Selected button to set the color of the highlight used to indicate a selection. Use a color that will contrast well with the other colors in your model when selecting a selected color.

Locked button

Use the Locked button to set the color of any locked component.

Guides button

Use the Guides button to set the color of any Guide Line entities in your model.

Inactive section button

Use the Inactive section button to define the color for the currently inactive (not highlighted) Section Plane entities in your model.

Active section button

Use the Active section button to define the color for the currently active (highlighted) Section Plane entity.

Section Cuts button

Use the Section Cuts button to define the color for the active section plane's section slice line.

Section cut width field

Enter a Section cut width to define the thickness (in pixels) of all cut lines in the active section plane.

Hidden Geometry checkbox

Selecting the Hidden Geometry checkbox displays hidden geometry or entities that have been hidden using the Hide menu item or context command. The Hidden Geometry checkbox 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> Selected, UnHide > Last, and Unhide > All menu items.

Color by Layer checkbox

Selecting the Color by Layer checkbox applies 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.

Guides checkbox

Selecting the Guides checkbox displays guide line entities and guide point entities.

Section Planes checkbox

Selecting the Section Planes checkbox displays Section Plane entities.

Section Cuts checkbox

Selecting the Section Cuts checkbox displays section cut effects.

Model Axes checkbox

Selecting the Axes checkbox displays the drawing axes.

Foreground Photo checkbox

Check the Foreground Photo checkbox to display the photo as an overlay image on the model faces. Uncheck the Foreground Photo checkbox to hide the overlay. This option is only applicable during Match Photo Sketching mode.

Opacity slider

Move the Opacity slider to the left to make the photo overlay more transparent. Move the Opacity slider to the right to make the photo overlay more opaque. This option is only applicable during Match Photo Sketching mode.

Background Photo checkbox

Check the Background Photo checkbox to display the photo as a background image to the model. Uncheck the Background Photo checkbox to hide the photo. This option is only applicable during Match Photo Sketching mode.

Opacity slider

Move the Opacity slider to the left to make the photo more transparent. Move the Opacity slider to the right to make the photo more opaque. This option is only applicable during Match Photo Sketching mode.

Fog

Fog options

There are a few options in the Fog dialog box for configuring fog.

Display Fog

Selecting the Display Fog checkbox displays fog in the drawing area. Deselecting the Display Fog checkbox hides fog.

Fog sliders

The left-most fog slider determines where you want the fog to start relative to the camera (your view). Fog can begin right in front of the camera which is zero on the scale or further away from the camera. Move the left-most slider to the right to start fog somewhere beyond the camera.
The right-most slider determines where you want the fog to be at 100% strength (known as zero visibility). Move the right-most slider to the left to establish 100 percent strength closer to the camera (your view). Move the right-most slider all the way to the left to have zero visibility right in front of the camera and extending out to infinity (you wont actually be able to see your model at all and at any distance).

Use Background Color

Selecting the Use background color checkbox uses the currently configured background color as the fog color. Refer to the Background Panel section of the Styles Browser topic for further information on setting the background color. Deselecting the Use background color and clicking on the color swatch selects a different color to be used as the fog color.

Note: Use the OpenGL panel in the Application Preferences dialog box to set hardware acceleration before using fog.

Fog is primarily used as a special effect during presentations. Use the Fog menu item within the Window menu to display the Fog dialog box. The following image shows a screenshot of the Fog dialog box on Microsoft Windows:
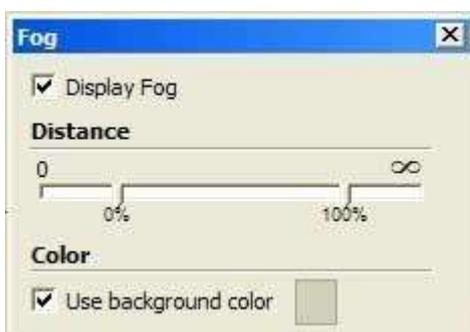
Use the Fog dialog box to apply Fog-like effects to your model. To apply Fog:

1. Select **Window > Fog.** The Fog dialog box appears.
2. Select the 'Enable Fog' checkbox. Fog might appear around your model (depending on your view of the model and the settings of the fog sliders).
3. Adjust the sliders.

Note: Fog sliders will move automatically when you change your point of view, such as when you zoom in or zoom out. As with real fog, the geometry will become more clear as you zoom closer to it, and less clear as you move away.

3D Warehouse

The 3D Warehouse is an online repository for downloading 3D models. These models are a combination of user-generated content (models created by non-SketchUp employees) and in-house content (models created by SketchUp employees). Following are some tasks you can perform with the 3D Warehouse:

- Share your SketchUp, KMZ models with others.
- Find and use components (eg. chairs, tables, cars, boats, ion cannons, doors, windows, etc...), stored in 3D Warehouse, in your SketchUp models. Downloading a model from the 3D Warehouse

You can download individual models from the 3D Warehouse. At this time, however, you can't download collections.

Accessing the 3D Warehouse from within SketchUp

In SketchUp, click **File > 3D Warehouse > Get Models...** or by enabling the **Warehouse** toolbar.To search for models and collections in the 3D Warehouse:

1. Type some keywords, such as "Ferrari car" in the **Search** field.
2. Click the gears icon in the **Search** field to specify if you would like to search for Models or Collections.
3. Click the Search button, which looks like a small magnifying glass. Thumbnail images of the models or collections with your keywords are displayed.

Downloading a model using an Internet browser

1. Visit http://3dwarehouse.sketchup.com
2. Search or browse for a model that you want to download.
3. Click on the thumbnail image (or link) for the model. The model's detail page is displayed.
4. Click on the **Download** button. A list of file types is displayed.
5. Click on the download link next to the file type that you want, such as SketchUp 2014 (.skp) [Download 3 mb]. Depending on your Internet browser, a dialog may appear asking you if you want to open the file with a specific application or save the file. Some Internet browsers are configured to download files automatically to a "Downloads" folder.
6. Select **Open with** and the specific application from the Application drop-down list. Or, select **Save File** to save the file to your computer's hard disk drive.

7. Click the **OK** button. The model is loaded into the specified application or saved to your computer's hard disk drive.

SketchUp settings

- File Menu
- Model Info
- Preferences

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 **Window > Preferences.**

Keyboard Shortcut: Ctrl+N

Open*⁶

Use the Open menu item 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

Save*

Use the Save menu item 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

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⁶ The ***** means : « *See the* Problem Detection *paragraph further on in this chapter* ».

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: If Auto-save is enabled within the General panel of the Preferences dialog box, a backup file is saved in the My Documents (Microsoft Windows) or Library/Application Support/Sketchup 2019/SketchUp/Autosave (Apple macOS) folder, by default. You can have SketchUp automatically save for you at a specific time increment (default is 5 minutes).
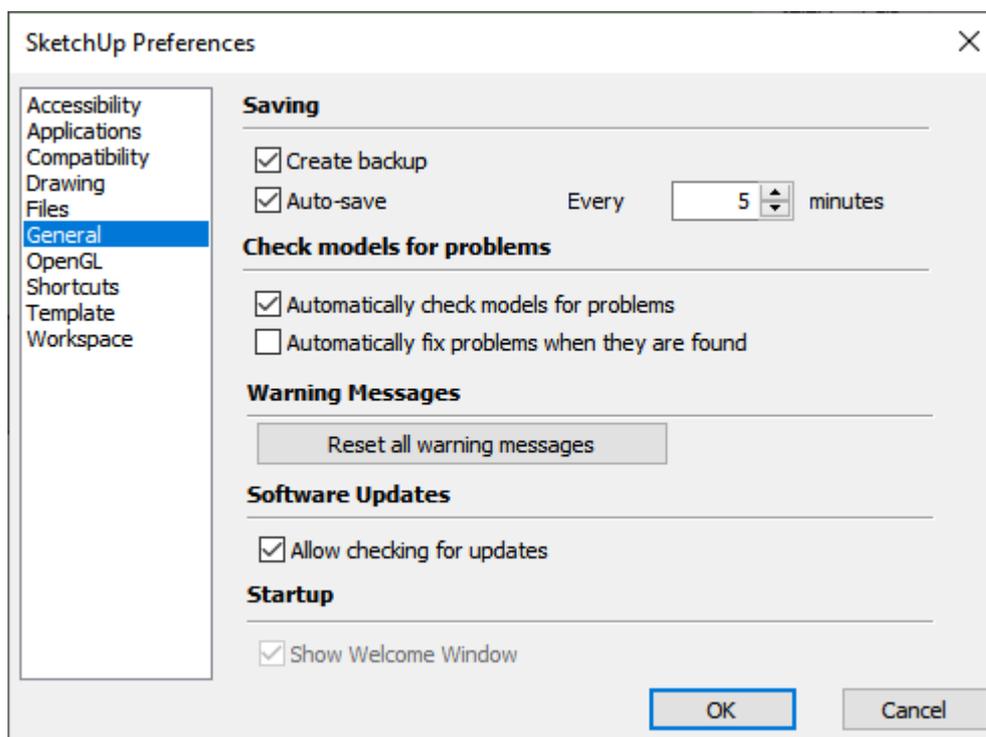
Save As*

Use the Save As menu item 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 become the current file in the drawing window.

Save A Copy As*

Use the Save A Copy As menu item 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.

Save As Template

Use the Save As Template menu item to save the current SketchUp file as a template. This menu item launches a dialog box where you can name the template and set the template as the default template (to be loaded every time you launch SketchUp).

Revert

Use the Revert menu item to revert your current document to its last saved state.

Send to LayOut

Use the Send to LayOut menu item to send the current model to LayOut.

Preview in Google Earth

Use the Preview in Google Earth menu item to quickly view your model in Google Earth while working on the model.

Geo-location

The Geo-location submenu contains menu items for Geo-locating your models.

Add Location...

Use the Add Location menu item to display the Add Location dialog box for selecting a location for your model. This menu item is only available when a location has not been added.

Clear Location

Use the Clear Location menu item to remove the location from your model.

Show Terrain

Use the Show Terrain menu item to toggle the Google Earth snapshot image between a 2D and 3D image.

3D Warehouse

The 3D Warehouse submenu contains menu items for accessing the 3D Warehouse repository of models.

Get Models

Use the Get Models menu item to download a model from the 3D Warehouse.

Share Model

Use the Share Model menu item to post your SketchUp model file and corresponding KML file to the 3D Warehouse. The 3D Warehouse is a repository where models can be shared with other Google Earth or SketchUp users.

Upload Component

Use the Upload Model button to post the currently selected component to the 3D Warehouse.

Export

Use the Export submenu 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

Use the 3D Model menu item to export you SketchUp file to 3D formats, such as KMZ files.

Additional file formats are available for export using SketchUp Pro.

2D Graphic

Use the 2D Graphic menu item to export 2D bitmap and dimensionally accurate, resolution independent, 2D vector drawings. Pixel-based images can be exported in JPEG, PNG, Epix, BMP, TGA, and TIFF file formats.

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

Use the Export 2D Section Slice menu item to output dimensionally

accurate 2D section slices in standard vector formats.

Animation

Use the Animation menu item to export a pre-rendered animation file containing the scene sequence you have created. This option makes it easy to archive your animations to CD or DVD and to create smooth animations of complex models.

Import

Use the Import menu to import information from other files into your SketchUp drawings.

Print Setup

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

Print Preview

Use the Print Preview menu item 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

Generate Report

The Generate Report menu item is used to generate a report of all attributes contained in a dynamic component.

(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.

* 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. We recommend 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 Open, Save, Save As, and 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 contains the option to quit SketchUp and send a report. We recommend 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.

Model Info

- Animation
- Classifications
- Components
- Credits
- Dimensions
- File
- Geo-location
- Rendering
- Statistics
- Text
- Units

The Model Info dialog box allows you 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.

Animation

The Animation panel allows you to set properties for animations. The following image shows the Animation panel of the Model Info dialog box on Microsoft Windows:
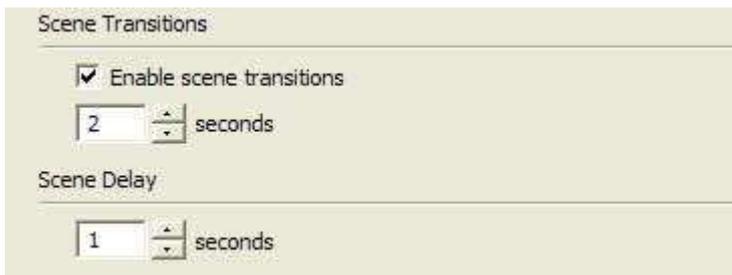
Scene Transitions

The Scene Transitions section of the Animation panel allows you to determine the amount of time it takes to transition from one scene in the animation to another scene.

Enable scene transitions: Use the enable scene transitions checkbox to enable scene transitions.

Seconds: Enter the number of seconds that will transpire before the next

transition changes to another scene.

Scene Delay

The Scene Delay section of the Animation panel allows you to determine the amount of time an animation will spend on each scene before transition to another scene.

Seconds: Enter the number of seconds that should transpire before the transition begins to another scene.

Classifications

Classifies components and groups.

For more on Classification, read this[7].

Components

The Components panel allows you 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. The following image shows the Components panel of the Model Info dialog box on Microsoft Windows:
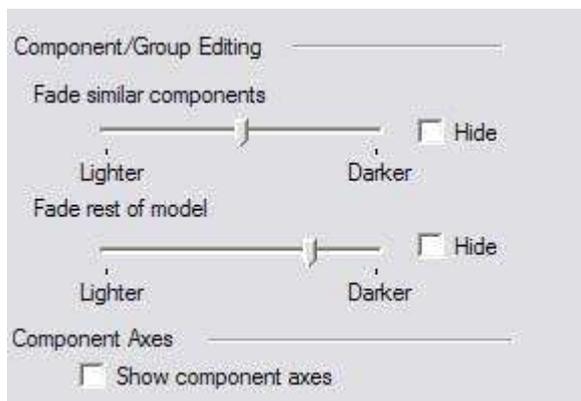
Component/Group Editing

The Component/Group Editing portion of the Components panel allows you to define how geometry outside a component appears when editing a component.

Fade similar components: Sliding the Fade similar components slider controls the degree of fading of other visible instances of the component

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[7] https://help.sketchup.com/en/sketchup/classifying-objects

you are editing. Use the Hide checkbox to toggle the display of components similar to the one you are editing.

Fade rest of model: Sliding the Fade rest of model slider controls the fading of geometry unrelated to the group or component you are editing. Use the Hide check box to toggle the display of the rest of the model while editing a component or group.

Show component axes: Components contain their own axes. Use the Show component axes checkbox to display component axes.

Credits

The Credits panel allows you to view contributors toward the current model. This panel also allows the current users to claim credit toward the current model.

Model Authors

The Model Authors portion of the Credits panel displays the currently logged in user. User's must log in using their Trimble Account to claim credit toward a model.

Claim Credit button: Use the Claim Credit button to claim credit on the currently selected model.

Component Authors

The Component Authors portion of the Credits panel displays the list of contributors toward the current model.

Dimensions

The Dimensions panel allows you to change the appearance and behavior of Dimension entities in your model. The following image shows the Dimensions panel of the Model Info dialog box on Microsoft Windows:
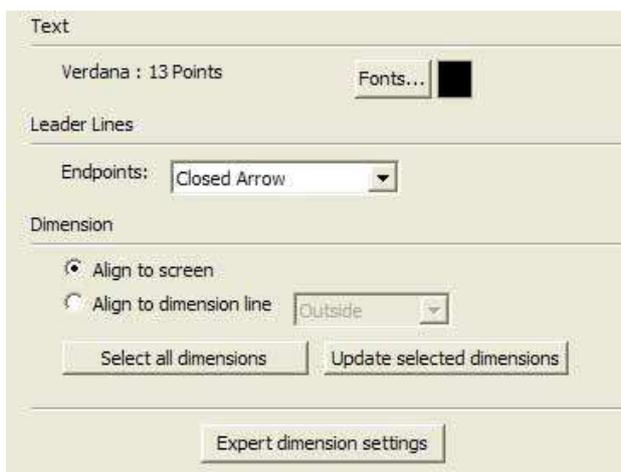
Text

The Text portion of the Dimensions panel allows you to select the font used by Dimension entities in the drawing area.

Fonts: Use the Fonts button to select the font to be used for all Dimension entities in SketchUp. Use the font color button to choose a font color for Dimension entities.

Leader Lines

The Leader Lines portion of the Dimensions panel allows you to choose an end point style for your leader lines.

Endpoints: Select the end point style for all of your end points from the Endpoints drop-down list. These styles are None, Slash, Dot, Closed Arrow and Open Arrow.

Dimension

The Dimension portion of the Dimensions panel contains default display options for Dimension entities.

Align to screen: Use the Align to screen button to allow Dimension entities to rotate as you orbit the model (always face the camera).

Align to dimension line: Use the Align to dimension line button 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.

Select all dimensions: Use the Select all dimensions button to select all dimension entities in the model.

Update selected dimensions: Use Update selected dimensions to update the currently selected entities with the currently selected dimension settings.

Expert dimension settings

Use the Expert dimension settings button to display the Expert Dimension Settings dialog box.

Hide when foreshortened: Use the Hide when foreshortened checkbox

to allow Dimension entities to hide when they become foreshortened. Slide the slider 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 an unclear Dimension text. Use the Hide when too small checkbox to automatically hide dimensions when they are hard to read. Slide the slider to set the size that will cause Dimension entities to be hidden.

Highlight Non-Associated Dimensions: Use the Highlight non-associated dimensions checkbox 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 allows you 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. The following image shows the Files panel of the Model Info dialog box on Microsoft Windows:

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: Click in the Description field to enter a description for your model.

Redefine thumbnail on Save: Use the Redefine thumbnail on save checkbox 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 (Microsoft Windows).

Alignment

The Alignment portion of the File panel allow you 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:Select a menu item from the Glue to menu to identify 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: Use the Cut opening checkbox to cause your model to 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: Use the Always face camera checkbox 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.

Shadows face sun: This option is only available when the Always face camera option is enabled. Selecting the Shadows face sun checkbox causes shadows to be cast from the components' current position as though the components were facing the sun. The shadow shape does not

change as the component rotates to face the camera. This option works best with components that have narrow bases (such as trees). This option does not work well with components that have wide bases (such as people in mid stride).

> Note: Ensure the component's axis is positioned at the bottom center of the component for best results.

Deselect this option to cause the shadow to be cast from the component's current position. The size of the shadow changes based on view point.

Geo-location

Models intended for Google Earth contain geographical information identifying their location in the world. The Geo-location panel allows you to view and specify a location for your model.
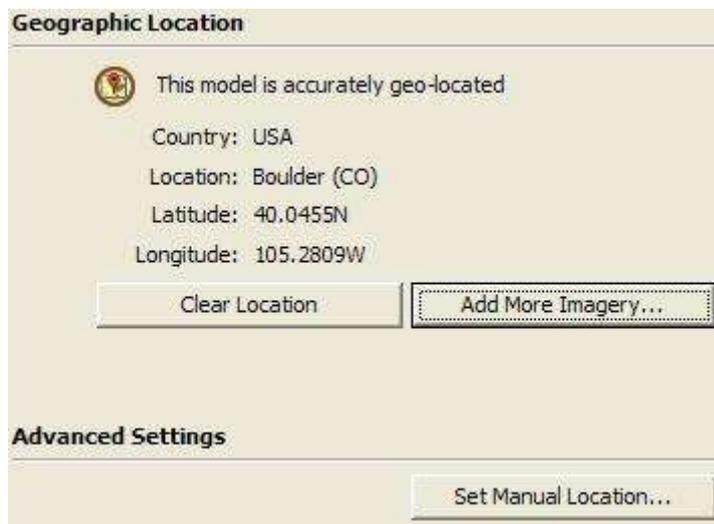
Geographic Location

The Geographic Location portion of the Location panel allows you to assign a geographic location to your model.

Add Location button: Use the Add Location button to display the Add Location dialog box for selecting a location for your model.

Clear Location button: Use the Clear Location button to remove the location from your model. This option is only available for a model that has geographical information.

Add More Imagery button: Use the Add More Imagery button to display the Add Location dialog box allowing you to choose additionaly imagery adjacent to your location to add to the model. Refer to Adding a Location for further information.

Advanced Settings

The Advanced Settings panel allows you set a manual location for your model.

Rendering

The Rendering panel allows you to enable anti-aliased textures.

Use Anti-Aliased Textures

Use the Use Anti-Aliased Textures checkbox to attempt to improve performance and quality of textures.

Statistics

The Statistics panel allows you 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.
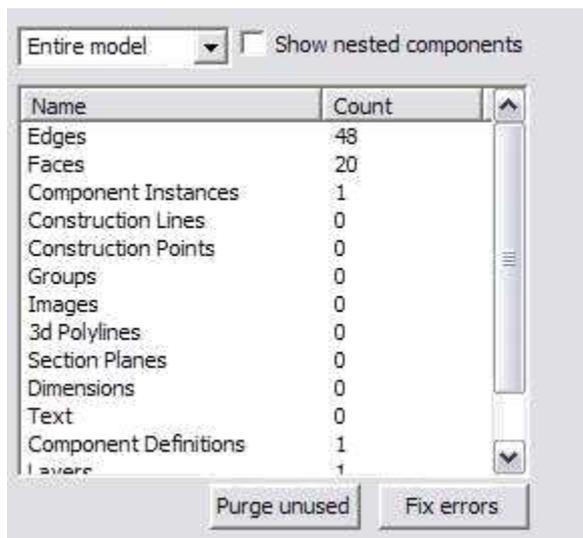
Entire model and Only components

The Entire model and Only 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

Use the Show nested components checkbox to include individual elements

within placed components in the statistics totals.

Purge Unused

Use the Purge Unused button to remove any unused components, materials, image objects, layers, and other extraneous information from your file.

Fix Problems

Use the Fix Problems button to find minor issues with your SketchUp model due to the infinite flexibility that SketchUp provides when designing in 3D. Use 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 allows you to change the appearance and behavior of Text entities in your model.
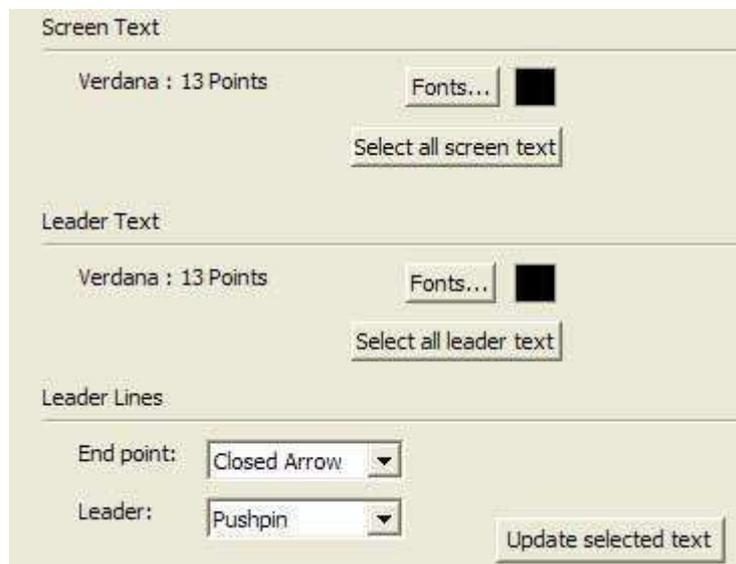
Screen Text

Screen text is text that is not attached, using a leader line and arrow, to a specific piece of geometry. The Screen Text portion of the Text panel allows you to select the font used by screen text in the drawing area.

Fonts: Use the Fonts button to define the font to be used for all screen text in SketchUp. The Font dialog box, containing the font, font style, and size for your dimensions font, is invoked.

Text color swatch: Click on the text color swatch to select the default display color for any unpainted screen text (those painted with the default material).

Select all screen text: Use the Select all screen text to select all screen text in the drawing area.

Leader Text

Leader text is text that is attached, using a leader line and arrow, to a specific piece of geometry. The Leader Text portion of the Text panel allows you to select the font used by leader text in the drawing area.

Fonts: Use the Fonts button to define the font to be used for all leader text in SketchUp. The Font dialog box, containing the font, font style, and size for your dimensions font, is invoked.

Leader text color swatch: Click on the leader text color swatch to select the default display color for any unpainted leader text (those painted with the default material).

Select all leader text: Use the Select all leader text to select leader text in the drawing area.

Leader Lines

The Leader Lines options allow you to identify the type of leader line used with Text. Text might or might not have leader lines.

End Point: Use the End Point drop-down menu to select the end point type for the leader line. Choose between None, Dot, Closed Arrow, and Open Arrow.

Leader: Use the Leader drop-down menu 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 rotates with your model as you change your view.
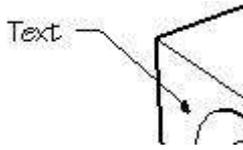

Update selected text: Use the Update selected text button to apply the current settings to any selected Text entities.

Units

The Units panel is allows you to set defaults for linear and angular unit measurements.

Note: You can override the default units using the Measurements toolbar to work in different units simultaneously.
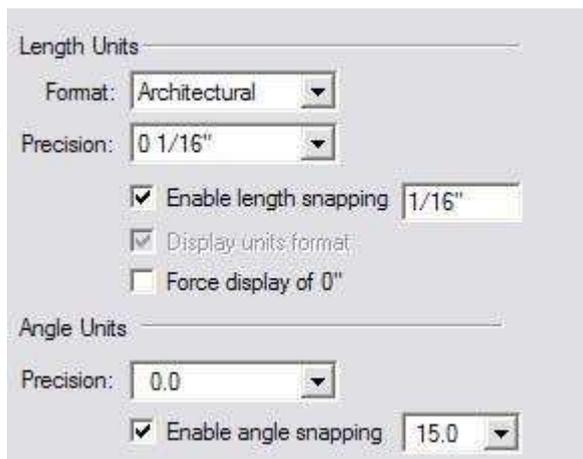
Length Units

The Length Units section of the Units panel allows you to set the default units for your model.

Format: Use the Format drop-down menu to set the type of units displayed for measurements, dimensions, and within the Measurements toolbar. 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: Use the Precision drop-down menu 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: Use the Enable length snapping checkbox to snap lines and other entities to the specified sized increments. Length snaps can be overridden by entering exact values into the Measurements toolbar. 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: Use the Display units format checkbox to display unit markers, such as the double quotes (") for inches. This option is only available when the format is set to Decimal.

Force Display of 0": Use the Force display of 0" checkbox 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 portion of the Units panel allow you to define how units are displayed for angled measurements

Precision: Use the Precision drop-down list 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 : Use the Enable angle snapping checkbox 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: Use the Snap angle drop-down list to choose the snap increment for angular measurements and rotations.

Preferences

- Accessibility
- Applications
- Compatibility
- Drawing
- Files
- General
- OpenGL
- Shortcuts
- Template
- Workspace

Accessibility

For people who experience color blindness, SketchUp makes seeing color-based modeling cues easier than ever before. Here are the features that make 3D modeling more accessible:

- New preferences settings: On the new *Accessibility* pane in the *Preferences* dialog box, you can customize the colors of the drawing axes and inference colors.
- A new preset style: When you apply the new *Color Blind* style to a model, color-based modeling cues stand out better for people who have some degree of color blindness.

The following images show the new *Accessibility* pane and an example of how customized axes colors appear with the *Color Blind* style applied. For help setting up SketchUp and your model styles, see *Changing colors of selected items and other on-screen aids* and select the Help Center documentation for SketchUp 2019.
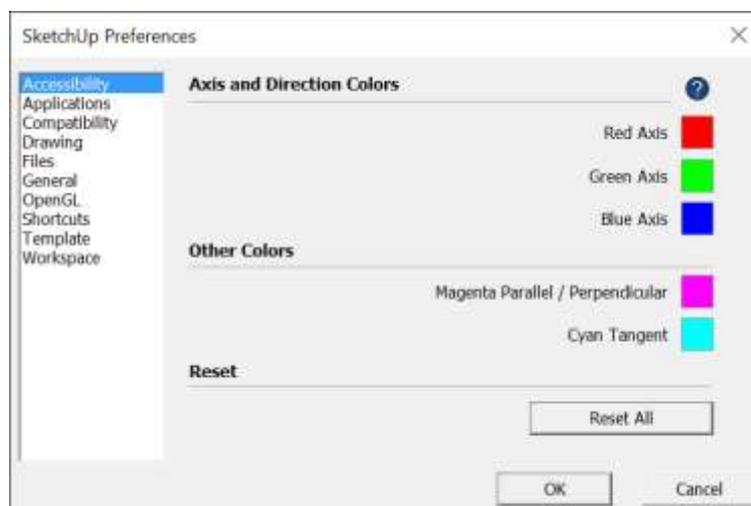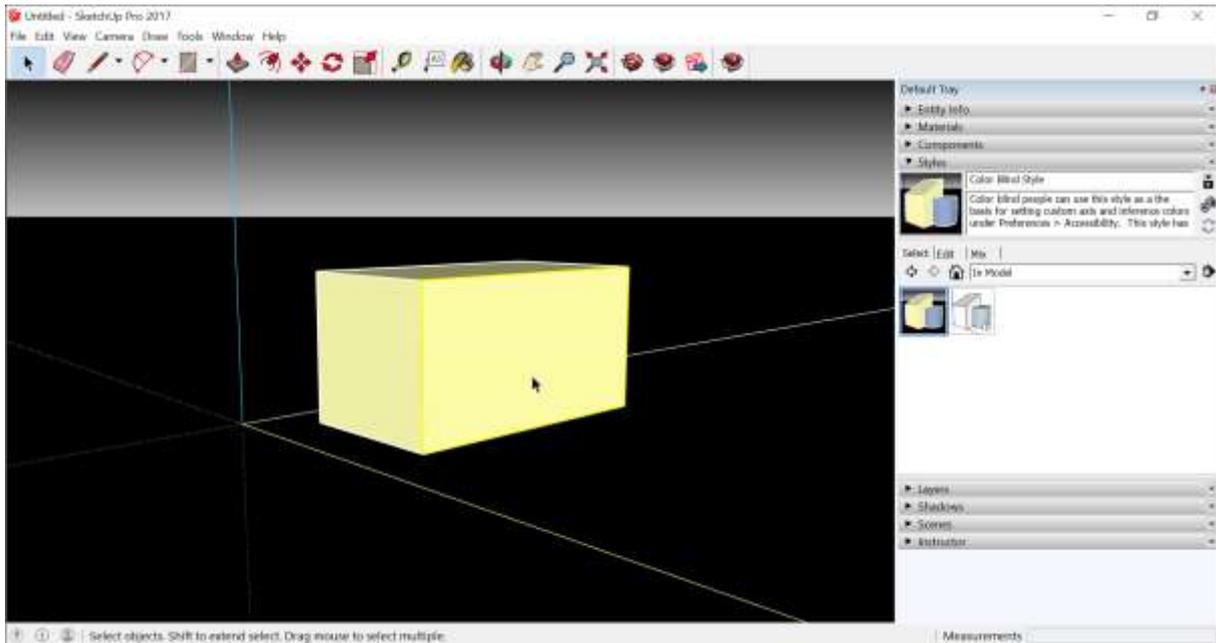
Applications

Set the default image editor.

Determines what image editor opens when you want to edit a texture in an external editor. Click *Choose* and select an application on your operating system.
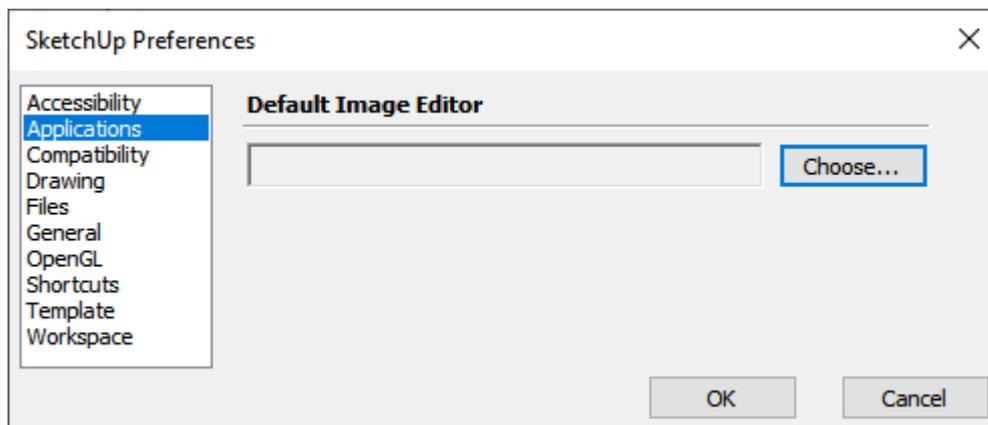
Compatibility

- Component/Group Highlighting

Bounding box only: Display a component bounding box and its edges in different colors. The *Bounding box only* feature is disabled by default. Select the *Bounding box only* checkbox, and a component's edges appear

in a different color from the component box when you select the component.

- Mouse wheel style

Invert: Select the *Invert* checkbox to invert mouse wheel direction.

Drawing

- Click Style

Click-Drag-Release radio button: Select this option if you want the *Line* tool to draw a line only if you click and hold the mouse button to define the line's start point, drag to extend the line, and release the mouse to set the line's end point.

Auto Detect radio button: When this option is selected (it's the default), you can either click-drag-release or click-move-click as necessary.

Click-Move-Click radio button: Force the *Line* tool to draw by clicking to define the line's start point, moving the mouse to extend the line, and clicking again to establish the line's end point.

Continue Line Drawing check box: When either *Auto Detect* or *Click-Move-Click* is selected, you can choose whether to select or deselect this checkbox. (It's selected by default.) When the checkbox is selected, the *Line* tool treats an end point as the start of a new line, saving you the extra click required set a new start point. If that behavior isn't your cup of tea, deselect the checkbox. Then go enjoy a cup of tea, knowing that the *Line* tool now works the way you always wanted.

- Miscellaneous

Display crosshairs: Display crosshairs on your cursor that correspond to axes colors. Select the *Display Crosshairs* checkbox in the *Miscellaneous* area.

Activate Paint Bucket Tool: Activate the *Paint Bucket* tool after you click a material swatch (Apple macOS only). This checkbox is enabled by default. Deselect *Activate Paint Bucket Tool* to disable it.

Disable Pre-pick on Push/Pull Tool: Toggle the Push/Pull tool's pre-pick feature. Pre-pick is enabled by default so that you can select a face, orbit, and push/pull the face even when the selected face doesn't appear in the current view. Select *Disable Pre-pick on Push/Pull Tool* to disable the feature.

Files

In SketchUp, the *Preferences* dialog box includes a *Files* option in the sidebar on the left. The *Files* pane enables you to set the default save locations for models, components, materials, styles, and more.

If the folder where you want to save certain files doesn't yet exist, create the folder using Windows Explorer or Finder first. After the folder is ready, follow these steps in SketchUp:

1. Select **Window > Preferences**.
2. Select *Files* in the sidebar on the left.
3. In the **Files** pane, click the **Change File Location Preference** icon ( ) and a *Select Folder* dialog box appears.
4. Navigate to the folder that you want to designate as the default location for the related files.
5. Click **Select Folder** in the *Select Folder* dialog box and click **OK** again in the SketchUp *Preferences* dialog box. Going forward, the files for that option are saved in the folder you specify.

Tip: To quickly navigate to a default folder, click the **Open This Folder** icon ( ) and the location opens in Windows Explorer or Finder.

The following table outlines how each option changes your default locations.

Option	Changes the Default Location For ...
Models	Opening or saving a model
Components	Opening and saving a collection via the Components browser
Materials	Opening or creating a collection via the *Materials* browser
Styles	Opening or creating a collection via the *Styles* browser
Texture images	Inserting images with the *File > Insert > Image As Texture* command
Watermark images	Images that you can use as a watermark
Export models	Models that that exported with the *File > Export > 3D Model* command
Classifications	Files used to set up classification systems for the *Classifier* feature
Templates	SketchUp template files you customized and saved to your hard drive

General

In SketchUp, you can set a few preferences for how the software works overall and how files are saved.

To access these preferences, select **Window > Preferences** (Windows) from the menu bar. Most of these preferences are on the *General* pane, which you click in the sidebar on the left. As shown in the following figure, your options include *Saving* preferences at the top and *Software Updates* preferences at the bottom.
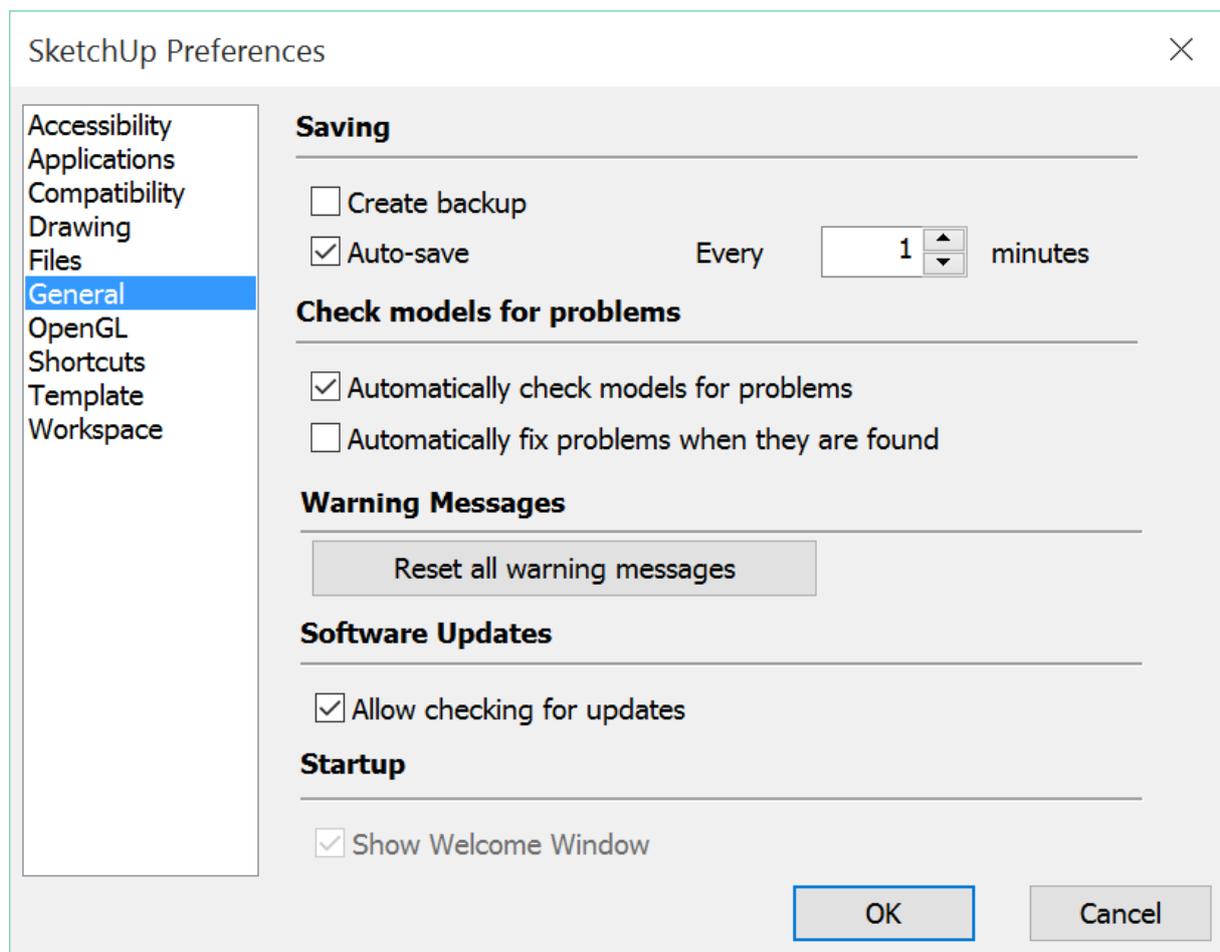
- Saving

Here's a quick look at the *Saving* options on the *General* preferences pane:

Create backup: By default, this option is selected because it helps you recover your work if anything happens as you're creating a 3D model. When *Create backup* is enabled, SketchUp automatically creates a backup file when you save a model. The backup file is the previously saved version of the file and lives in the same folder as the model file. For

example, if your model file is hotel.skp, on Microsoft Windows, the backup is hotel.skb, and on Apple macOS, the backup file is hotel~.skp. On either operating system, you find both files in the same folder.

Auto-save: This option is also enabled by default and helps you recover any changes you make to a model if you experience computer problems. When enabled, *Auto-save* tells SketchUp to automatically save changes to your model into a temporary file at specific time interval. By default, the interval is 5 minutes, but you can make that shorter or longer if you like. Just enter a new interval in minutes in the text box. Note that the auto-save file is preserved only if SketchUp suddenly crashes. If SketchUp closes successfully, the auto-save file is deleted.

Tip: The SketchUp team strongly recommends leaving Auto-save enabled, because it helps you keep your work. During the auto-save, SketchUp checks your model for unrecoverable errors so that a good auto-save file isn't overwritten. In the rare case that SketchUp finds unrecoverable errors in your model, a dialog box appears with the option to quit SketchUp and send a report. Click the Quit SketchUp and Send Report button to terminate SketchUp and preserve your valid auto-saved file. This report contains valuable information regarding the unrecoverable errors.

Note: Want to find an auto-saved file? Recovered files will always be accessible from the Welcome Window's main screen. If you're seeing a recovered file, you have the ability to open it and re-save it to a new location. If you've disabled the Welcome Window you can always access it by going to Help > Welcome to SketchUp.

- Check models for problems

The *General* preferences pane includes a section called *Check model for problems*, where you find the following two options:

Automatically check models for problems: This option is selected by default and enables SketchUp to check your model before it auto-saves, as mentioned in the preceding section. When this option is selected, SketchUp checks for problems when your model is loaded or saved. Because SketchUp offers so much flexibility for designing a 3D model, it's important to check and fix major or minor issues and thus optimize your model's performance. Although deselecting this option isn't recommended, if you do so, you can check for problems manually by selecting *Window > Model Info*, selecting the *Statistics* panel, and clicking the *Fix Problems* button.

Automatically fix problems when they are found: This option is deselected by default, so that when SketchUp finds a problem, you see a dialog box that asks whether and when you'd like to fix the problem. If

you select this option, you don't see the dialog box, and SketchUp fixes the problem behind the scenes.

- Warning Messages

Reset all warning messages[8].

- Software Updates

Allow checking for updates: On the *General* preferences pane, the *Software Updates* section has only one option, *Allow checking for updates*, which is enabled by default. When enabled, SketchUp checks for updates when you're connected to the Internet.

- Startup

Show Welcome Window at startup: By default, this option is selected because, coupled with the auto-save setting, it helps you recover your work if anything happens as you're creating a 3D model. You'll find *Recovered Files* in the *Welcome* window as well as be able to start new files by choosing your desired template on launch every time you open SketchUp.

OpenGL

- OpenGL Settings

OpenGL stands for Open Graphics Library, and it's the API (application programming interface, which is like an application's building blocks) used to render SketchUp's 3D graphics.

Anti-aliasing is a technique for making jagged edges on graphics look smoother.

Now that you know what OpenGL and anti-aliasing are, you have an idea of what the OpenGL and anti-aliasing preferences do. If SketchUp has performance issues on your computer, adjusting a few OpenGL settings may help SketchUp render your model faster.

To access the OpenGL preferences, select **Window > Preferences** (Microsoft Windows) or **SketchUp > Preferences** (Apple macOS). Then select the OpenGL option in the sidebar on the left. The following list outlines the settings you find and what each one does:

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[8] « *It is there so that you can restore the warning dialogs if you have ticked them to not show. Things like changed style settings in scenes and the naming dialog for section cuts etc* » (From the SketchUp Community Forum).

Multisample Anti-Aliasing: Multisample anti-aliasing (MSAA) can produce a high-quality image. The higher the setting, the less jagged the lines but the more computer power required to display the image. If you need a high-quality rendering of your model, choose a value greater than 0x from this drop-down menu.

Use Fast Feedback: Fast feedback improves SketchUp's performance, especially if you're working on a large model. When you start SketchUp, it checks whether your graphics card supports fast feedback combined with multi-sample anti-aliasing. If so, this option is selected by default. If this option is unselected and disabled, you may be able to enable it by choosing a different multi-sample anti-aliasing setting.

In the Model Info window, you can toggle the anti-aliasing of textures on or off. (Select **Window > Model Info** and select the *Rendering* option in the sidebar on the left.) By default, the *Use Anti-Aliased Textures* checkbox is selected, because this feature improves SketchUp's performance and textures' appearance. If you see blurry textures or unusual display effects, deselecting this checkbox may improve your model's appearance.

Use Maximum Texture Size: By default, this setting is deselected, because using maximum texture sizes slows down SketchUp. (Textures are images applied to your model.) Specifically, when this checkbox is deselected, SketchUp won't render graphics at a resolution higher than 1024 x 1024. You might select this option to see an image at a resolution higher than 1024 x 1024. However, if you have enabled this option and SketchUp seems slow, deselect this checkbox.

- Capabilities

Graphics Card Details: The *Graphics Card Details* button displays the computer graphics card details and any OpenGL warnings.

(see picture next page)
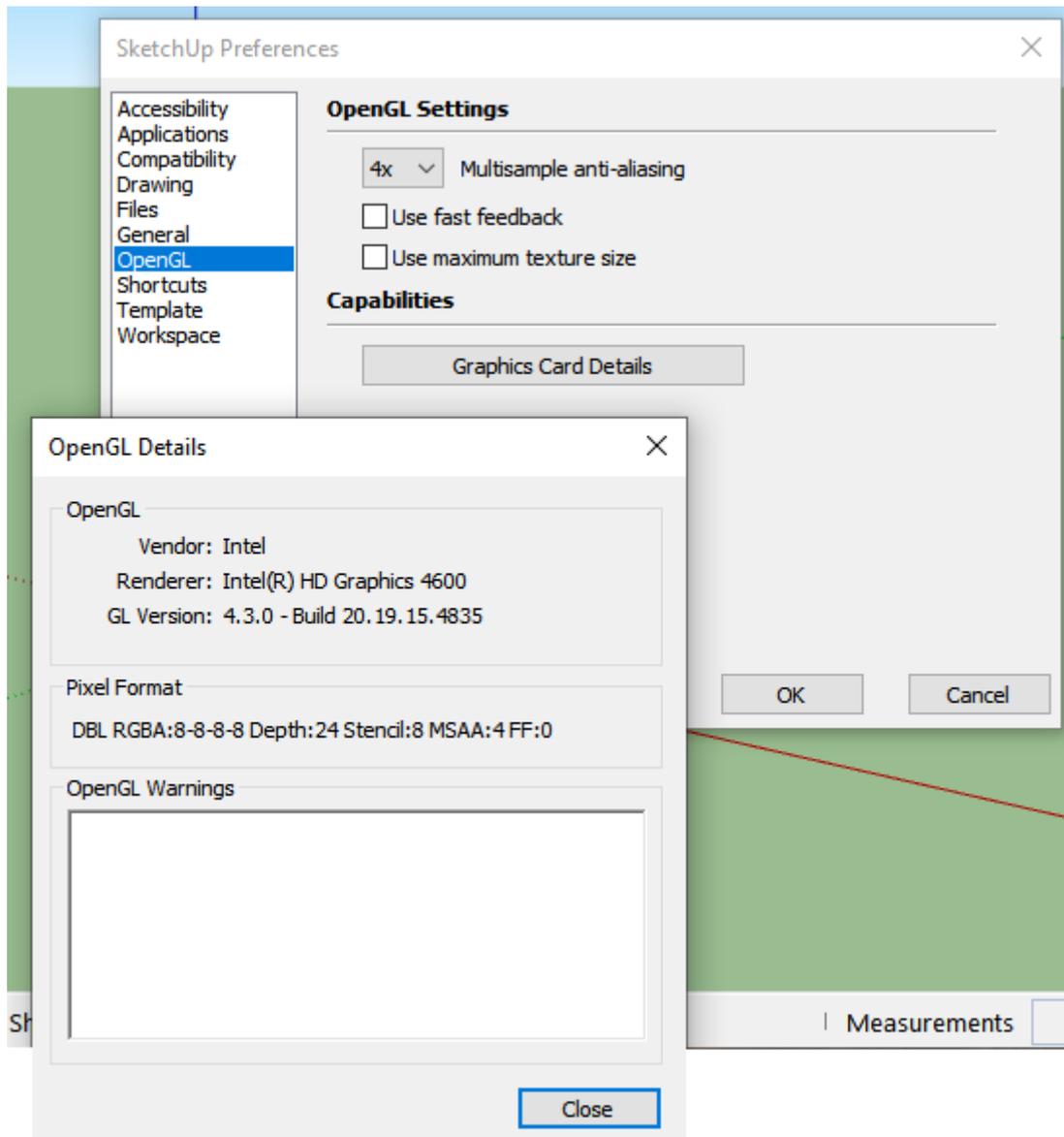
Shortcuts

In SketchUp, you can assign keyboard shortcuts to the commands you use most often, so that the commands are literally at your fingertips.

For the most part, you can customize the keyboard shortcuts however you like, but here are a few guidelines to help you understand what you can and can't do as you assign shortcuts:

- You can't start with a number because that would conflict with the functionality of SketchUp's *Measurements* box, and you can't use a few other reserved commands.
- You can add modifier keys, such as the **Shift** key.
- You can't use shortcuts that your operating system has reserved. If a shortcut is unavailable, SketchUp lets you know.

- You can reassign a keyboard shortcut that already exists in SketchUp. For example, by default, the **O** key is the shortcut for the *Orbit* tool, but you can reassign the **O** key to the *Open* command if you like.

To create your own keyboard shortcuts, follow these steps:

1. Select **Window > Preferences**.
2. In the *Preferences* dialog box that appears, select *Shortcuts* in the sidebar on the left.
3. In the *Function* list box, select the command to which you want assign a keyboard shortcut. If your selection already has a keyboard shortcut assigned to it, that shortcut appears in the *Assigned* box.

> Tip: When you type all or part of a command's name in the *Filter* text box, the *Function* list box options are filtered to only those options that include the characters you type. For example, typing "*mater*" filters the list down to three commands related to materials, as shown in the following figure:
>
>
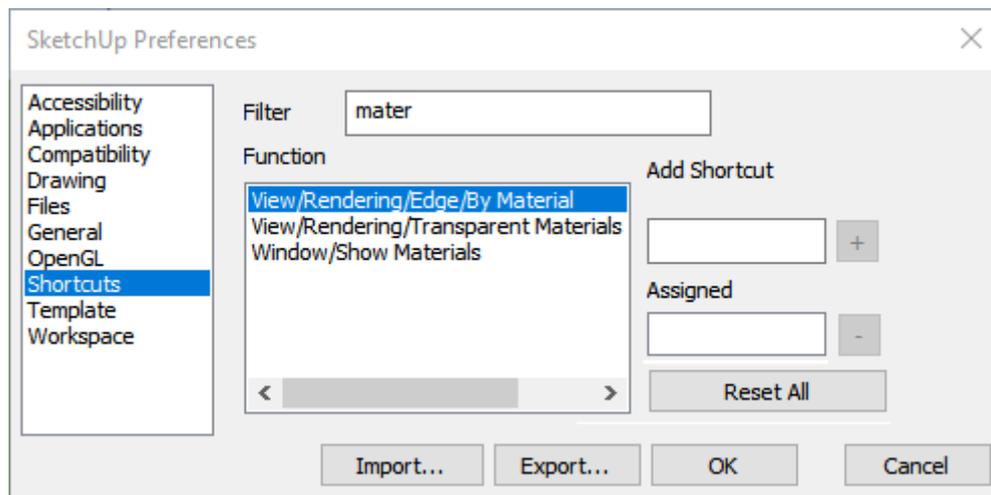
4. In the *Add Shortcut* text box, type the keyboard shortcut that you want to assign to the command and click the **+** button. The shortcut you type moves to the *Assigned* box. If the shortcut you chose is already assigned to another command, SketchUp asks whether you want to reassign the shortcut to the command you selected in step 3.
5. Repeat Steps 3 and 4 until you've created all your desired shortcuts. When you're done, click **OK**.

Tip: If a shortcut is getting in your way, you can remove it. Simply select the command with the offending shortcut in the *Function* list box. Then select its shortcut in the *Assigned* box and click the minus sign button. The shortcut vanishes from the Assigned box — nay, from your copy of SketchUp.

If you ever want to reset all your keyboard shortcuts to the defaults, click the *Reset All* button on the *Shortcuts* preference panel. If you want to load your keyboard shortcuts onto another copy of SketchUp, find out how to export and import preferences in *Customizing Your Workspace*.[9]

Template

List default drawing templates.

Workspace

- Tool Palette

Use large tool buttons: Change the size of the toolbar buttons. The large buttons are enabled by default because they're easier to see and use. For smaller buttons and a larger drawing area, deselect the *Use large tool buttons* checkbox.

- Workspace

Cascade Main Windows: Select whether windows cascade and the default window size (Apple macOS only). On a Mac, windows cascade by default. Deselect *Cascade Main Windows* to turn off cascading. Click the *Save Current Window Size* button to make the size of the active window your default window size in SketchUp.

Reset Workspace: See the toolbars and panel trays after closing them. Click the *Reset Workspace* button and your toolbars and panels instantly reappear in the most recent docked position.

[EOF] – v1.1-0 – October 23rd, 2019.

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[9] https://help.sketchup.com/en/sketchup/customizing-your-workspace