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# SKETCHUP FAQ & ISSUES SHEET

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## I Recommend a Mouse with a Mouse Wheel

I use the mouse wheel. If you don't have a mouse with a wheel I highly recommend getting one. It is not necessary, but extremely convenient.

The thumb wheel zooms in and out as you turn it. However, it also helps you to “pan” an image; if you place the mouse on the left side of the work area and zoom out the image will move left; if you place the mouse on the right side of the work area and zoom in the image will move right. Just the reverse if you place the mouse on the left side of the work area. A similar thing happens when you place your mouse at the top or bottom of the work area. And of course you can combine these actions by placing your mouse in the corners. Try it and experiment with the moves.

One other thumb wheel function you should be aware of: press down on the thumb wheel and the Orbit cursor appears. Now you can orbit by moving the mouse. The combination of placing the mouse while zooming with the mouse wheel, and pressing the mouse wheel to orbit is very useful; with practice you will soon be able to use just the thumb wheel and mouse to move completely around the model.

## What Is Reverse Faces Used For? And What Is the Difference Between a Front and Back Face?

Reverse Faces is a context menu tool used to reverse the face colors of a selected face or faces. Front and back face colors are specified by going to menu Window/Styles, click on the little house icon to choose In Model styles, select the style of interest and then the Edit tab. Lastly select the Face Settings icon and there you can set the colors for Front and Back faces.

What is Front and Back faces. If a six sided cube is modeled correctly, all outside faces have the color specified in the Styles dialog box for Front color; all inside faces have the color specified for Back color. If an outside face has the wrong color you can use the Reverse Faces tools to correct it.

You might ask is it important to correct this situation and the answer is that depends. Internal to SketchUp the face color sets the direction of the face's normal either positive or negative. Many plugins use this to determine how to operate on a face, in which case the wrong face color will cause the plugin to operate on the face incorrectly. So I suggest you always correct the situation unless you know that the tools or plugins you intend to use do not count on face color.

## Can I Use/Copy Components from Other SketchUp Models?

Absolutely; I'll call one the source file and the other the destination file. Have both files open. In the source file select the component you want to copy and then go to menu Edit/Copy. In the destination file go to menu Edit/Paste or Paste In Place.

Edit/Paste will let you choose where you want to paste it. Edit/Paste In Place will put it at the same point (RGB point) in the destination file that it existed in the source file.

Be aware that when you copy a component from another model into a new model its layer(s) come with it.

It's that simple.

## How Do I Access the Warehouse to Download SketchUp Models, Share Models and Components and Get Ruby Script Plugins?

You can get thousands of models/components from the SketchUp Warehouse. On a Windows platform go to menu View/Toolbars, select the Toolbars tab and check Warehouse at the bottom of the list. On a Mac OS X platform go to View/Tool Palettes/Warehouse. This will open the Warehouse toolbar.

- **To Get a Model:** Click on the Get Models icon and it will take you to the library. You can do a search for what you want. You will need a Google account to access it.
- **To Share a Model:** If you wish to put your model in the SketchUp Warehouse, click on the Share Model icon.
- **To Share a Component:** If you wish to share one of your components by placing it in the Warehouse click on the Share Component icon.
- **To Get Ruby Script Plugins:** If you wish to access the Warehouse to download a Ruby Script Plugin click on the Extension Warehouse icon.

## Is There a SketchUp Quick Reference Card for Microsoft Windows & Mac OS X?

Yes, you can get a SketchUp Quick Reference card by going to the following web page:

<http://help.sketchup.com/en/article/116693>

Follow the link shown in the image below:

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1. Selecting a template
2. Exploring the SketchUp interface
3. Learning how to use SketchUp tools
4. Viewing the Quick Reference Card
5. Creating your first 3D model in SketchUp
6. Saving and reopening a model

Then choose either Microsoft Windows or the Mac OS X download link.

On the Quick Reference Card you will find handy keyboard short cut keys and use instructions. For Mac users you might want to download both the Microsoft Windows and Mac OS X versions to get a comparison of key commands. E.g. on the PC a Move/Copy requires the Ctrl key with the Move tool. On the Mac you use the Option key instead. Study this card, it is very useful.

## Which Version of SketchUp Do I need?

All you need is the free version, SketchUp Make 2014. In Lesson 1 I show you where to get it, download it and how to install it. Note, however, when you install SketchUp Make 2014 you will have an 8 hour license for the Pro version with all its capabilities including two additional applications called LayOut and Style Builder. After 8 hours you may be asked if you want to upgrade to the Pro for \$590. If you answer no the license will revert to the free Make version. We will not use any of the Pro capabilities in this class, so just ignore them.

## How Many Segments Are There in a Circle, Arc or Polygon? How Can I Change The Number?

If you select the Circle tool and look in the VCB (Value Control Box) you will see that it says 24 which is the default welded segments in a circle. Similarly if you select the Arc Tool you will see 12 which is the default number of welded segments in an arc. If you select the Polygon Tool you will see 6; this time it is the default number of un-welded segments in a polygon. All of these can be changed to any number of segments you wish by simply typing something like 60s in the VCB provided you don't select another tool before doing so. 60s in this case means 60 segments. 83s would mean 83 segments and so on. You will learn more about this in Part 3.

One to be aware of is that the larger you make the number of segments the more surfaces that are created which means more computations and slower performance. So it is a tradeoff between model visual quality and performance.

## **Is There Documentation of Each SketchUp Command?**

You will find the User's Guide at the following link.

<https://help.sketchup.com/en/article/116174>

Be careful. I don't know how recent this document is i.e. whether it is up to date for version 2014. Early in my own learning of SketchUp I found this document often wrong or misleading. But it may be improved since then. For example, one of the headings is "Modification tools". These tools are now called "Edit".

## **Is There a Way To See a Component's Dimensions (Length, Height, Width) After It Has Been Created?**

If you installed my Construction Plus Ruby script you have a tool called Get Component Dimensions. It's a hand with a dimension line and a 3 on the dimension line. Select this tool and then click on any component. Look In the VCB for its Component Name and Part Name and in the Status Bar for its dimensions.

## **Is There a Way to Draw an Object On a Plane Other Than the Green/Red Plane When the Drawing Work Space Is Empty?**

You will be happy to know the answer is yes and it is quite simple. If you want to draw on the Red/Blue plane select the Front or Back view. If you want to draw on the Blue/Green plane select either the Left or Right side view. If you want to draw on the Red/Green plane select either the Top or Bottom view (the Bottom view can only be reached by going to menu Camera, selecting Standard Views and then Bottom). Iso view variants of these will almost always draw on the Red/Green plane. However, experiment with the Orbit tool and you will find that as a view approaches one of the primary orthogonal views you will draw on the appropriate plane.

## **What Is the Difference between Components and Groups?**

I have to begin by saying I never use groups in my models except as temporary tools. Further, I never use a hierarchy of components. I never use groups because components are a much better object (object oriented programming object that is) for many reasons that are technical and require knowledge of the Ruby language and the implementation of SketchUp. If you have a programming background and would like some further details write me at [jpz@srww.com](mailto:jpz@srww.com) and I will discuss it with you.

I can give you a more practical answer. Many plugins written for SketchUp don't recognize groups (a lot of script writers with knowledge of group object shortcomings choose not to deal with them). One such script is my own CutList Bridge. It completely ignores groups. So if you group primitives instead of making them a component and use my plugin to create a cut list that group would be missing from the cut list.

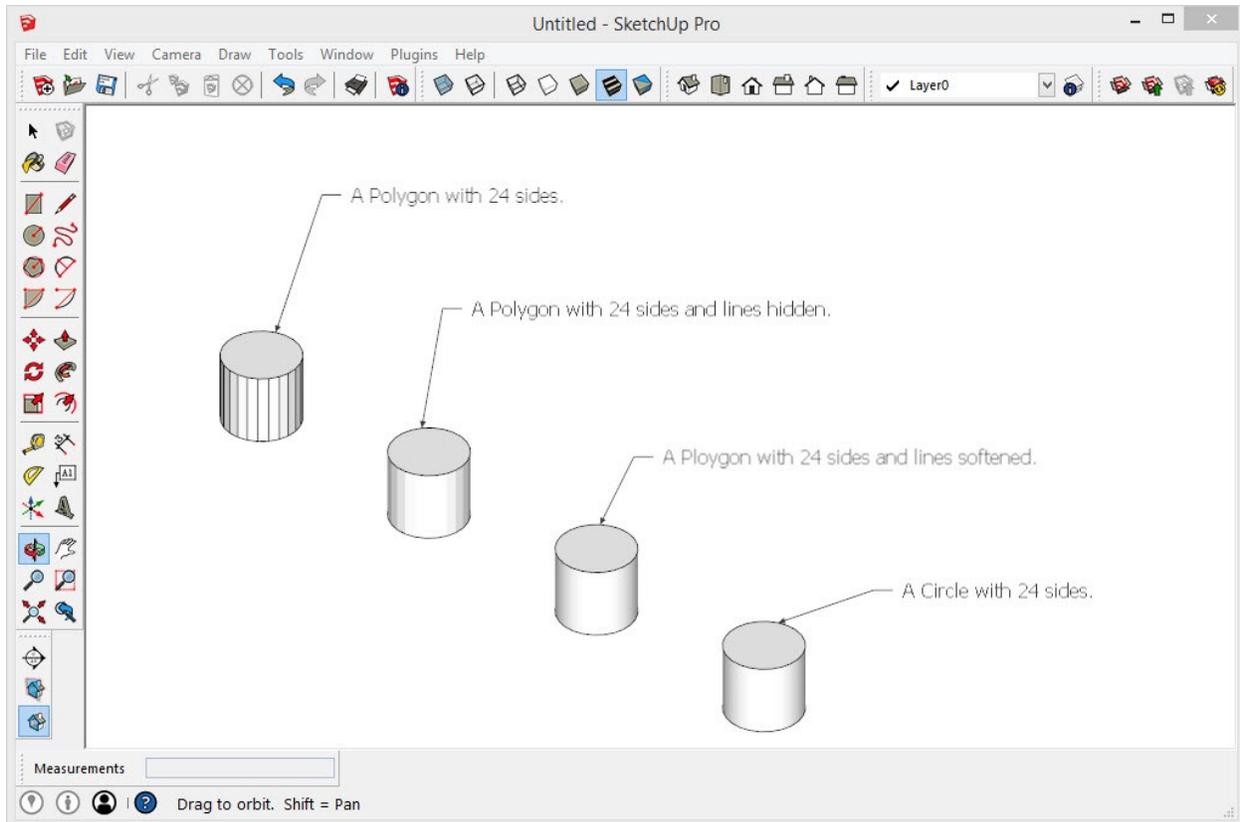
If however, you took all the individual drawer components of a drawer and put them together as a hierarchy of components called Drawer, my CutList Bridge plugin would report a cut list with one Drawer included, but not the components that make it up.

Most experienced woodworker modelers avoid groups except as very temporary devices. I highly recommend you do too.

This may be getting into the technical side of components versus groups too much, but in reality (in the internal coding) a Group is really a Component. However, the coders chose to expose only a few of its attributes. When you make a Group you give it a Name. That name does not have to be unique; you can make 100 different groups with different geometry, shapes and size and give them all the same name and SketchUp doesn't complain. Why? Because the Name field on the Group dialog box is the same field as the Name on the Component dialog box. The Definition field is missing from the Group dialog box because it is supplied internally by the code and is different for each Group you make. So this brings up another reason why I don't use Groups. SketchUp does not keep track of the Group names and let me know that I haven't chosen a unique one. Can you imagine a cut list for a house where all 2 x 4s of different lengths have the same name? So again, I suggest you use Group as only a temporary tool.

## **Hide and Soften – What Is the Difference?**

There is a big difference between hide and soften. Hide simply hides a line so that you don't see it in shaded renderings. Soften does this plus adjusts the shading of the faces on either side of a line so it appears that they come together in a gentle curve. This can be seen in the jpg image of four extruded cylinders below.



The first three cylinders on the left were created with the Polygon tool by setting the number of sides to 24. The last cylinder was created with the Circle tool whose default number of sides is 24.

Notice that when you extrude a polygon that you see the lines separating the faces. In the second cylinder I simply hid those lines using Hide (Eraser with Shift key). In the third copy I softened those lines using Soften (Eraser with Ctrl key). Notice the Third cylinder and the fourth look very similar. That is because a circle, formed with the Circle tool, is made of 24 line segments which are "welded" together so that their vertices form softened lines when extruded. That is effectively what we did in the third cylinder except we softened the lines after extrusion.

**“Windows Media Player cannot play the file. The Player might not support the file type or might not support the codec that was used to compress the file.”**

It sounds like you do not have the appropriate codec installed to play .mp4 files. I would suggest downloading a very good and free player called VLC. You can get it from <http://www.videolan.org/vlc/index.html> . When installing it carefully read all options to avoid installing a program you don't want. Free software often pays for itself from revenues paid to

the distributor for installing other software. VLC, however, has a very good reputation, just be sure to read carefully.

## **Solid Modeler or Surface Modeler**

The difference between solid and surface modeler is small as far as woodworkers are concerned, but large as far as mechanical designers are concerned.

Picture a six-sided box that is empty on the inside versus a six-sided solid cube. If you model the cube in SketchUp and cut it in half you will end up with two five-sided empty boxes, each with infinitesimally thin walls. If you model that same solid cube in a solids modeler and cut it in half you will end up with two solid six-sided cubes. SketchUp knows nothing about the insides of a model; it only has knowledge of its surfaces.

There are plug-ins that attempt to hide this fact such that when you model a six-sided solid cube and cut it in half you get two six-sided empty boxes. But they are still empty, not a solid. So if you are interested in such things as mass and center of gravity, that doesn't help.

This is a very basic answer to the differences. But I think you get the picture.

## **What is Two-Point Perspective?**

In real life, as the eye sees things, objects diminish in size in all three dimensions: x,y and z or R, G and B. You can simulate this and compute it much quicker if you use only two vanishing points: one in the RB plane and one in the GB plane. These are the two vanishing points most used by the eye when viewing things. After all, we don't usually view things from above or below. So Two Point Perspective is computationally more efficient (faster and easier). Note, however, if you go to the Camera menu while viewing the completed Bedside Table from the Top view, Two-Point Perspective is not available to you. That's because that vanishing point is not being computed. Hope this helps.

## **Bezier Tool vs. BezierSpline**

There may be some confusion around the Bezier tools because there are two of them. One is called the Bezier Tool written by the SketchUp Team and the other is called BezierSpline by Fredo6. Both need to be enabled by going to menu Window/Preferences and choosing Extensions. The Bezier Tool has no toolbar associated with it. You simply use the command Draw/Bezier Curves. BezierSpline has a rich toolbar with lots of icons. You will need to install both of these plugins for this course.

## Oversized Bounding Box

Sometimes, when you select a component or group, you may see a bounding box that appears significantly oversized. It is usually due to unintended geometry attached to, or included in a component or group. And sometimes you cannot even see the unintentional geometry because it is hidden. To locate and remove this geometry do the following:

1. Triple click (three fast clicks) on the component or group. This will place it in Edit Component (Edit Group) mode and select all primitives attached to the line or face you clicked on. It will not necessarily select all included geometry if that geometry is not touching the line or face you triple clicked on.
2. Go to menu View/Hidden Geometry. This will expose all geometry in the component or group.
3. Examine the component (group) closely to see if there is geometry that is not intended. Sometimes the RGB axes can hide geometry if it is coincident with it. So inspect closely.
4. Delete any unintended geometry and exit Edit Component (Edit Group) mode.
5. Re-select the component or group and the bounding box should be corrected.

## Beware of Context Menu Explode!

When writing answers to question in the student exercises this morning I was reminded of something I should emphasize. Beware of the Explode function's behavior; it always moves a component's or group's primitives to the layer the component or group resided on before the Explode occurred. For example, consider a component called Leg which is on the Legs layer and its primitives are on Layer0, as they should be. If you Explode the Leg component its primitives are now on the Legs layer. You need to put them back on Layer0 before using them.

## Bonus Videos - How to Create a CutList Using CutList Bridge

Along with the course material you got my CutList Bridge plugin and a User's Guide for it. If you didn't download the User's Guide you can get it by Opening SketchUp, going to menu Window/Extended Entity Info, selecting the Setup tab and clicking on Open User's Guide. This all assumes you installed CutList Bridge correctly.

I also have two videos that show how to create a cut list using CutList Bridge. They are a little old and need to be updated for new releases of CutList Bridge, but you may still find them quite useful none the less. You can find them on the right side of my blog under the heading "CutList Bridge Tutorial". My blog and website address are below. Let me know if you have any questions.

## Layers Manager Layer Tool Functionality

There may be some confusion on how my plugin Layers Manager Layer Tool Add Invisible Layer and Add Visible Layer works. Here is how they work:

Add Invisible Layer always adds a layer to the Layers list but its Visible check box is unchecked in all scenes. If there are no scenes a layer is added and its Visible check box is CHECKED.

Add Visible Layer always adds a layer to the Layers list but its Visible check box is unchecked in all scenes EXCEPT the scene that was active (scene tab is blue) when you added the layer. If there are no scenes a layer is added and its Visible check box is checked.

Hope this helps.

## The Nagging Welcome to SketchUp Screen

Apparently, with SketchUp Make 2014 release Trimble saw fit to nag you each time you open SketchUp to buy a Pro license. They took out the little check box that allowed you to skip this screen. It is still in the Pro version; why not, you already have the Pro license.

If you are a Microsoft Windows user you may want to read this thread:

<http://www.woodtalkonline.com/topic/15368-sketchup-welcome-dialog-nag-screen/>

I am not endorsing it but it is there for your reading and if it solves the problem for you, great. Mac OS X users: I have not found a similar solution. But if I do I will let you know.

## Is Sophie Still In Your Component Library?

If you opened SketchUp with a template other than the woodworking templates you may have Sophie, a female image component, in your custom template file. To delete her from the component library do the following:

1. Open a blank SketchUp file.
2. Immediately go to menu Window and select Components.
3. In the Components dialog box choose the In Model library by clicking on the little house icon.
4. Immediately right click on Sophie. Do not left click on her. Select Delete.
5. In the File menu choose Save As Template.

6. Enter your template name in the Name: field. You can use the same name as your current template.

7. If "Set as default template" is not checked, check it.

8. Click in the Description: field. If you wish enter a description. Now click Save.

## **Drawing an Object on Axes Planes**

A student asked about how to draw a rectangle (or any object) on the Red/Blue plane. Here is the general answer for any axis oriented plane:

If you want to create a rectangle (or other drawing object) in the Red/Green plane use the Top, Bottom or Iso view. If you want to create a rectangle (or other drawing object) in the Red/Blue plane use the Front or Back view. If you want to create a rectangle (or other drawing object) in the Green/Blue plane use the Left or Right view.

If you have SketchUp 2015 you have a new tool called Rotated Rectangle which can be used to draw a rectangle on any plane.

## **Cut Lists, Materials Optimization and Viewers**

A student asked me if you can produce cut lists and then use that list to optimize materials use. He also asked if there was a way for clients to view a textured and rendered model. The answer to all is yes.

You already have my CutList Bridge plugin with a User's Guide in PDF format. That plugin puts a command in the File menu called Export to CutList Plus fx. CutList Plus fx is a very powerful materials optimization tool and does many other things such as produce cost estimates and job pricing. It is not terribly expensive and probably pays for itself in two or three jobs. Go to <http://www.cutlistplus.com/> for more information.

SketchUp provides two viewers for free which clients can use to view your models in 3D (allows orbiting and zooming). One runs on the PC and Mac and the other on IOS and Android phones and tablets. Go to <http://www.sketchup.com> and look under Products.

## **Back Apron Not Correctly Positioned**

A student noticed that I made a mistake in Lesson 3. I did not move the Back Apron in 1/8" like the sides are moved in. It should have been inset 1/8" and because it is not that means the

Mortises in the back legs for the back apron are not in the right place either. The completed Bedside Table.skp model and the PDF are correct. It is only the video that is incorrect. My Bad!

## **How Did I get Concentric Circles in My Drawer Pull? Or, How Did I Get Parallel Lines In My Curve?**

What tool did you use to make the curve of the drawer pull edge? I used the Arc Tool in the video. When you draw an arc with the Arc Tool the arc will be one welded curve, just like the Circle tool. If the arc in your model appears to be connected line segments, not a welded arc, it could be that you used the Arc Tool but then selected it; accidentally right clicked on it and then chose Explode Curve, which would break the curve into connected segments. When you use the Follow Me tool on connected segments you get a number of concentric circles like you did, which are created by the un-welded connections.

A similar problem may occur if you use the Follow Me tool on an arc and end up with lots of parallel lines. Again, your arc has somehow become un-welded.

Suggestion: Before using the Follow Me tool check all curves to be sure they are welded curves; that is curves whose line segments are all connected. You can test this by selecting a curve at any point and you should see the whole curve highlighted. If only one segment is highlighted, you have an un-welded curve.

You can repair an un-welded curve with the BezierSpline tool. Select all its segments. Right click on it and choose BZ – Convert to/Polyline.

## **When Checking A Dimension with the Tape Measure Tool My Dimensions Begin with a Tilde (~)**

This can, and most often does, indicate a problem that must be corrected. So, how do you distinguish between a real problem and a non-problem? If the measurement is of an entity that you intended to specify exactly such as  $2\frac{1}{2}$ " or the spacing of table legs which you specified exactly, then it is a real problem that must be corrected. If, however, the measurement is of an entity such as a line that is the product of other exactly specified dimensions, then it is likely not a problem.

For example, suppose you draw a right triangle with exactly specified sides of 1" and 7" and connect the open endpoints of those sides to form the hypotenuse. When you measure the length of the hypotenuse with the Tape Measure tool it will read something like  $\sim 7\frac{5}{64}$ . The square root of one squared plus seven squared is approximately 7.07106781186548 which is  $\sim 7\frac{5}{64}$ . So the measurement is not a problem in this case.

In the Bedside Table this occurs with the taper lines. See image below. If you look at the Right Front Leg with the Front view selected you see a right triangle with sides of  $\frac{3}{4}$ " and  $16\frac{1}{2}$ ". Using the equation for a right triangle the square root of three quarters squared plus sixteen and one half squared is approximately 16.5170366591589 or  $\sim 16\frac{33}{64}$ . So in the case of the leg if I use the Tape Measure tool to measure the distance up from the floor that the taper starts it should read exactly  $16\frac{1}{2}$ ". But if I measure the length of the taper line it should correctly read  $\sim 16\frac{33}{64}$ ".

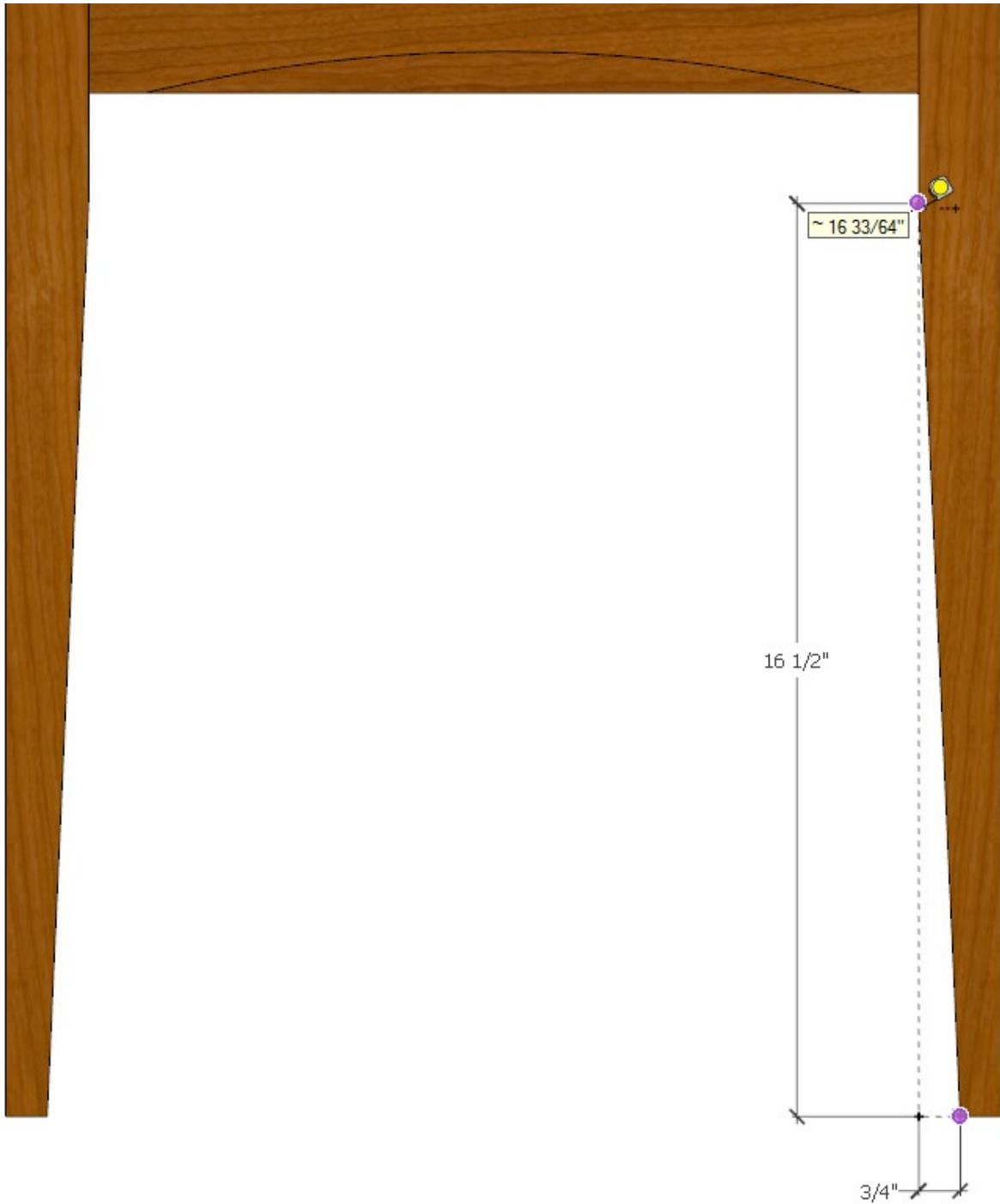
## **My Layers Don't Seem To Be Working Properly. When I Make a Layer Invisible Some Parts of a Component Remains. What's Up?**

What you have likely done is what almost all SketchUp beginners do at one point or another. You made a layer other than Layer0 active either accidentally or intentionally at some point. You should never make a layer other than Layer0 active. The solution is simple. However, before I tell you the solution check to see if you have Layers Manager installed; go to menu View and select Toolbars (Tool Palettes on Mac) and scroll down to see if Layer Tools is checked. If not check it. If it is not on the list you need to install the plug-in.

The solution which I call the Triple Click Test:

1. Go to each component in your model one at a time and triple click on it with the Select tool so that all the primitives are selected and the component is in Edit Component mode.
2. Check the Layers drop down. It should say Layer0. If it is blank, or if it says something besides Layer0, use the Layer drop down box in the Entity Info dialog box to select Layer0. This will place all primitives on Layer0.
3. Repeat the above for each component and the problem should be solved.

When you intentionally or accidentally make a layer other than Layer0 active primitives are placed on that layer. That is why Layer0 should always be the active layer.



Having said all that, whenever you see a  $\sim$  when using the Tape Measure tool you should investigate and convince of its reason for existence. The problem you may be having is a very common problem for people just beginning with SketchUp. But it is one you have to nip in the bud quickly or you will get a long ways along in a model before you realize you may have to

redo the entire model. You will then become frustrated and unhappy with SketchUp. So here is what to look for and what you need to proceed.

I suggest using the Tape Measure tool frequently while learning SketchUp to check your steps, lest you get way down the road and find a problem that can't be easily repaired. If you see a ~ zoom in close and you will see that a line is not connecting to the point it should connect to; there is a gap.

The solution to your problem is to keep a very close eye on the Inference Engine. The Inference Engine is the mechanism that produces those tool tips such as End Point, Midpoint, Midpoint in Component, On Edge, On Line, On Red Axis etc. When you are trying to connect a line or polygon to a point, slow down with your mouse and be sure you click it only when you see the Inference Engine tooltip you desire. This will ensure you are connecting two points exactly. Often, what new students do is locate the correct Inference Engine tooltip but then when clicking do it with just enough force to move the mouse slightly off the desired point. Over time you will develop a motion that is reliable, but early on you have to be careful and conscience of clicking on the tooltip. Also, use the mouse wheel to zoom in close when trying to connect a line or polygon to a point. Don't try to connect things when the entire model is in the window because likely you are not close enough to get an exact attachment.

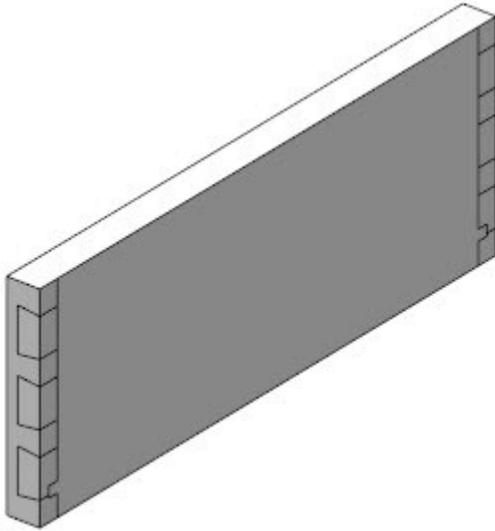
One other thing that may cause the problem you are seeing is a setting in Model Info. Go to Window; choose Model Info and then Units. Make sure that "Enable length snapping" is unchecked.

## **Answer to Lesson 4 Student Exercises Question 7**

I was asked by Johnathan Ericson about the answer to question 7 in the Lesson 4 Student Exercises. He must have caught me in a good mood because I gave him the answer. I'll repeat the question here and then provide you the answer too.

Q: Look at the method I used for forming the drawer front including the pins and rabbet. Redraw the drawer front with at least two substantially different methods, but still using the sides and the Intersect Faces tool to form the pins.

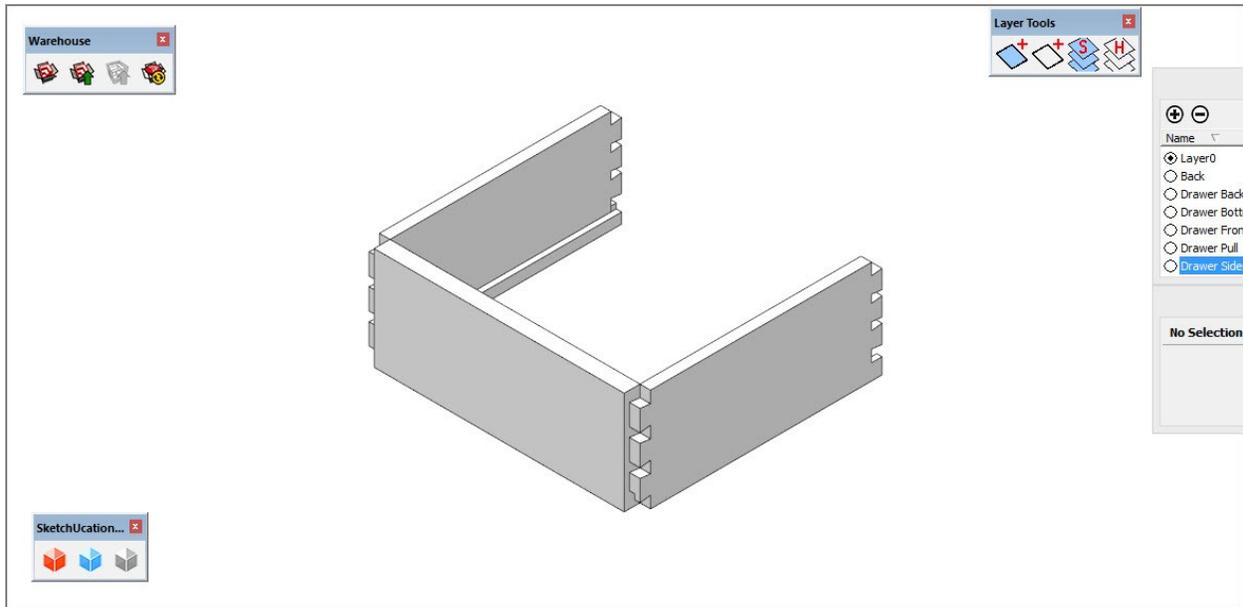
A: Recall in Lesson 4 that I formed the Drawer Front by creating a rectangle and pulled it out to the most forward faces of the tails making it 1/2" thick. Then used the Intersect faces tool. If you look at the dimensions provided for the Drawer Front you would notice that the Drawer Front is actually 3/4" thick. In Lesson 5 I added the other 1/4".



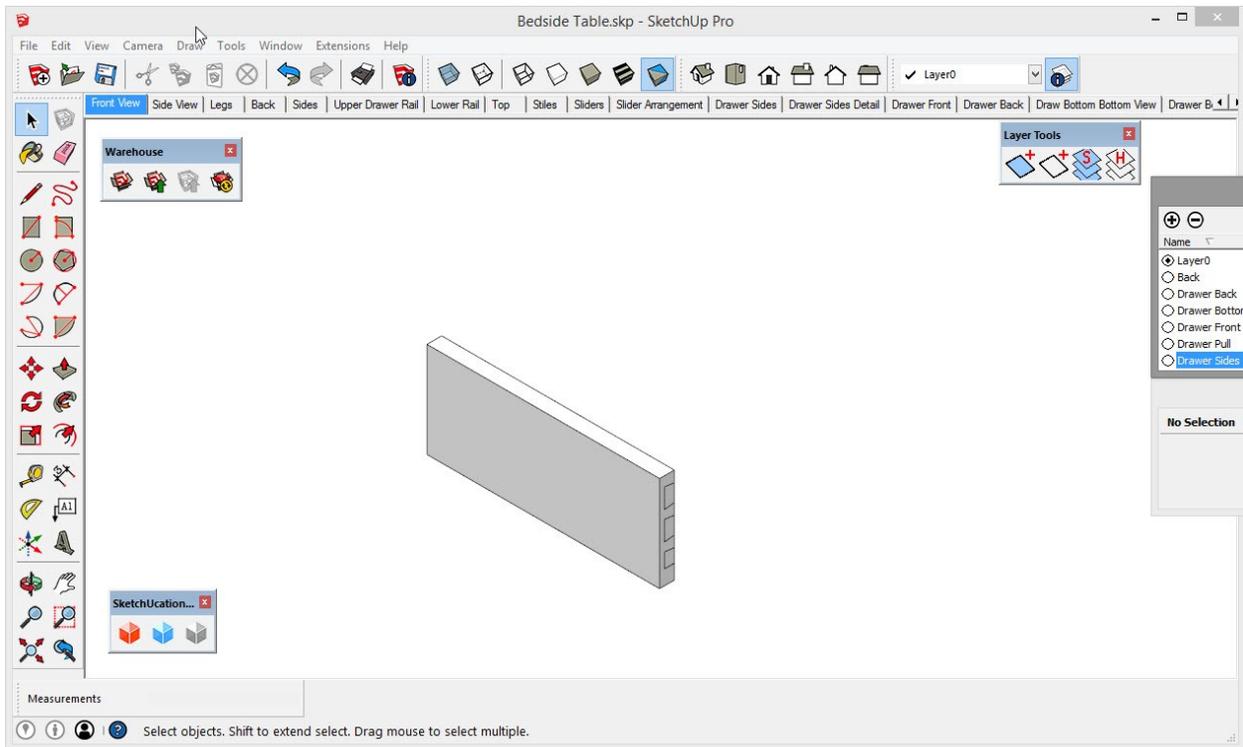
## Slide 1

But suppose in Lesson 4 I created the rectangle and pulled it out 3/4" right away and then used the Intersect Faces tool? I would be creating blind dovetails instead of through dovetails which I want in the end anyway. See Slide 1 above. From this point you should be able to do the cleanup and Push/Pull operations.

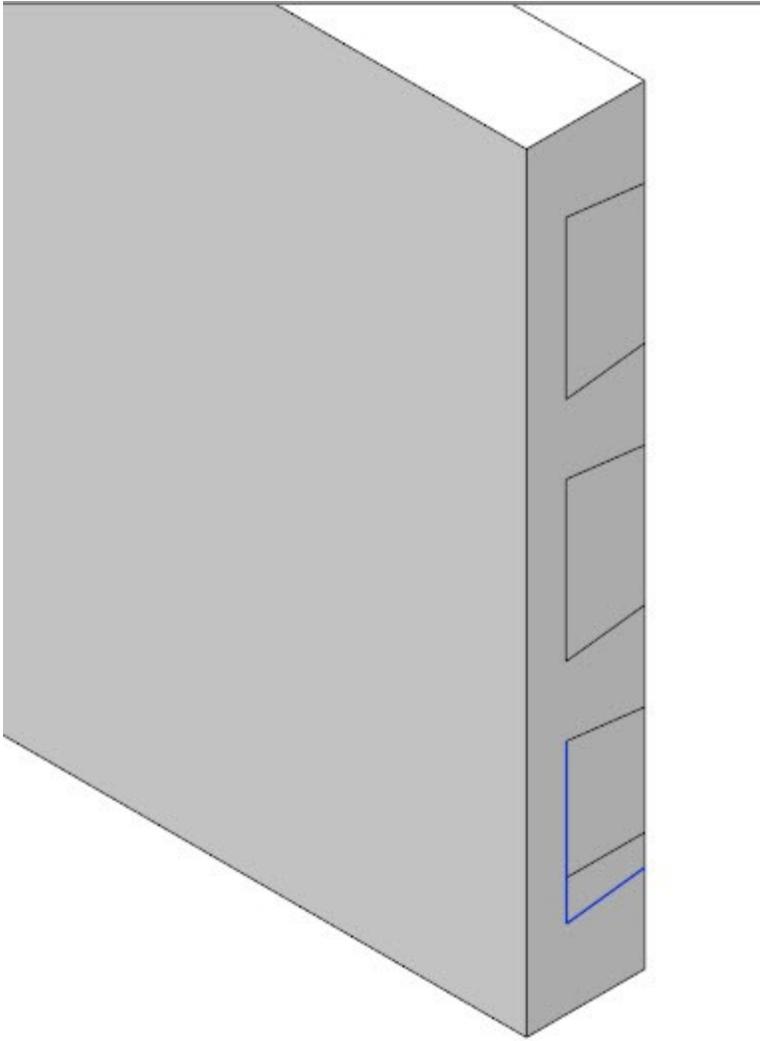
Here is a third way. Look at the remaining pictures below. Of the three methods this one actually requires the least work and gets me the blind pins I want. Instead of drawing the rectangle from outside corner to diagonally opposite outside corner, I draw it inside corner to diagonally opposite inside corner. Then pull it out 3/4" shown in Slide 2. Slide 3 shows the result of the Intersect Faces operation. However, the half pin on the bottom isn't correctly formed. See Slide 2 to understand why. Slide 4 shows how I copied two lines from the middle tail/pin outline and pasted it on the bottom tail/pin outline. I now have one line to erase and one Push/Pull operation to get the result in Slide 5. Pretty cool Eh?



## Slide 2

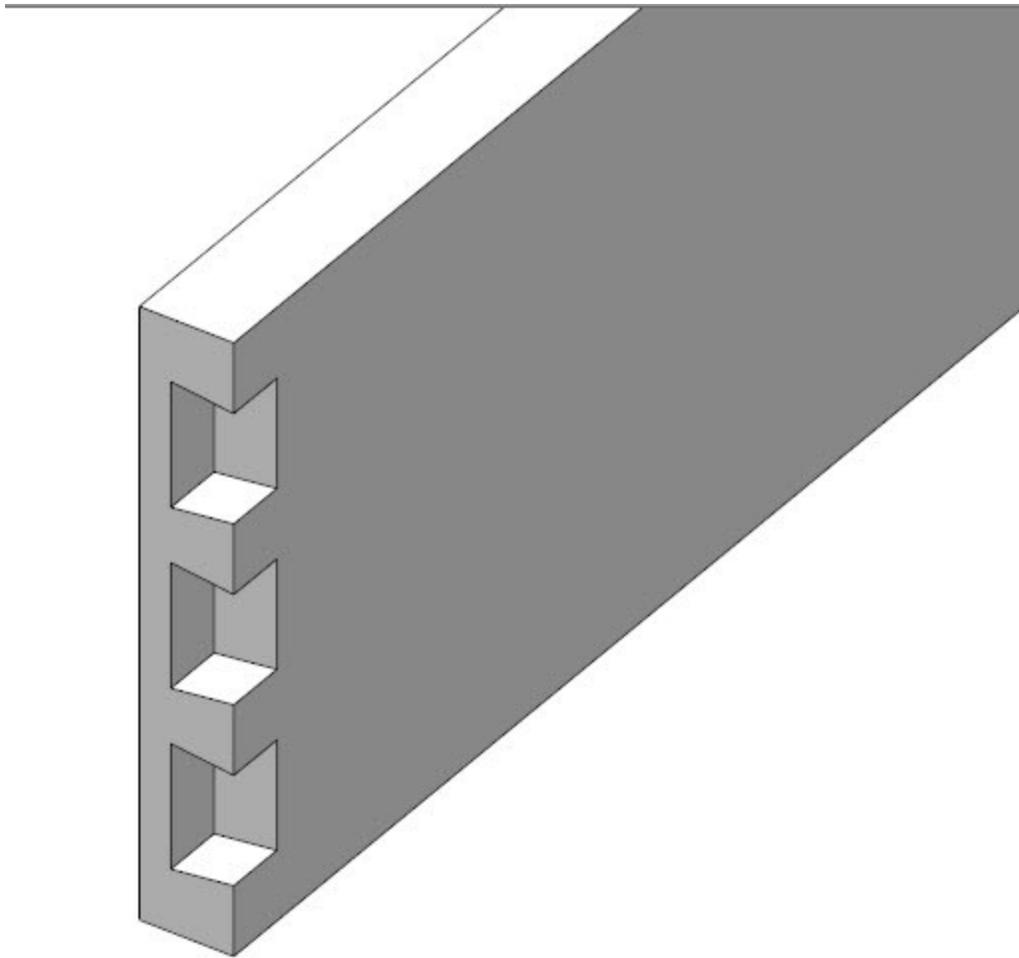


## Slide 3



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**Slide 4**



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## Slide 5

### Beware of Un-welded Arcs

A lot of you are having problems with the Drawer Pull in Lesson 5. After using the Follow Me tool you are ending up with a pull that has a lot of concentric circles. I went back to review the video to see if there was something I did in the video to create this problem and I can't see anything. I can't figure out exactly what you may be doing wrong. However, I can give you a tip as to how to avoid it. First a little background.

Draw an arc using the Arc tool; anywhere in unused space. Then select it with the Select tool. Notice that the entire arc is selected. Next, while it is still selected right click and choose

Explode Curve. Now deselect the arc by clicking in white space and reselect the arc again. Notice this time that you can only select segments of the arc. The arc as initially drawn has segments that are welded together. The Explode Curve breaks those welds.

Those of you who are ending up with concentric circles in your draw pull are victims of un-welded arcs. How they became un-welded I can't tell you without watching your every modeling step. But here is what I suggest. Just before selecting the circle for the Follow Me path select the arc and make sure it is still welded. If not either redraw it using the Arc tool, or select all its segments and use the Weld tool to re-weld it. Then select the circle as the path and then the Follow Me tool. That should solve the problem.

## **A Students Objects Disappear**

What you have done is likely what almost all SketchUp beginners do at one point or another. You made a layer other than Layer0 active. Send me your .skp file and I will check to see if this is the case. But I strongly suspect so. If it is the solution is simple. But before I tell you the solution check to see if you have Layers Manager installed; go to menu View and select Toolbars (Tool Palettes on Mac) and scroll down to see if Layer Tools is checked. If not check it. If it is not on the list you need to install the plug-in.

The solution:

1. Go to each component in your model one at a time and triple click on it with the Select Tool so that all the primitives are selected.
2. Check the Layers drop down. It should say Layer0. If it is blank or if it says something besides Layer0 use the drop down to select Layer0.
3. Repeat above for each component and problem should be solved.

I call this the triple click test. When you intentionally or accidentally make a layer other than Layer0 active primitives are placed on that layer. That is why Layer0 should always be the active layer.