Rhino & Grasshopper Efficiency workshop notes

Rhino

Open Rhino and draw a mixture of objects (curves, points, solids, surfaces).

Selection

Click + drag from left to right: select everything that is fully within the selection box.

Click + drag from right to left: select everything that is fully or partially within the selection box.

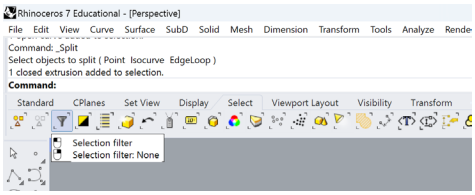
Hold shift and click on different objects to select multiple items.

Deselection

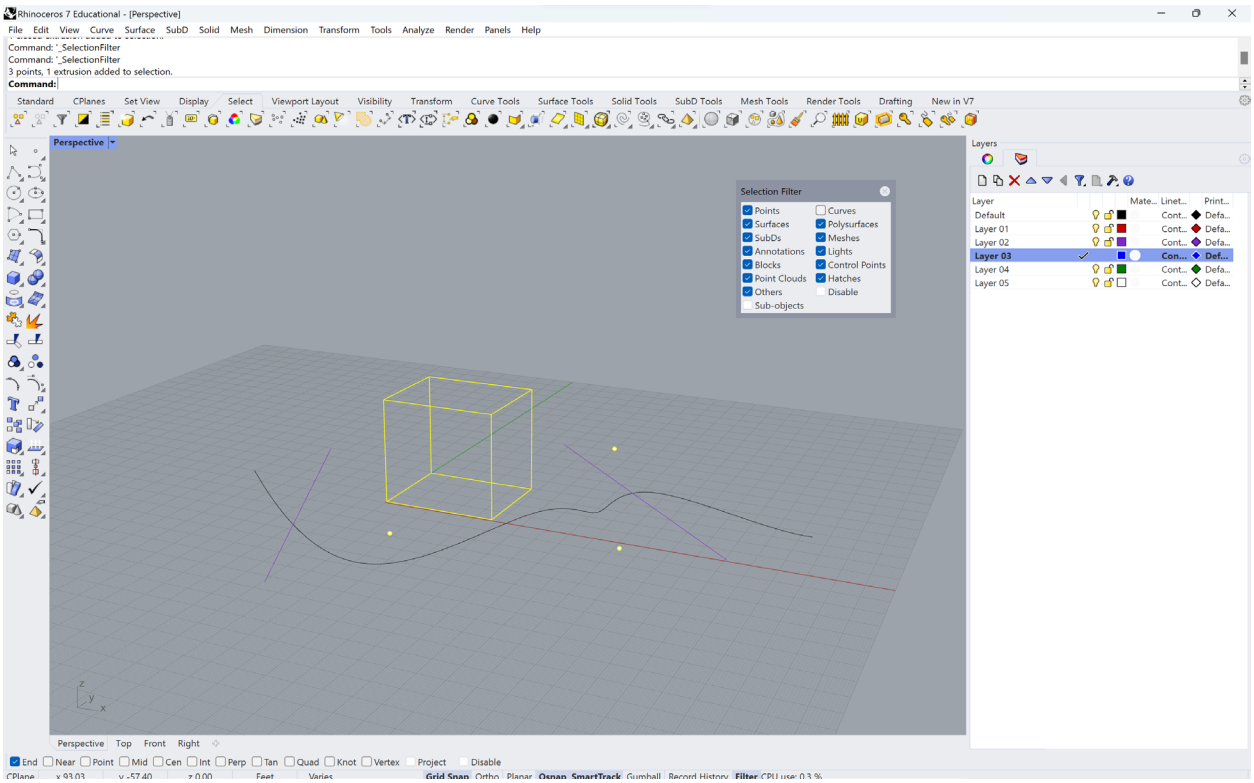
Control + click or Control + click + drag to deselect.

Selection Filter

Select tab → Selection Filter → left click

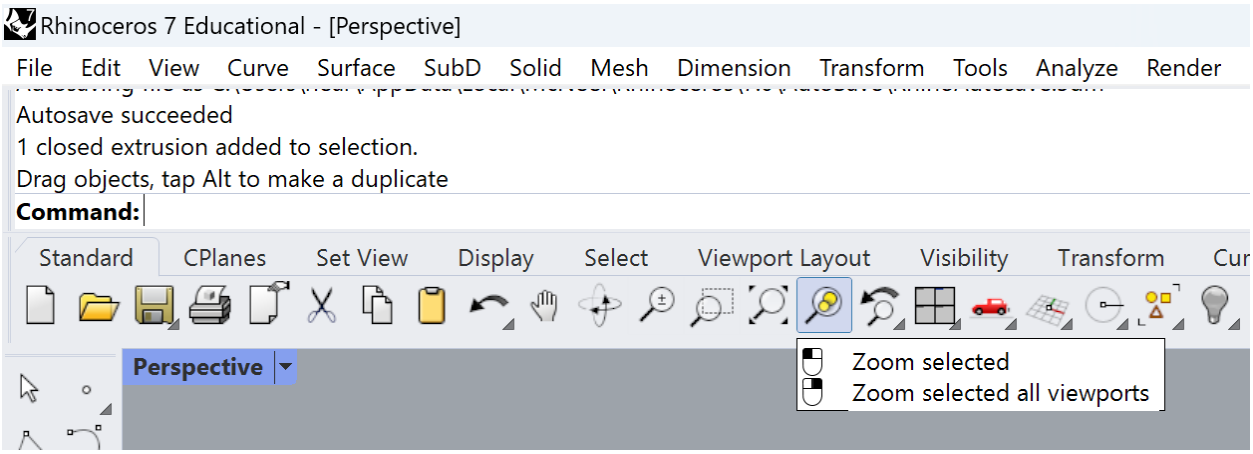


Check and uncheck boxes to select only a certain type of elements. For example, below I cannot select the curves right now. So even if I click + drag to select multiple elements the curves will not be selected. I find this is especially helpful if I've used many points or curves to construct several solids, and I want to move the solids away from the curves and points with ease.



Center object in view

Standard tab → Zoom selected

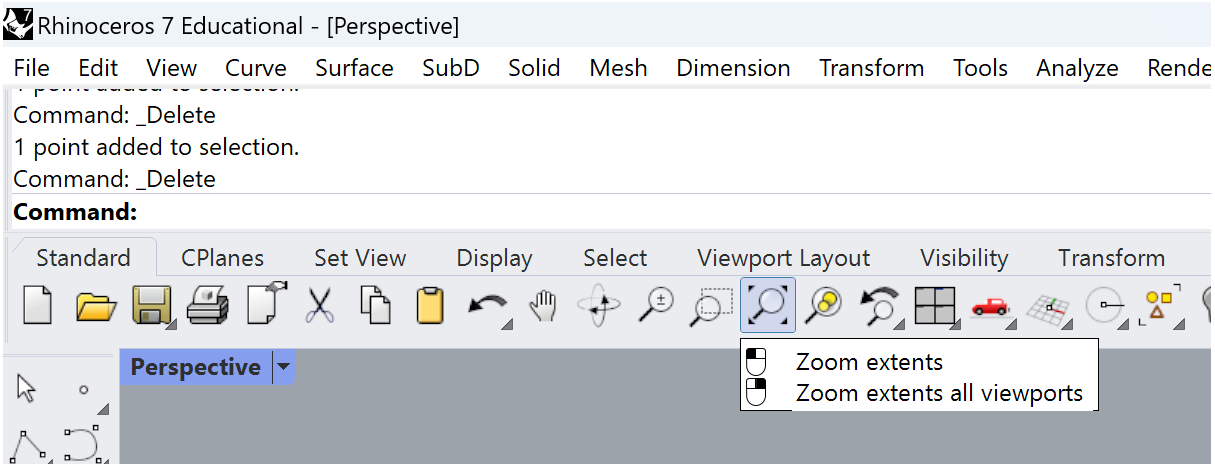


This allows you to rotate more easily around an object that is not near the origin.

Zoomed out too far

If you have zoomed too far out and you do not know where your model is anymore, it is easy to fix.

Standard tab → Zoom extents



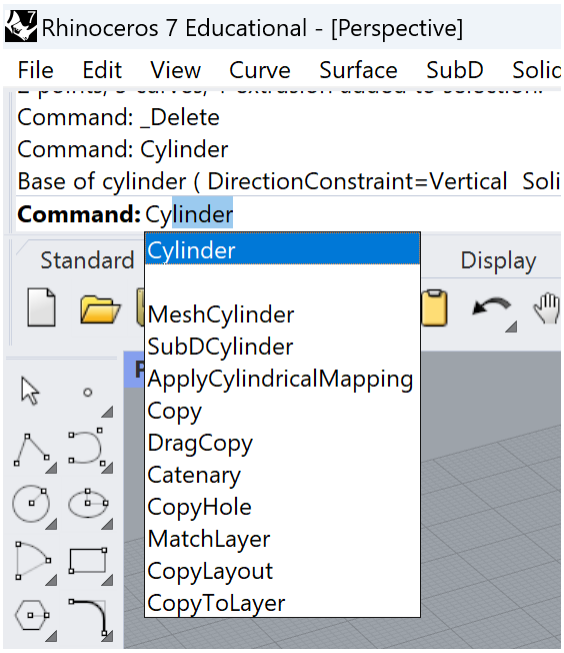
This will reorient the view you're in to include all elements in your model with as little extra space as possible.

One other tip to prevent you from zooming out too far and being unable to find your model: place a point close to the origin. Points do not disappear no matter how far out you zoom.

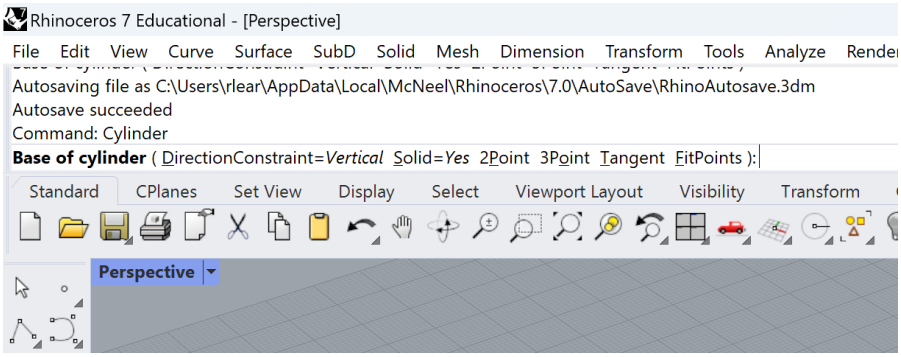
Command Bar Use

The command bar is extremely useful for finding new commands, understanding tools, and working more efficiently. For example, let's make a hollow cylinder without ever touching an icon.

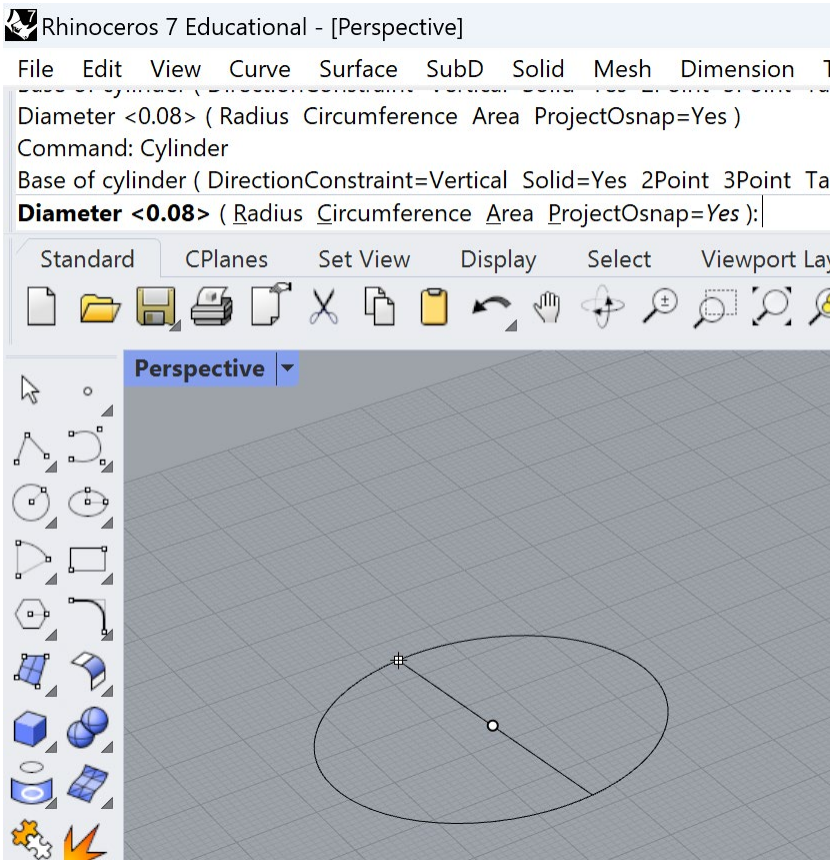
1. Type "cylinder." It should autofill quickly and once it says cylinder in the bar below, you can hit Enter. If you've misspelled the command, it will likely pop up in the list of options or similar commands, which is useful.



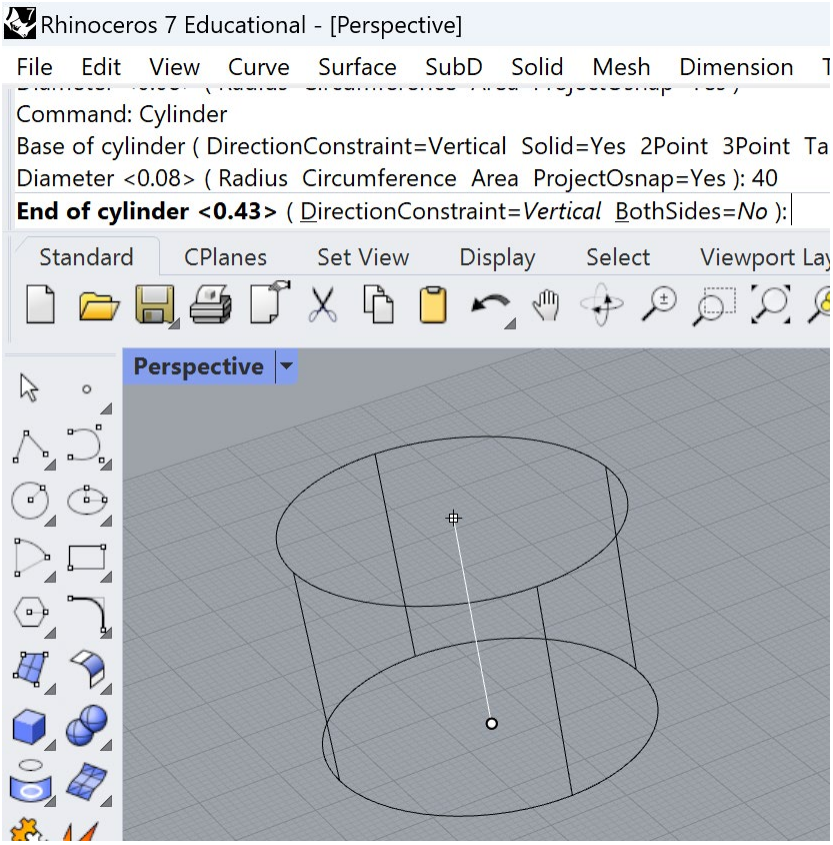
2. Read the command bar to figure out what to do next. In this case, pick a point to be the base of the cylinder.



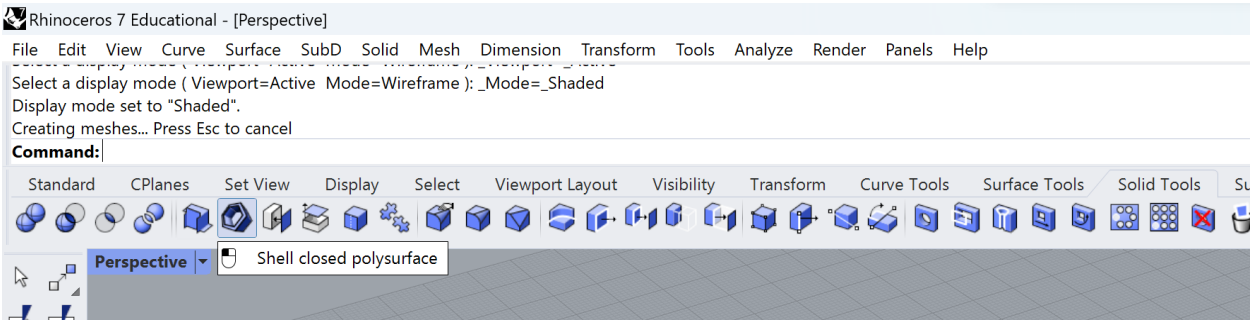
3. Read the command bar again and select a diameter for the base of the cylinder. You can either select a point on the grid or type a number to create a specific diameter. Notice that there are other options in the command bar, for example "Radius." If you click on radius, you can do this step by selecting a radius for the cylinder instead of a diameter.

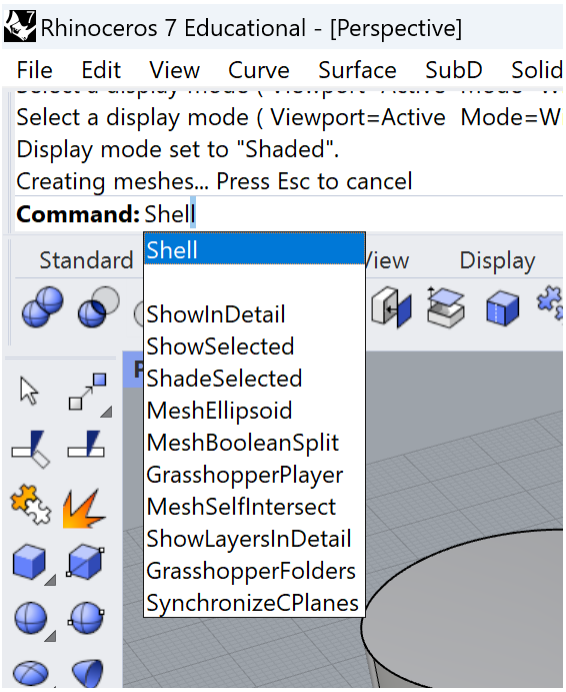


4. Read the command bar again and select a height for the cylinder. Once again, you can either pick a height or type in a specific height.

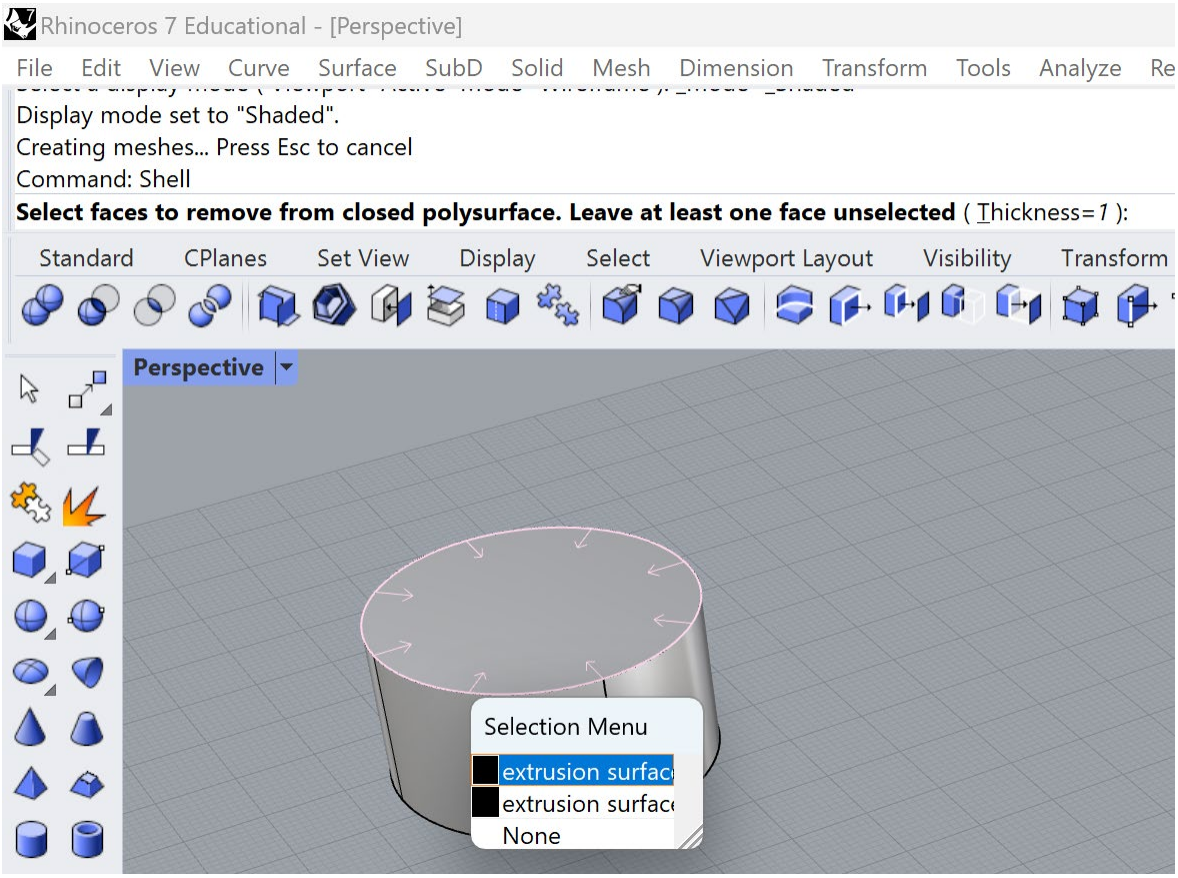


5. Now, let's make the cylinder hollow. You can either go to Solid tab → Shell closed polysurface or begin typing in "Shell," then hit Enter.

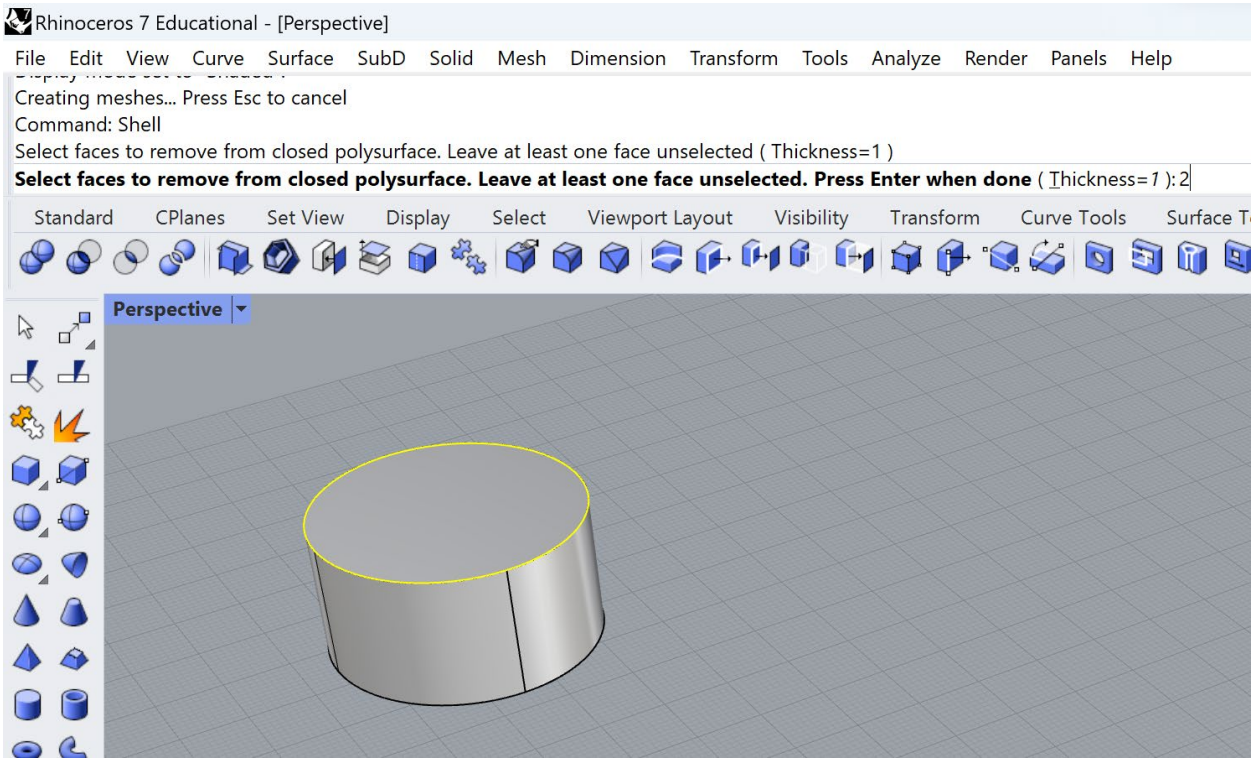




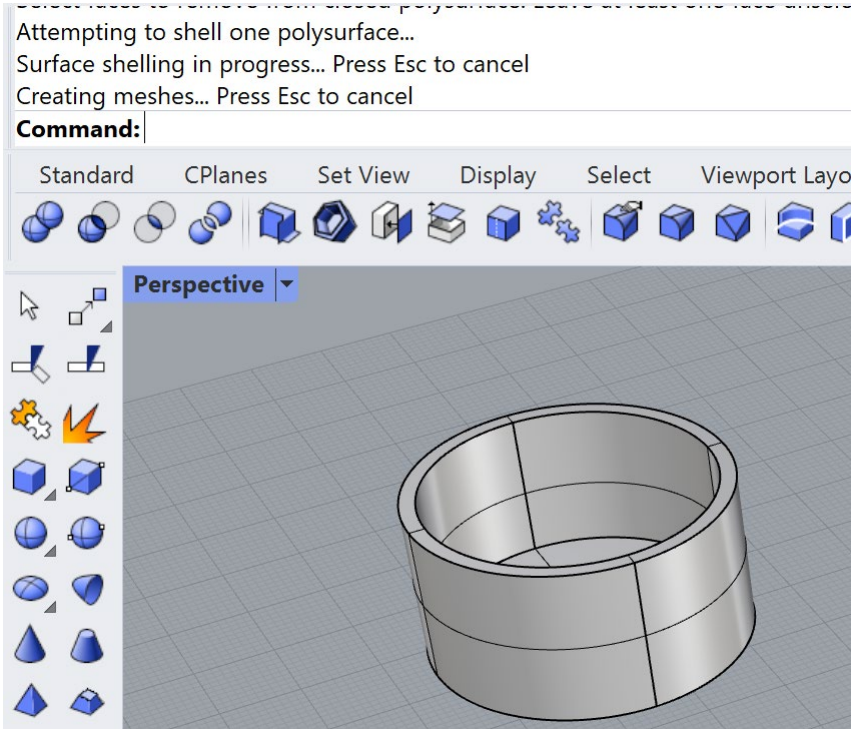
6. Follow the command bar instructions to pick a face to remove. You can hover over different parts of the cylinder to see what faces you can select.



7. Select at least one face. Then, you can type in a number to change the thickness of the remaining walls. I typed in 2.



8. Hit Enter. You now have a hollowed-out cylinder.



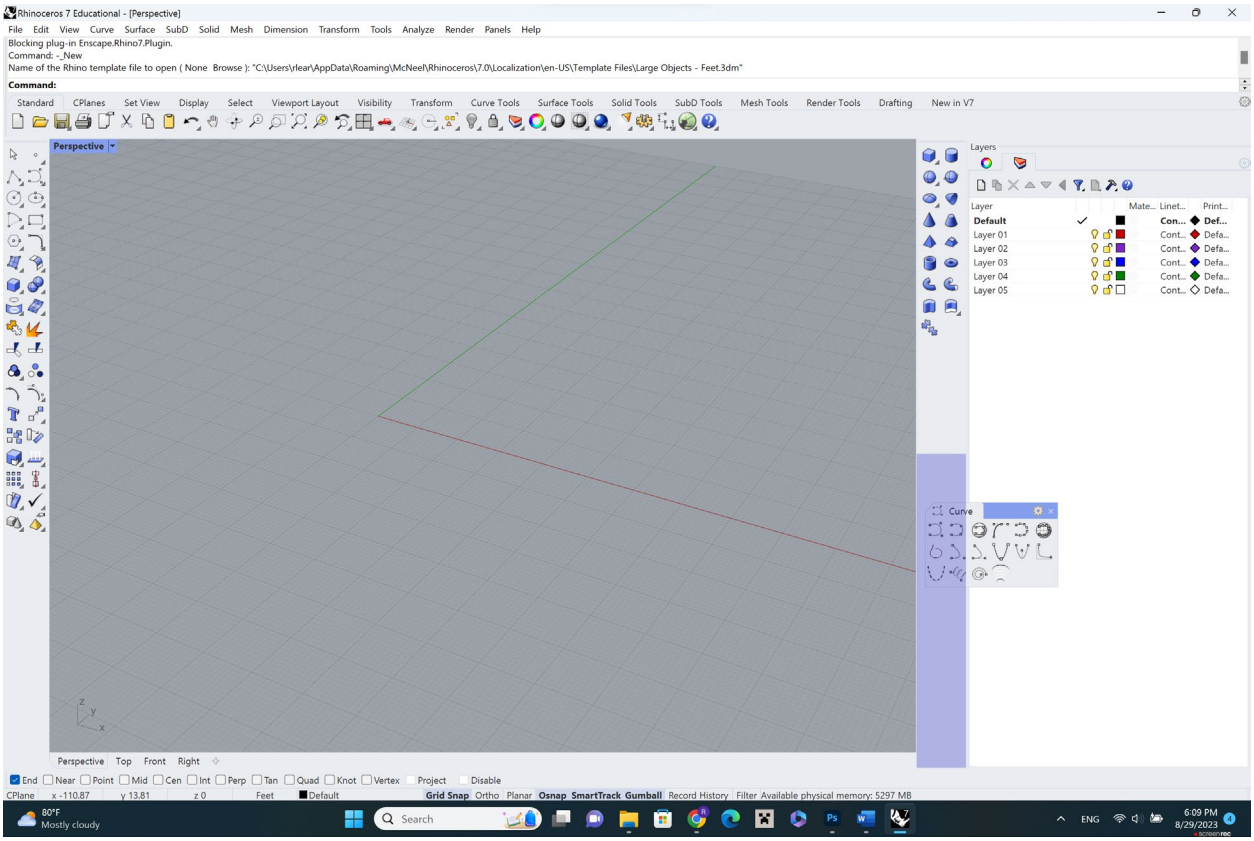
Other command bar notes:

You can scroll up through your command bar history. This can help you troubleshoot a potential missed step.

Check your command bar frequently, especially if you're having trouble selecting something. Likely you have an open command in the command bar. Hit Escape to clear it.

Using tool palettes

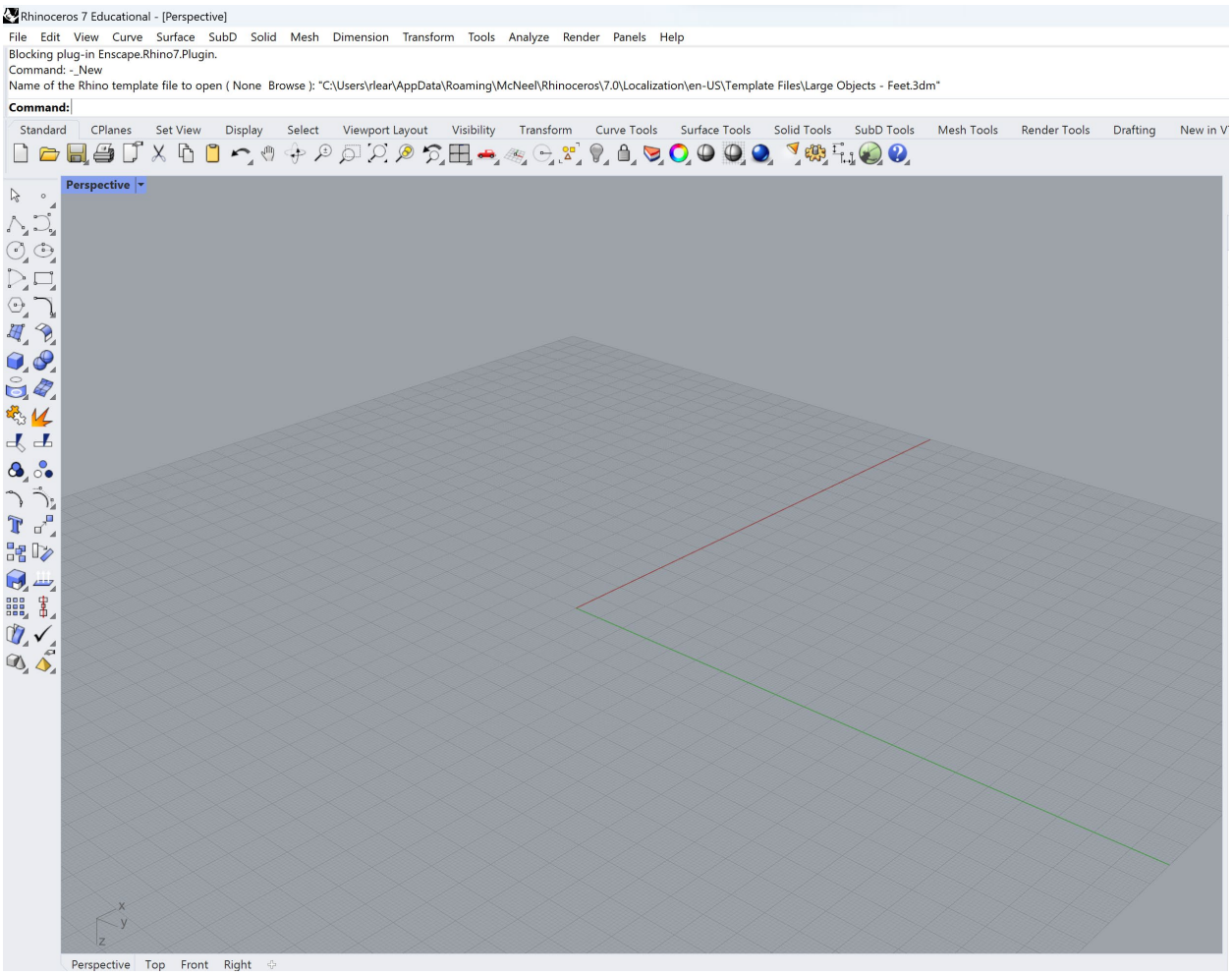
If you like to have tool palettes open (this will be especially useful for rendering tool palettes), but you don't like it when you have lots of pop-up boxes on the screen, you can move a toolbar to the edges of the screen. There are many places on all the edges where a toolbar can be moved.



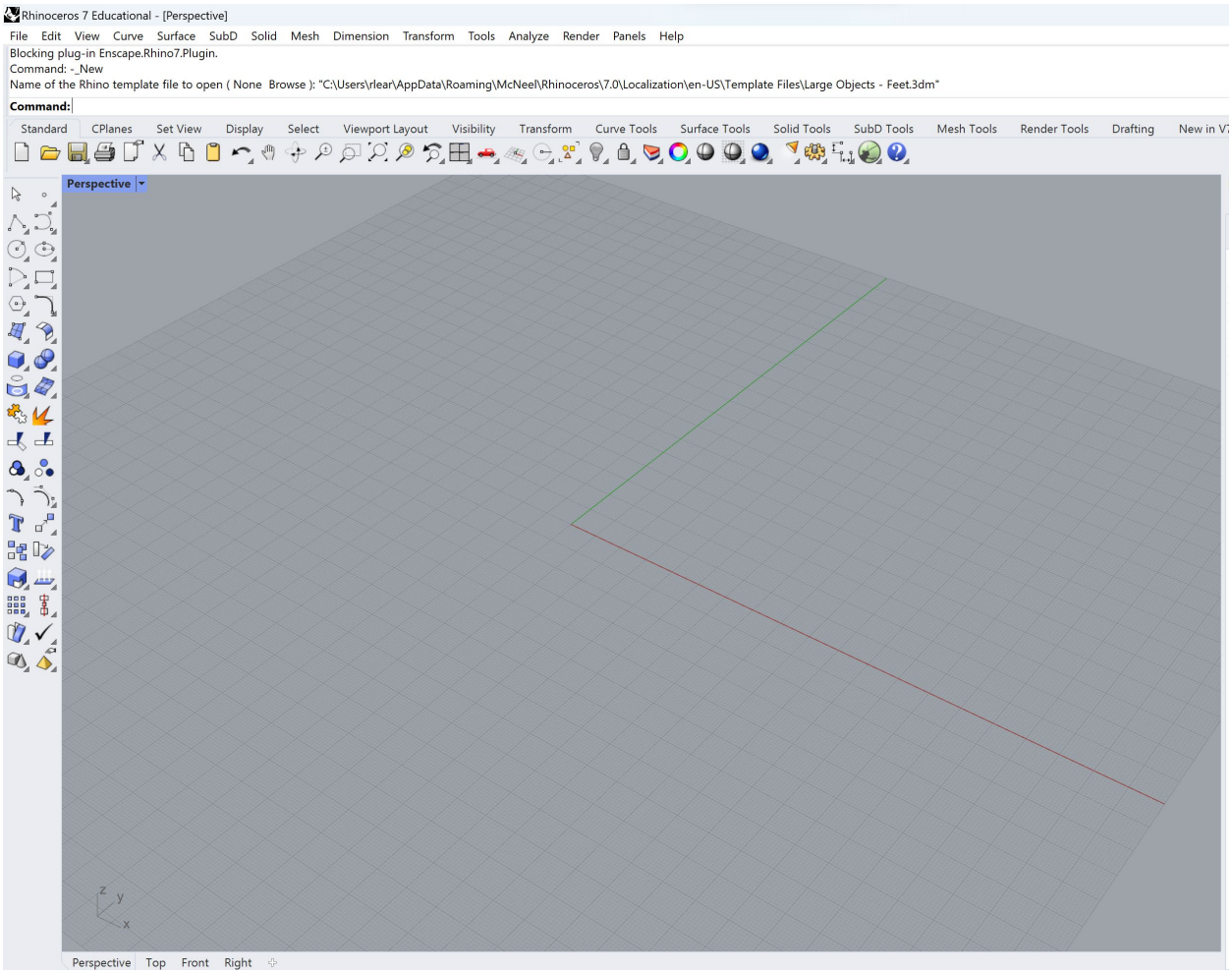
If you don't want the toolbar there anymore, you can hover on the upper left-hand corner of the bar, left click once you see the move symbol, and put it somewhere else or get rid of it.

Rotated to upside-down

If you're trying to rotate around your model and it seems weird, perspectives are off, or a Z-vector in grasshopper doesn't seem to be working right, you're likely looking at your model from a worm's-eye view rather than a bird's eye view. Another way to tell is if the red axis appears to be to the left or counterclockwise from the green axis.



If this happens, carefully rotate the model so that the green axis is to the left of the red axis, and you should be good to go.

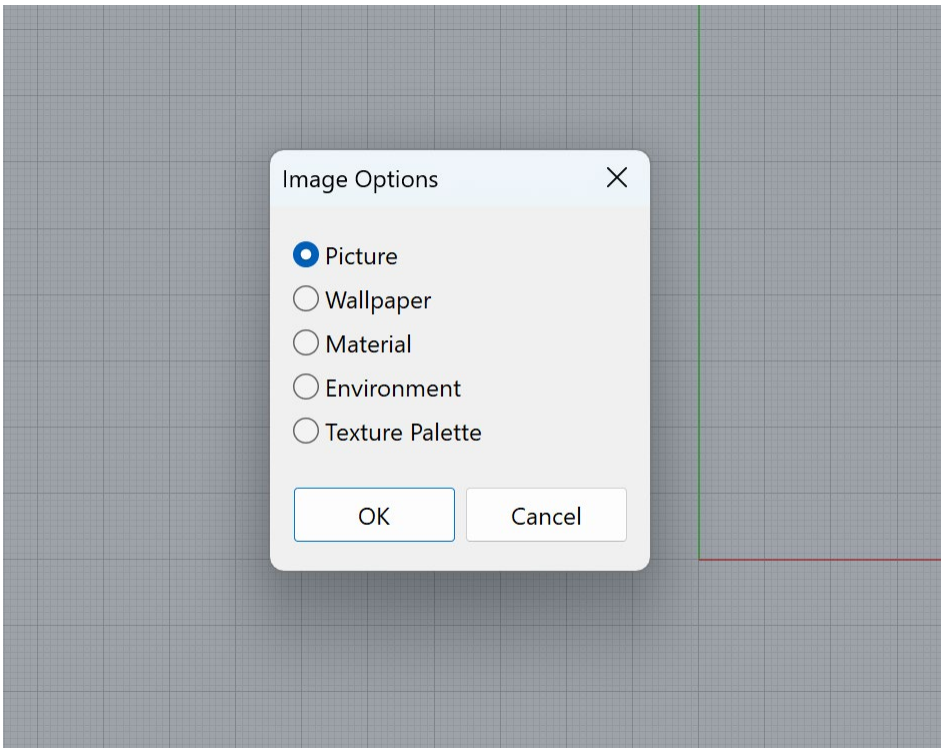


Scaling Drawings

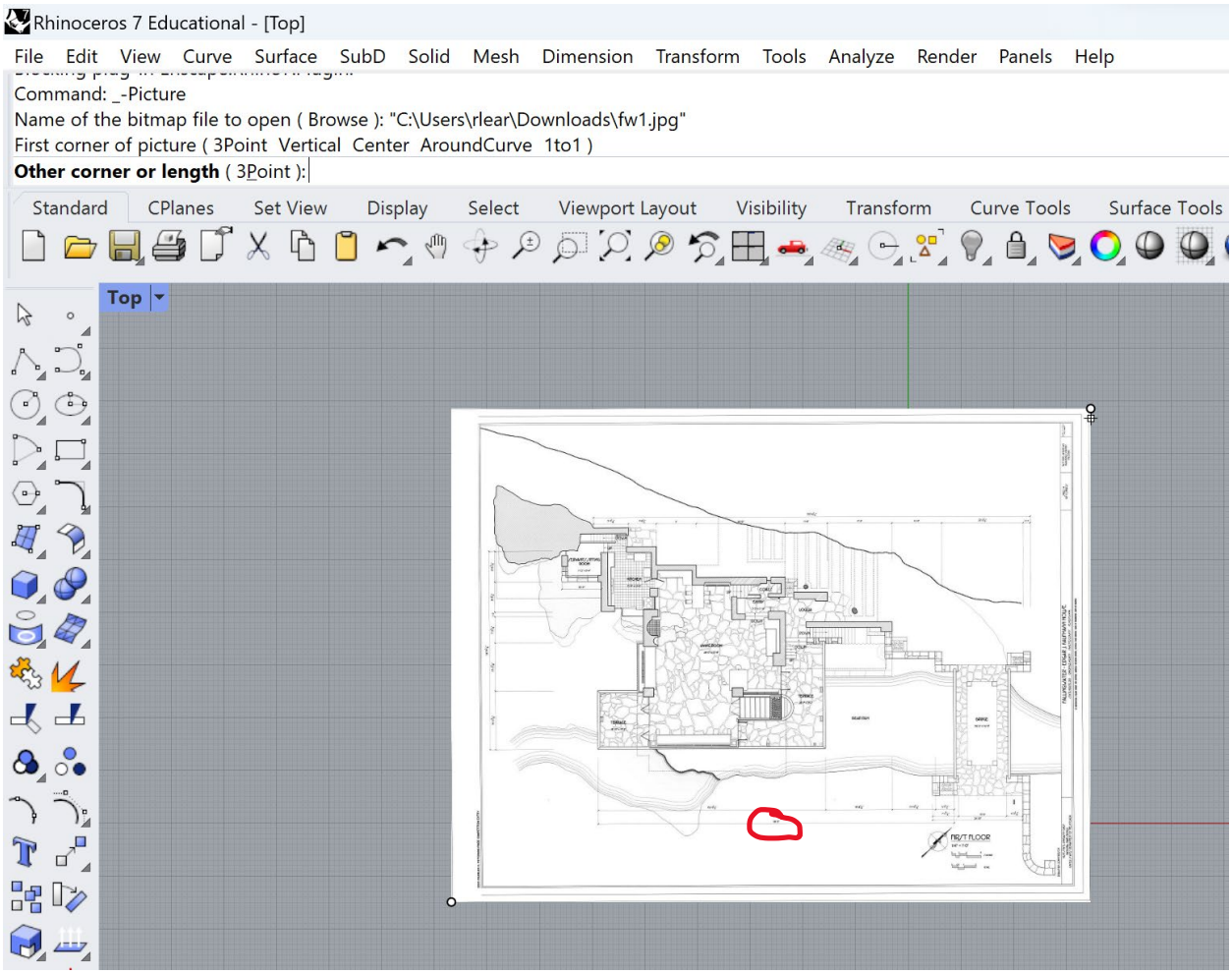
Probably many of you have already done portions of this in order to complete the exercise. However, your scale might still be off, or perhaps your drawings are scaled to one another and not to a real dimension. Here's how to fix that.

Save the drawing you wish to import to Rhino to your computer. Make sure it's a drawing with some sort of scale indicator (either a scale bar or a dimension). For the purposes of this exercise, go to classes → examples → Folder named SIA_Workshop → FallingWaterPlan.

Drag + drop the image from your computer into the rhino "Top" view, at which point an "image options" popup will appear. Hit "OK."

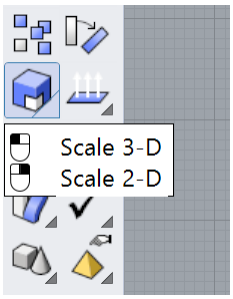


It will ask you to place the two corners of the image. Just pick a point in the bottom left and another in the top right, the exact size does not matter right now.

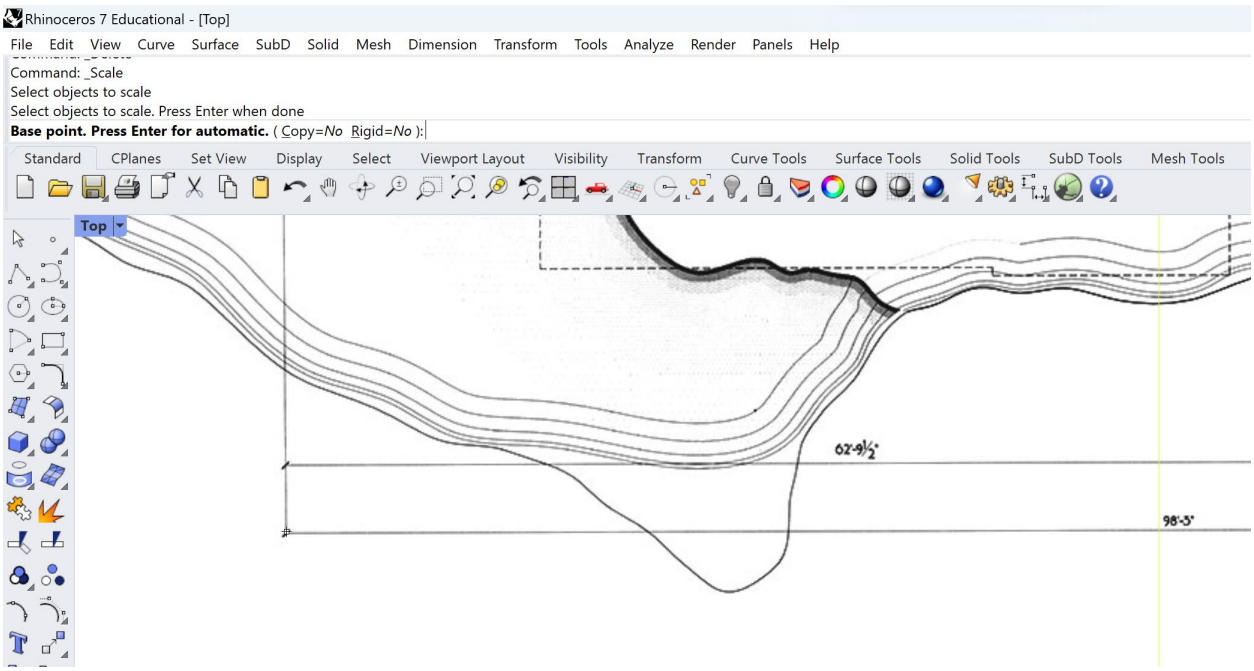


Look at your image to find any sort of scale indicator. It is usually helpful to pick a longer dimension if possible. For mine, I am going to use the dimension at the bottom that says 98’3,” which is circled in red above.

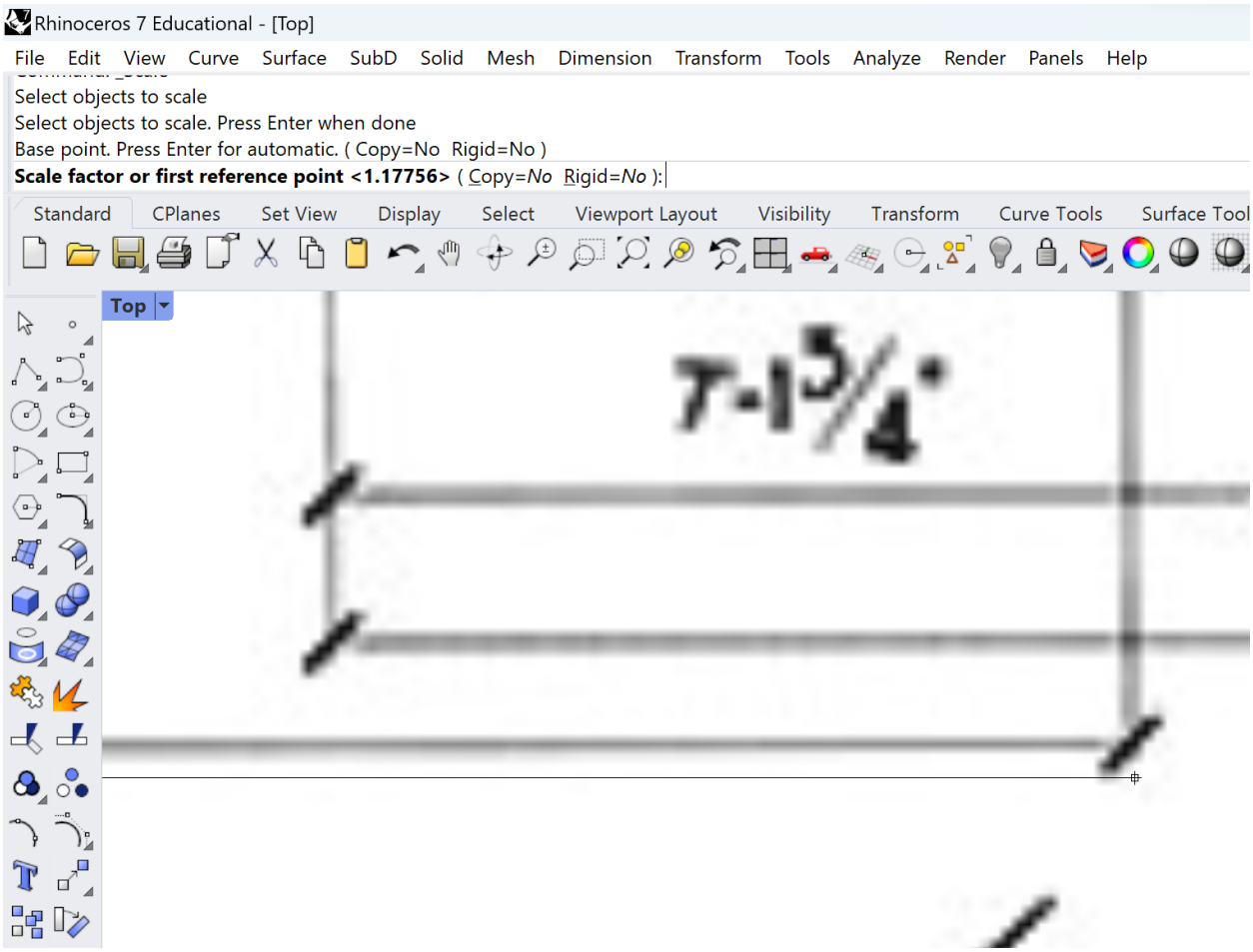
Click on the Scale tool.



Select the drawing. Press Enter. If you have already drawn your plans, you can select those as well before hitting Enter. Zoom in on the drawing and click on one end of the chosen dimension line as precisely as possible.

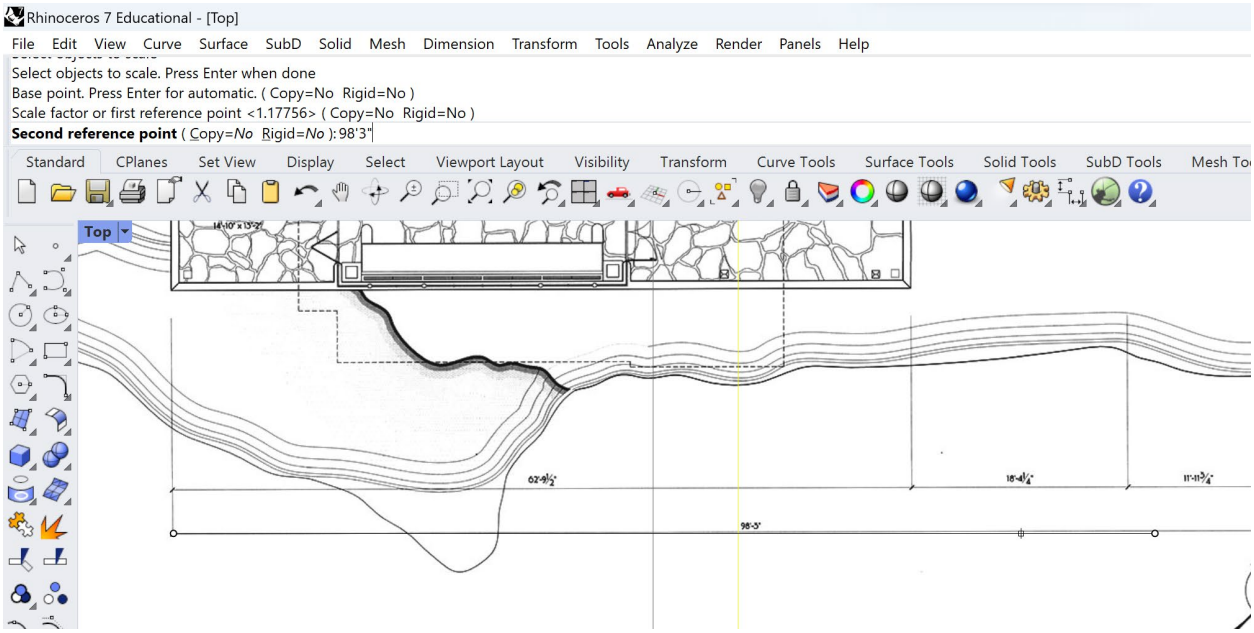


Now click on the other end of the chosen dimension line as precisely as possible.

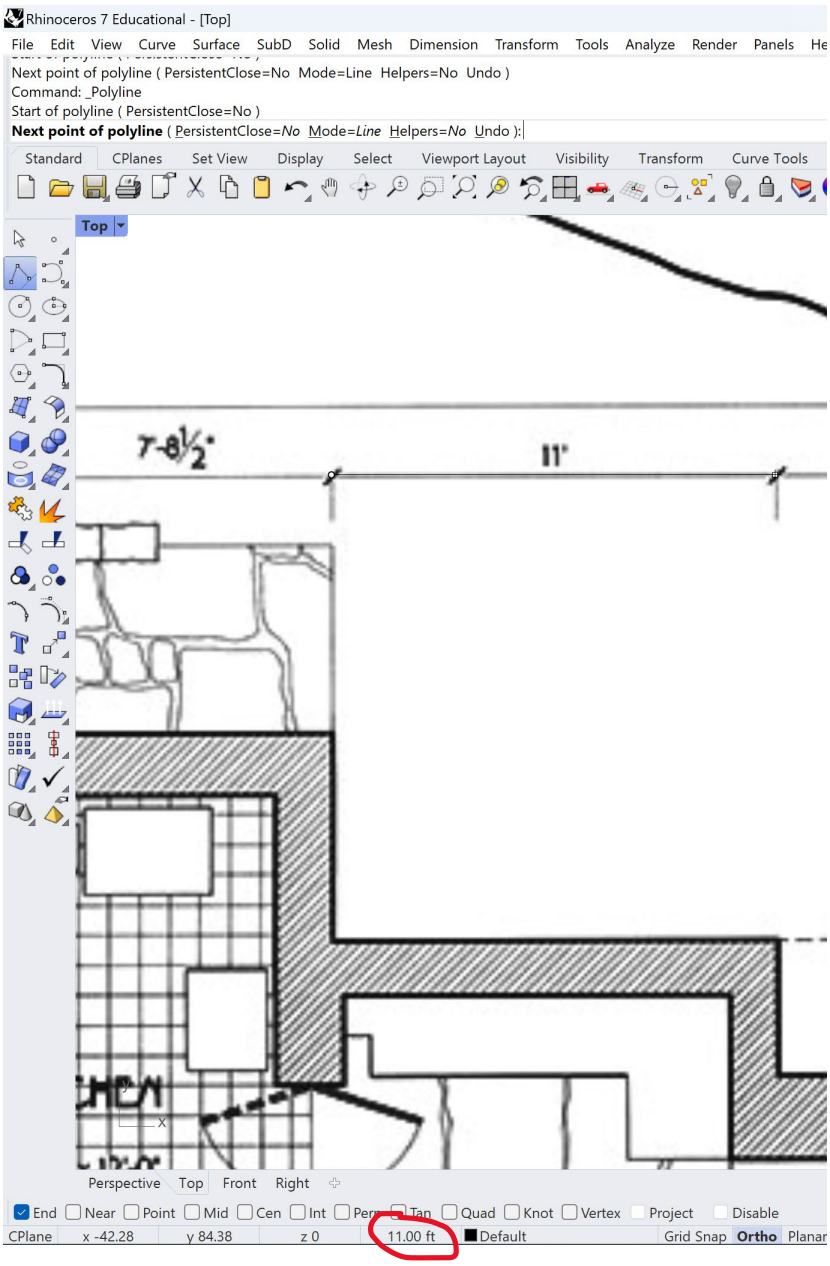


Sometimes, the dimension lines in images are not perfectly straight. If you have picked a long dimension, this slight difference shouldn't matter much.

You will see that your drawing is scaling as you move around. Type the written length of the dimension into the command field. In this example, that number is 98'3."



Hit Enter. Your drawing will now be scaled correctly, and so any lines you draw now should be at their real length. If you want to check your work, pick another dimension, and begin to draw a line along that dimension. As you reach the other end of the dimension line, read the length field at the bottom (circled in red). It should be close to the length of the dimension.



Grasshopper

Clear everything out of rhino. Place two points. Open Grasshopper. Bring out two points, the catenary curve component, a z-vector and a y-vector. Connect them to make a catenary curve. We're going to use this to do the workshop.

Deleting elements

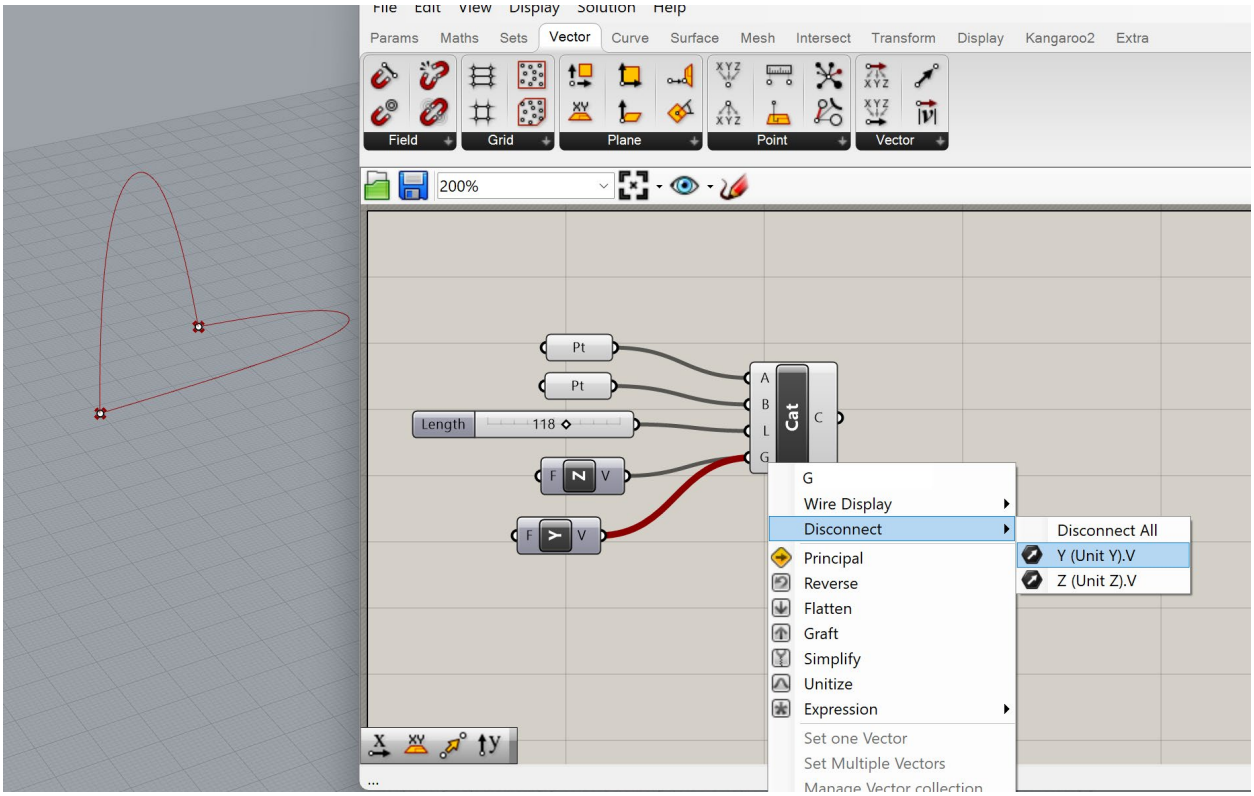
Use Control + X or the Delete key (not Backspace) to delete items you don't want.

Disconnecting elements

Right click on the element on the label of the connection you want to get rid of.

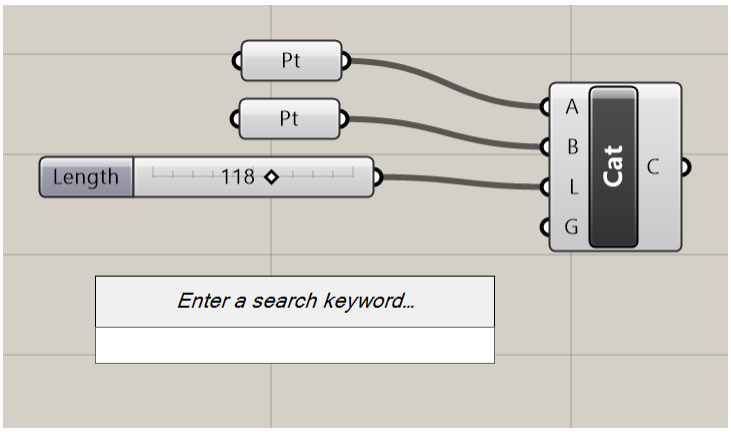
Go to Disconnect → Select the element you want to disconnect.

As you hover between different options, the corresponding line will turn red. Click on the option you want, and the element will disconnect.

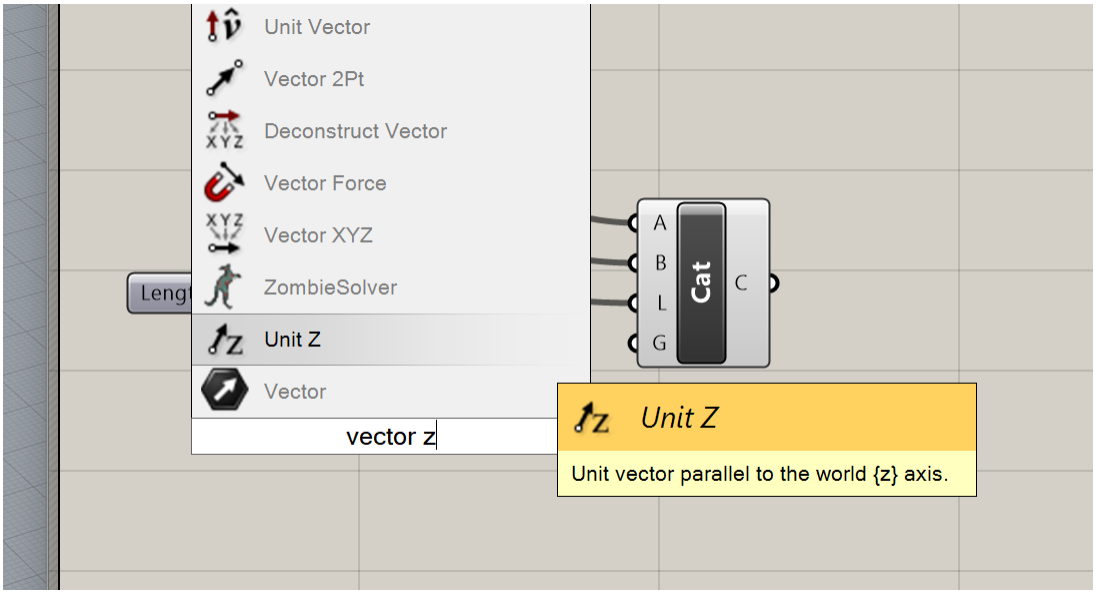


Search bar

If you forget where to find an element you need, but you know the name of it, Grasshopper has a command bar you can use. Simply double-click in any blank area of the grasshopper workspace and a search field will pop up. We've used this before to create number sliders.

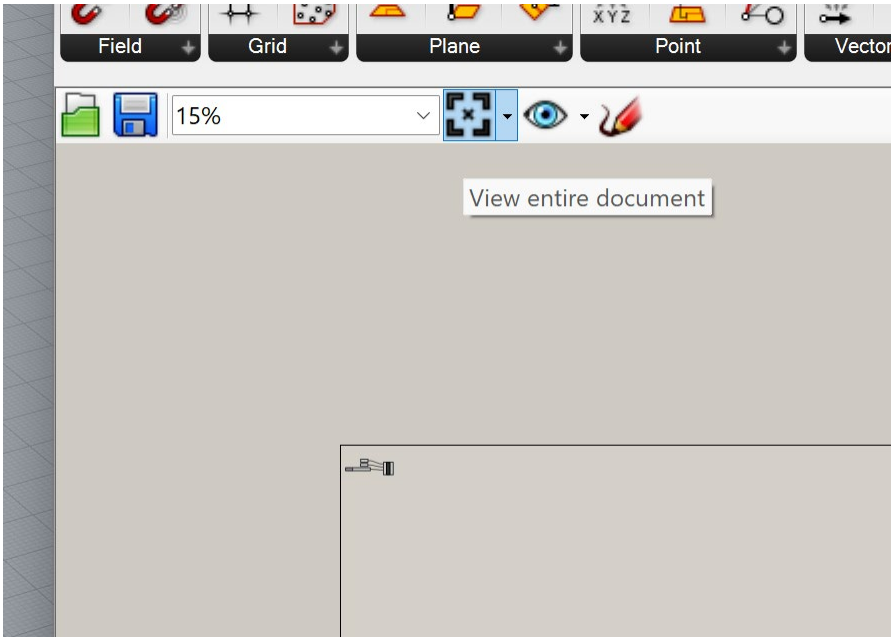


However, you can look up any other element you might need, and the way results pop up is similar to in Rhino.



Zoomed out too far

If you ever get zoomed out too far in Grasshopper, simply click the icon just below the tool tabs that looks like a square with an x inside.



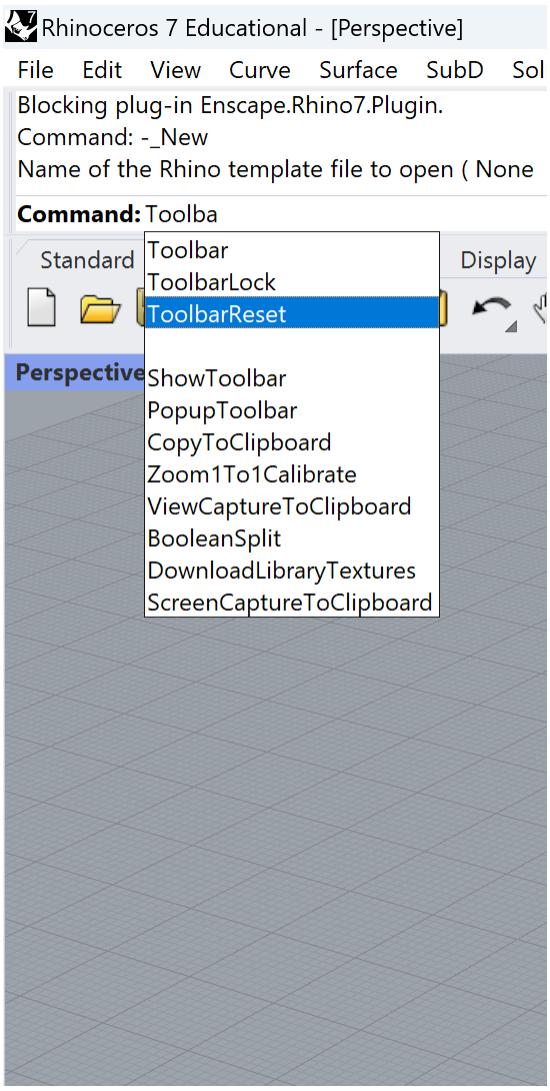
ADDITIONAL HELPFUL NOTES

Fix toolbars

If you accidentally do something like delete a main toolbar, or if your toolbars don't look right or aren't opening, you can reset them to default.

Save your work before doing this!

Type the command "ToolbarReset" and hit Enter. You will now have to close Rhino and re-open it, but when you do, the toolbars will be back to normal.



Unroll surface

This will not be accurate and is not applicable for a number of surfaces, referred to as non-developable surfaces, such as a sphere. However, it will generally work on developable surfaces. These typically include singly curved surfaces such as a cylinder or cone. It will also work on extruded one-way curved walls.