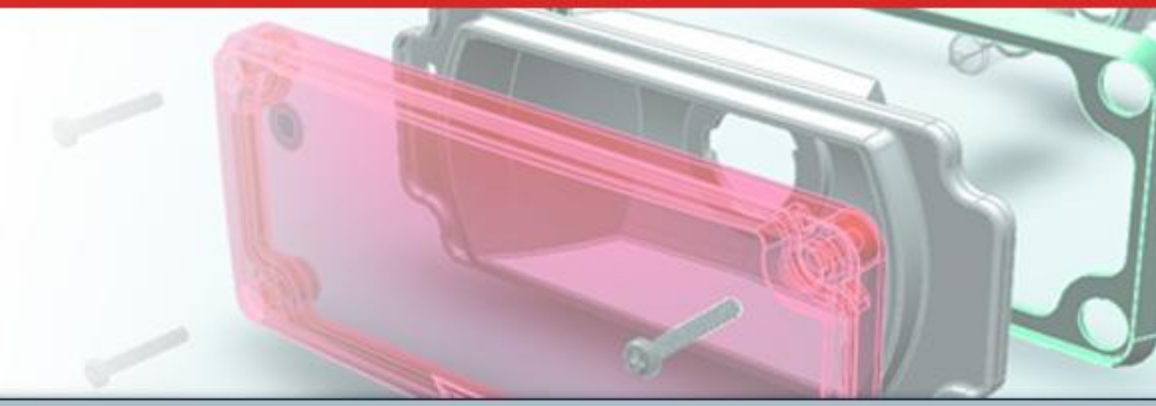
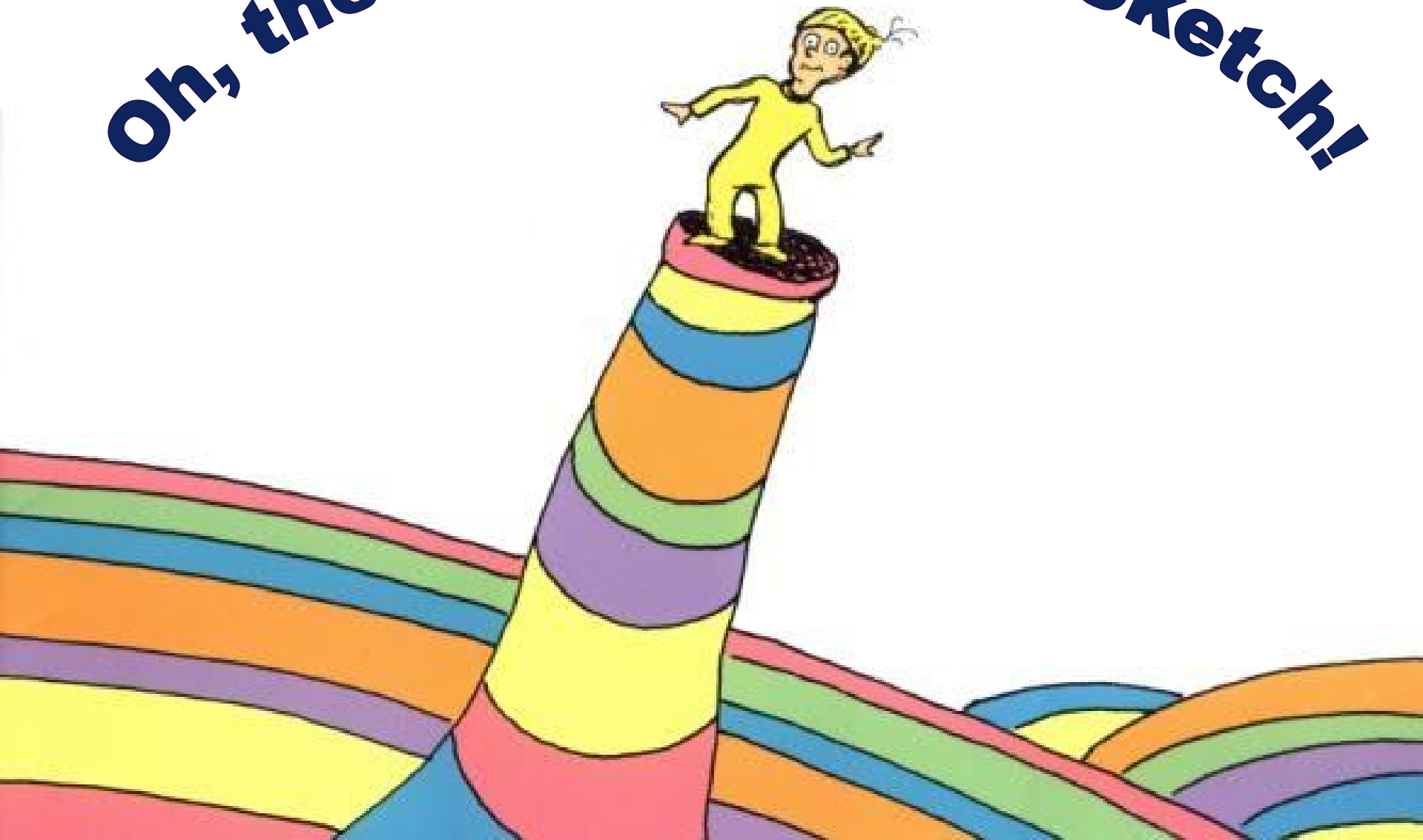
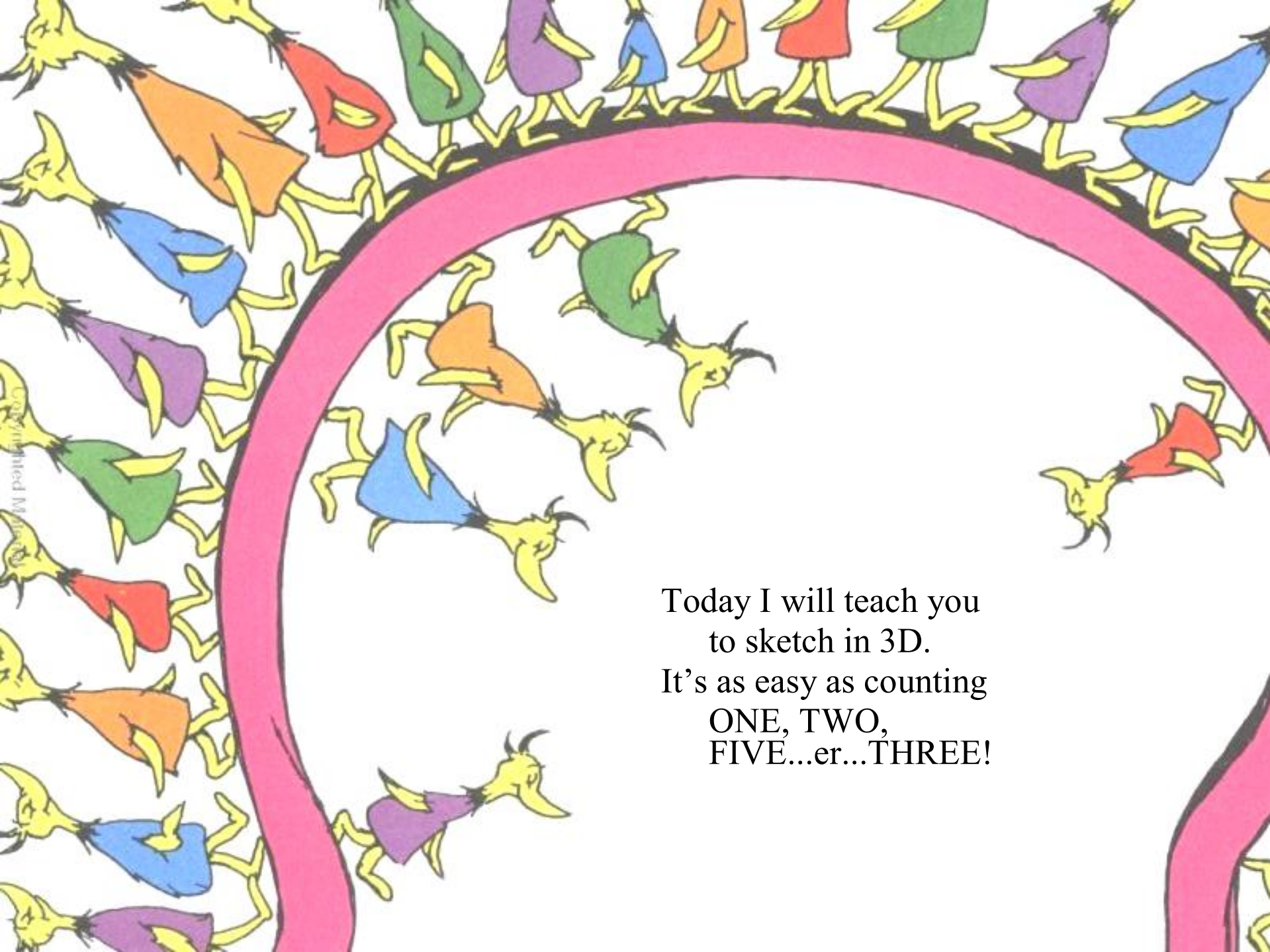


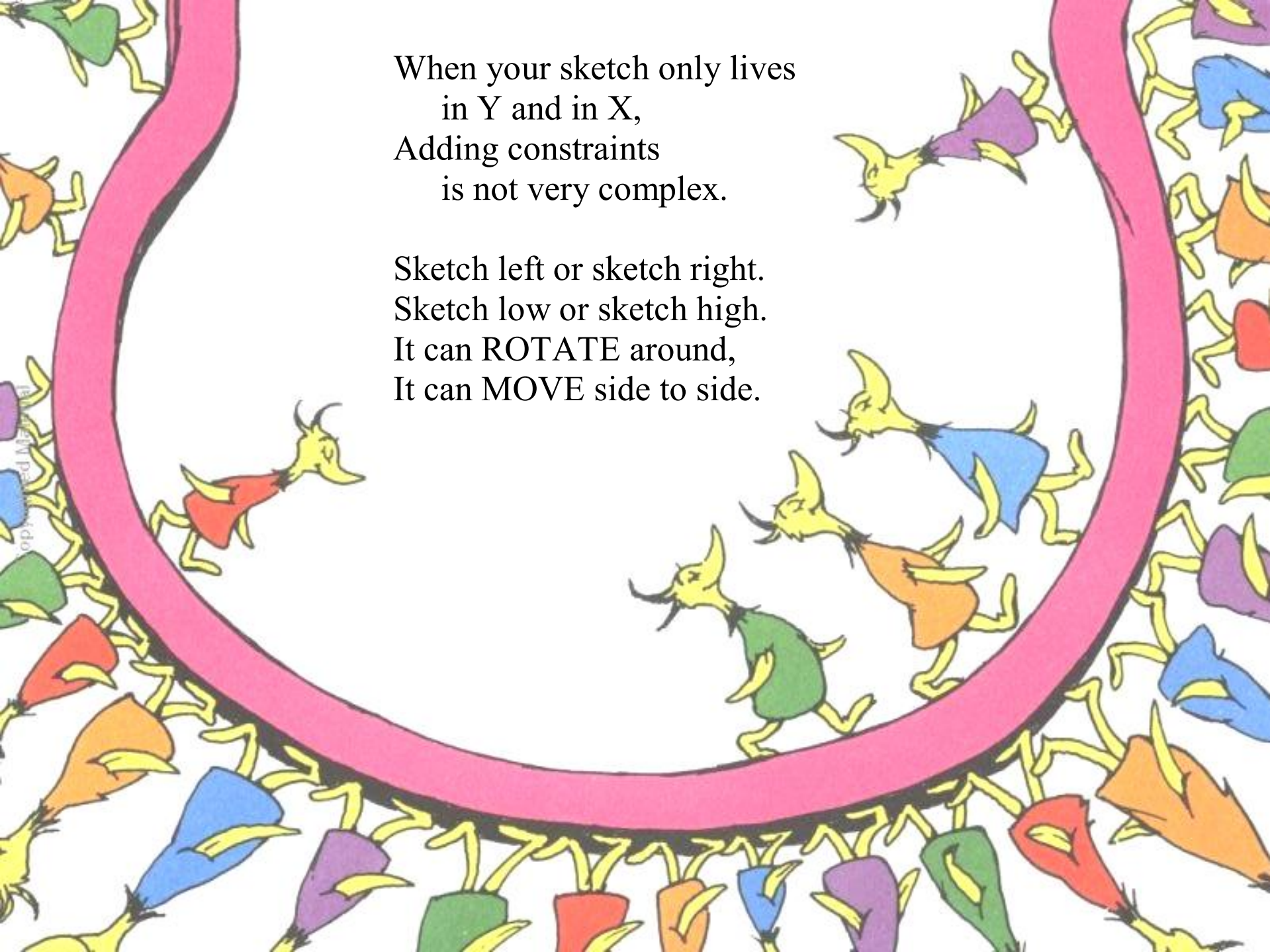
# 3D Sketching Made ~~Easy~~ EASIER

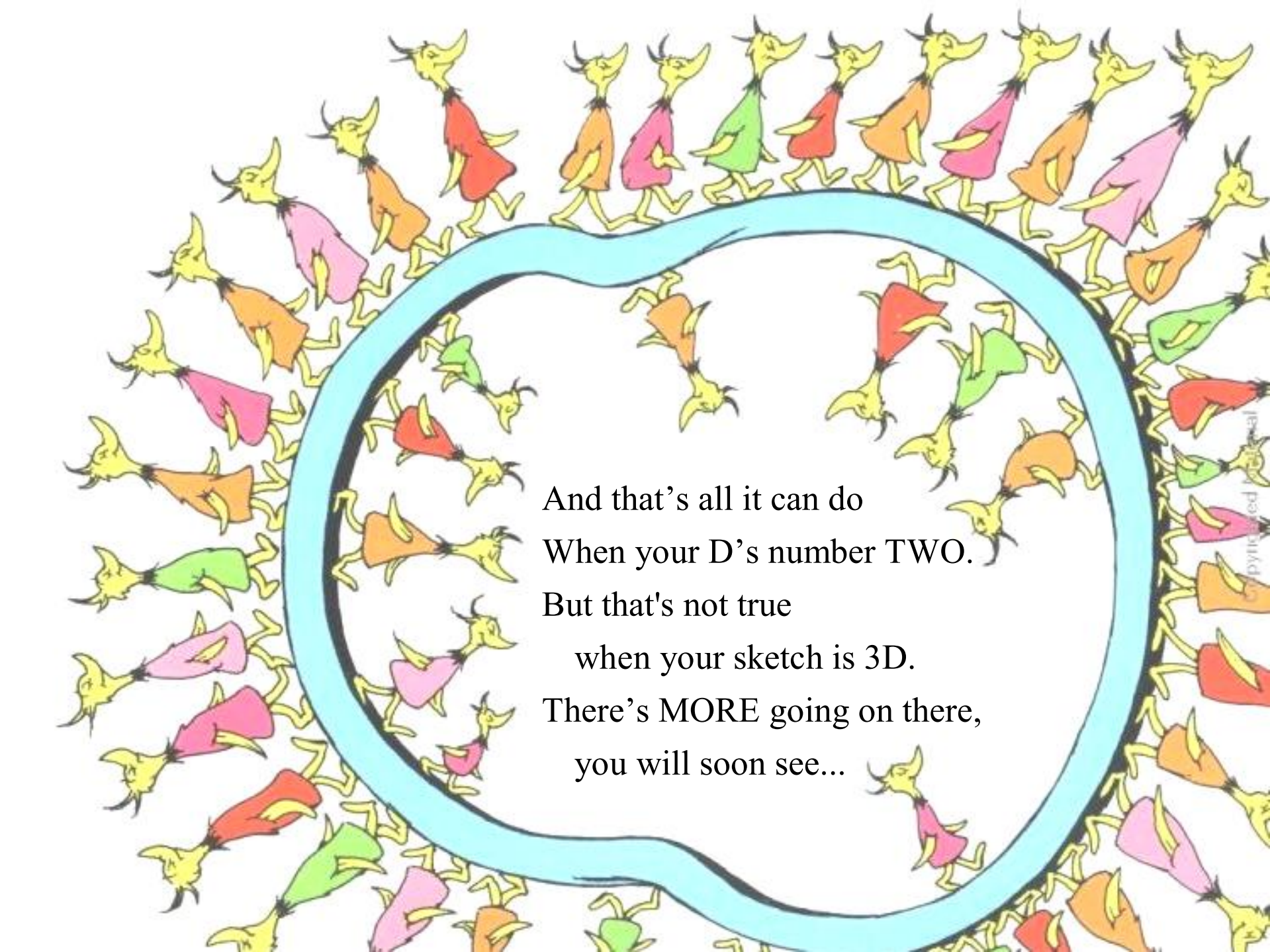
Jason Pancoast
Engineering Manager

Oh, the Sketches you'll Sketch!




Today I will teach you
to sketch in 3D.
It's as easy as counting
ONE, TWO,
FIVE...er...THREE!


When your sketch only lives
in Y and in X,
Adding constraints
is not very complex.
Sketch left or sketch right.
Sketch low or sketch high.
It can ROTATE around,
It can MOVE side to side.


And that’s all it can do
When your D’s number TWO.
But that's not true
when your sketch is 3D.
There’s MORE going on there,
you will soon see...

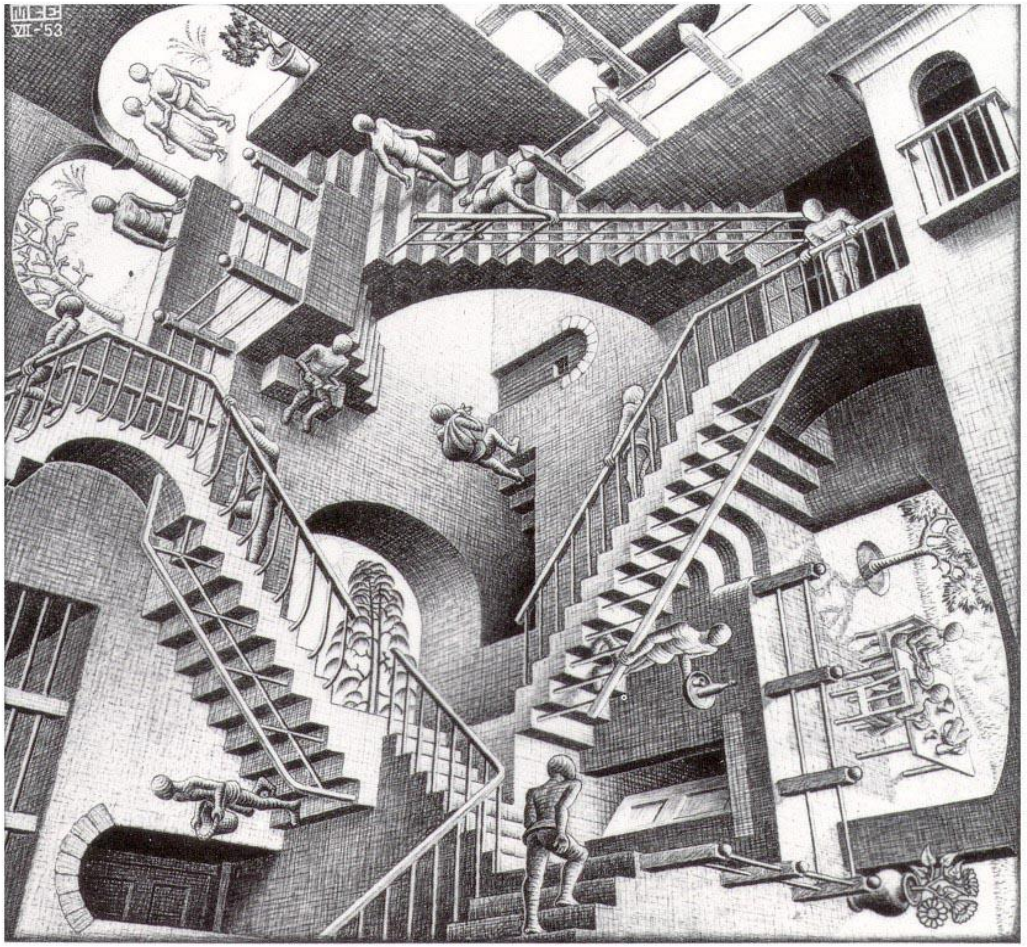
Just when you think that you've sketched something well,
In 3D it's NOT always easy to tell,
And you'll find
what you THOUGHT
you sketched well,
looks like hell!

But THREE is such a good number of D's
For SolidWorks and you
to sketch in
with EASE!

# Yes! 3D Sketching is hard!

- Why is 3D Sketching hard?
  - Because you have TWICE as many degrees of freedom to deal with!
  - Because your mouse only moves in 2D!
  - Because your monitor is only 2D!
- Adding ONE extra axis, actually DOUBLES the number of possible ways an entity can move.
- That means even a simple line is twice as hard to do in a 3D Sketch.

# But it’s not impossible…




Escher’s “Relativity”

# What is it good for?

- Routing
- Weldment
- Sweep Path
- Guide Curve
- Animation Path
- Extrude
- Hole Wizard location
- Explode Line
- Trimming Surface
- Curve-Driven Pattern
- Split Line
- Assembly skeleton

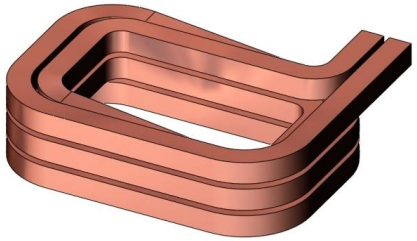
# The Most Important Slide

- There is one simple fact that you MUST know.
- Without this fact, it's pretty much impossible to sketch in 3D.
- If you remember NOTHING ELSE from this presentation, you need to remember this.
- I'll give you a hint...


- Yes. The TAB key.
- The TAB key is the key to 3D Sketching.
- The first example will show you...

# Examples

1. Shower Caddy
2. Handlebars
3. A-Frame
4. Bulb Filament
5. Square Coil
6. Axe Head
7. Fly-Through

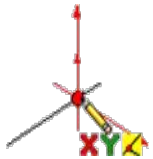



# Example 1: Shower Caddy

- Using the TAB key to change current workplane.
- Understanding Along X, Along Y, and Along Z relationships.
- Drawing lines and arcs in 3D.
- Using construction lines to help constrain a shape in 3D.
- Applying Fillets to a 3D Sketch.
- Using a 3D Sketch as a Sweep path.

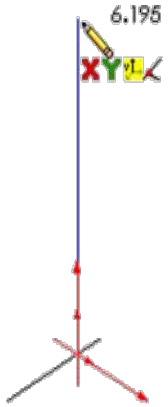
# Caddy – Step 1

- Begin by sketching a line, snapped to the origin.



# Caddy – Step 2

- Drag the line upward and snap ALONG Y.

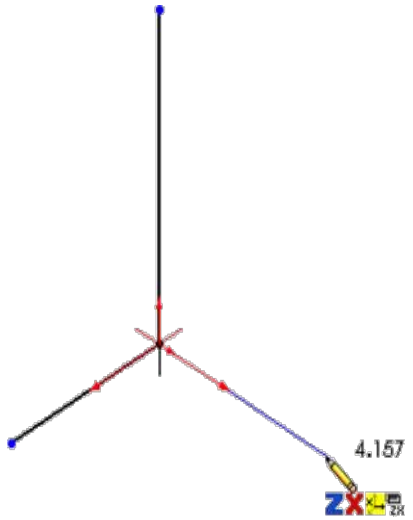


# Caddy – Step 3

- Create another line, starting at the origin.
- Hit the TAB key to toggle to the YZ plane.
- Snap this line ALONG Z.

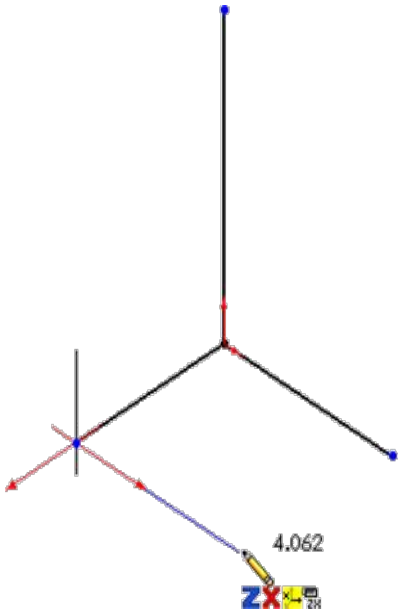
# Caddy – Step 4

- Add a third line.
- Snap to the origin.
- Hit the TAB key to toggle to the ZX (or XY) plane.
- Snap ALONG X.



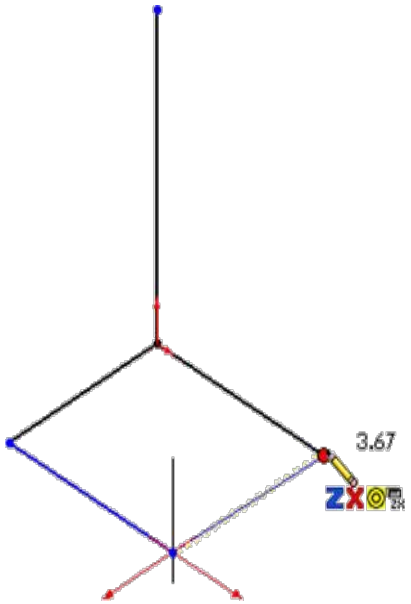
# Caddy – Step 5

- Add a fourth line
- Start from the endpoint of the existing line.
- Snap ALONG X.
- Make it approximately the same length as the previous line.



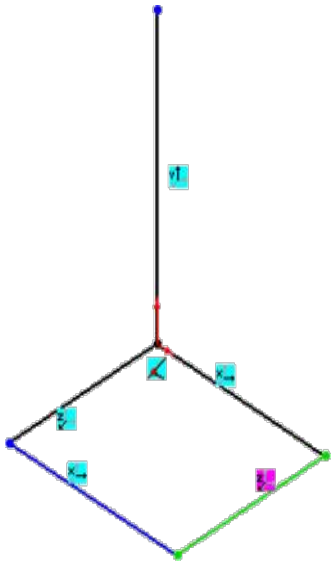
# Caddy – Step 6

- Sketch a fifth line.
- Snap from endpoint to endpoint of the existing lines.



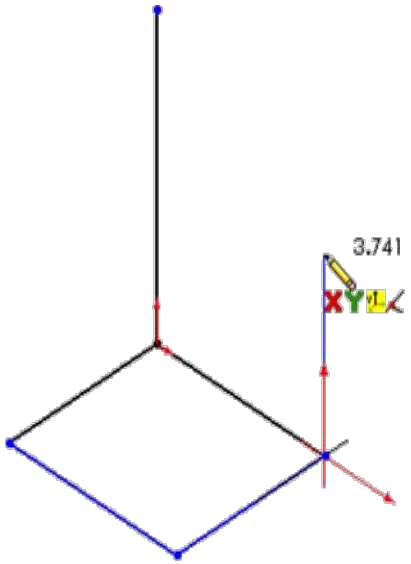
# Caddy – Step 7

- Add a relation.
- Set this line to be ALONG Z.
- This ensures the base is rectangular.



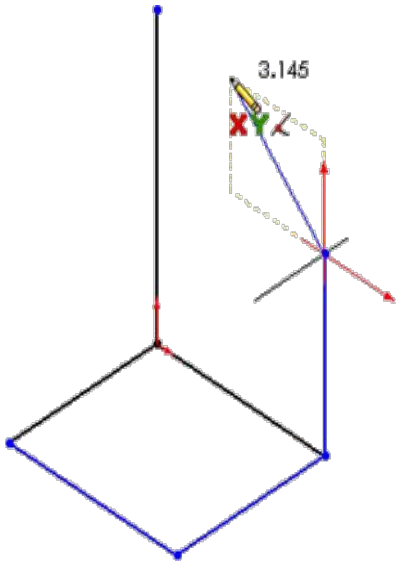
# Caddy – Step 8

- Sketch another line.
- Snap to the endpoint of the existing lines.
- Hit the TAB key to toggle to the XY plane.
- Snap ALONG Y.



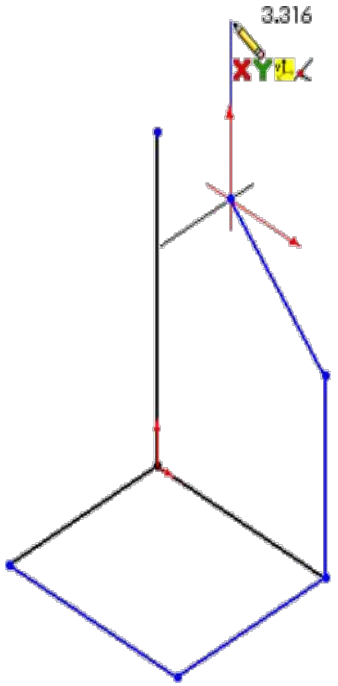
# Caddy – Step 9

- Continue a second line segment.
- Create it in the XY plane, at an angle as shown.



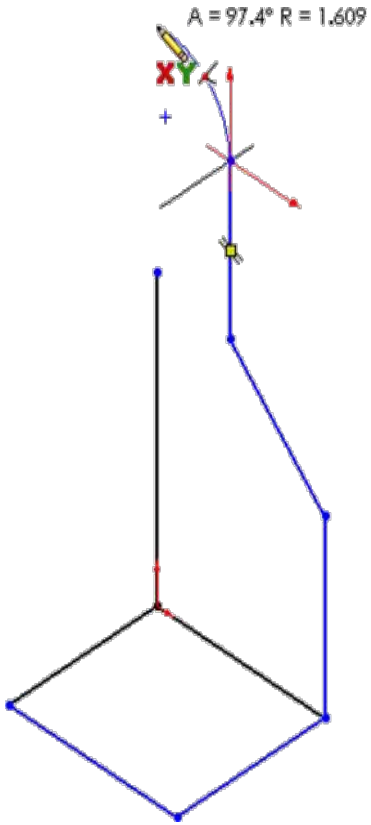
# Caddy – Step 10

- Continue with another line segment.
- Snap ALONG Y.



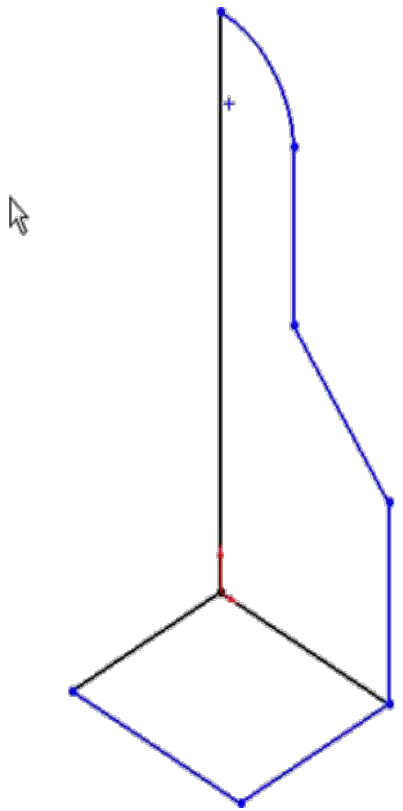
# Caddy – Step 11

- Sketch a Tangent Arc.
- Begin at the endpoint of the existing line.
- Make sure you are still in the XY plane.
- Create an arc about 90 degrees, ending approximately above the origin.



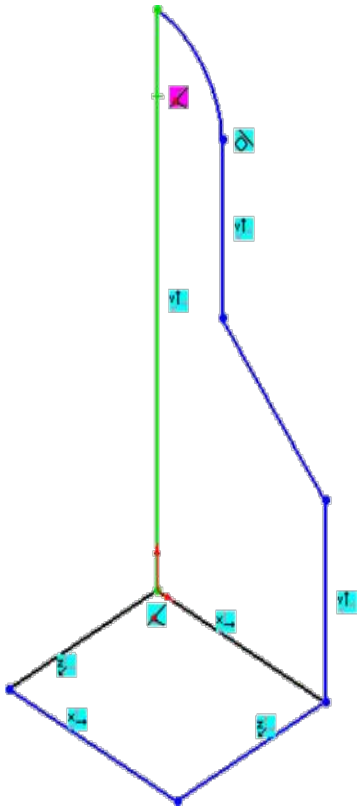
# Caddy – Step 12

- Drag and drop the endpoint of the existing line to close the contour to the arc.
- If necessary, drag the free endpoint of the arc to get it closer.
- You could also select both endpoints and add a MERGE relationship.



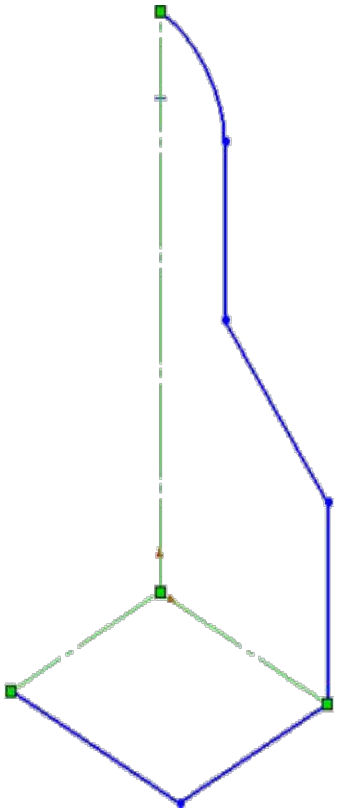
# Caddy – Step 13

- Add a relation.
- Make the centerpoint of the arc COINCIDENT to the line.



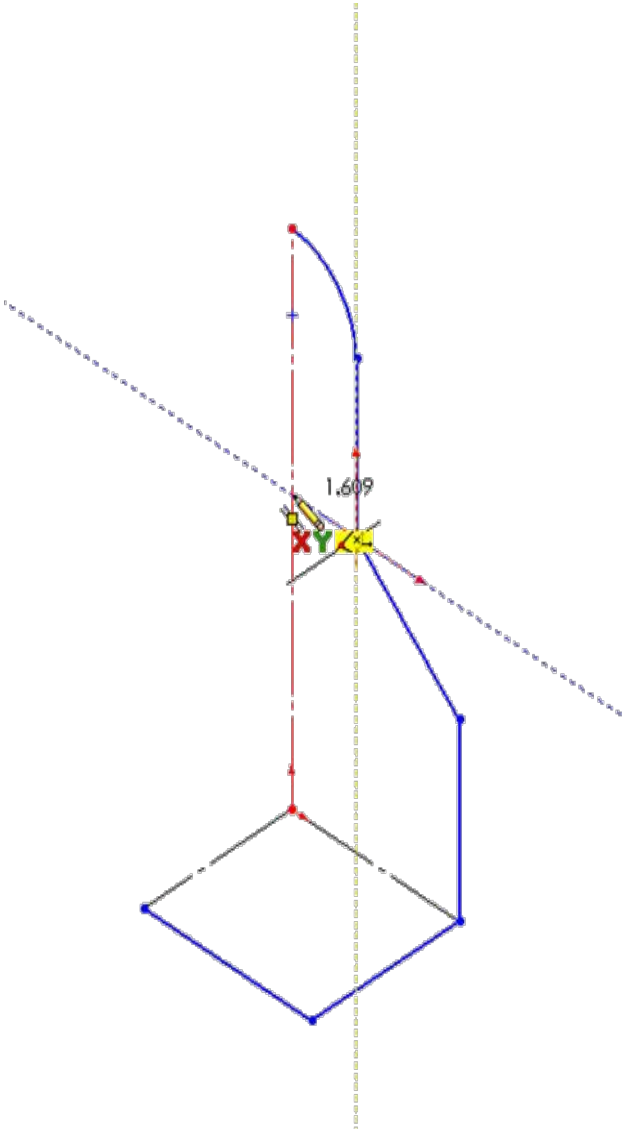
# Caddy – Step 14

- Select the first three lines drawn, and make them into Construction Geometry.



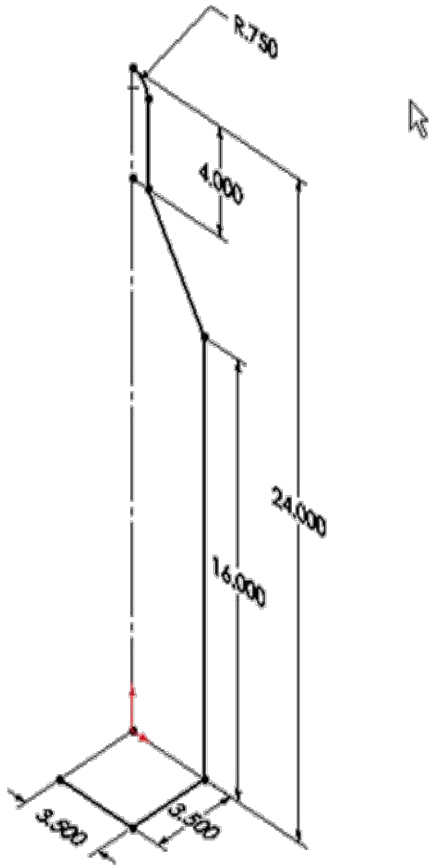
# Caddy – Step 15

- Add one final line to define the shape.
- Begin at the inside corner of the two existing lines.
- Snap the endpoint COINCIDENT to the tall construction line.
- Snap or add an ALONG X relationship to this final line.



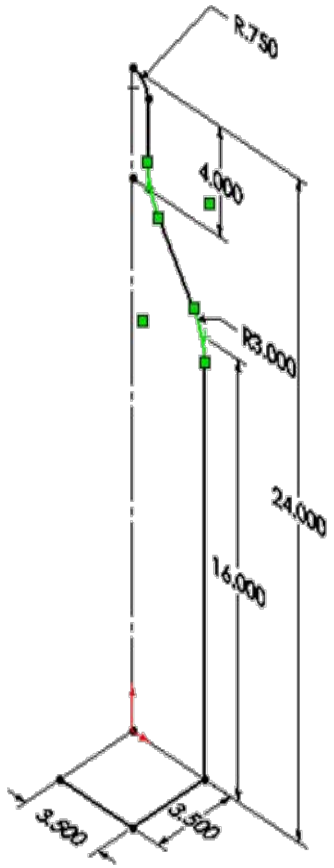
# Caddy – Step 16

- Add dimensions to fully define the shape.
- Where possible, dimension to a line instead of a vertex, to better control the orientation of the parameter.



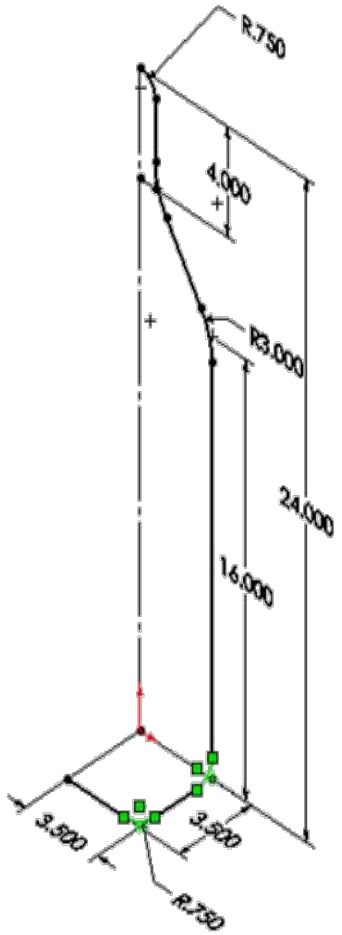
# Caddy – Step 17

- Use the Sketch Fillet tool to add R3.00” fillets to two corners.



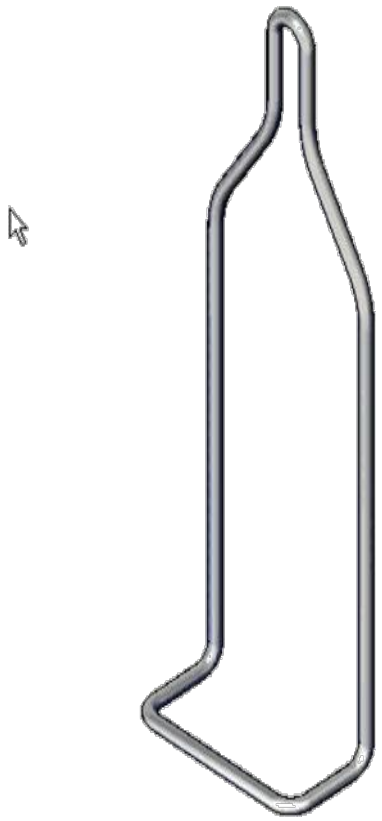
# Caddy – Step 18

- Use the Sketch Fillet tool to add two more fillets, this time R0.75”, to two more corners.
- Note that the fillets do not need to lie in the same plane.



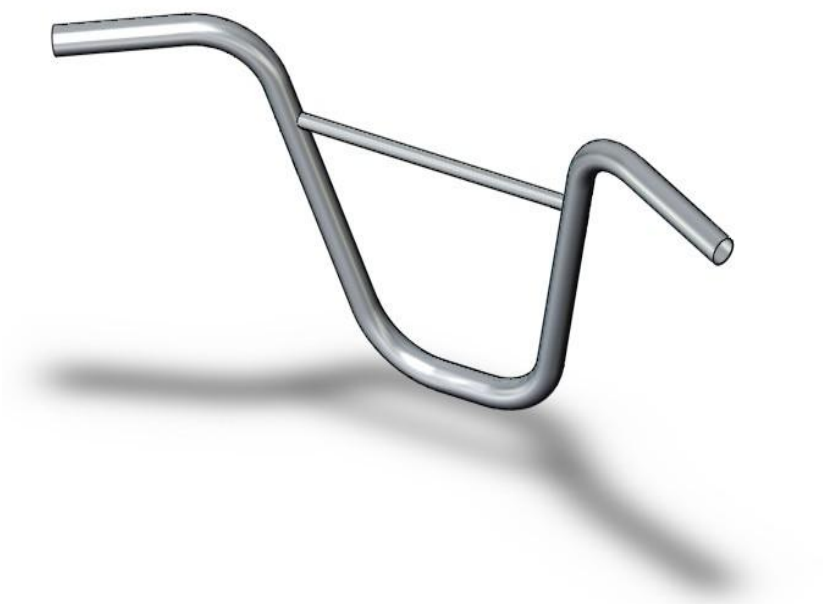
# Caddy – Done

- Finally, create the solid.
- Insert a 2D sketch to create the profile for a sweep (profile is a Ø0.50” circle).
- Create a Boss-Sweep, using the 3DSketch for the Path.
- To complete the model, insert a Mirror feature to create the other half.



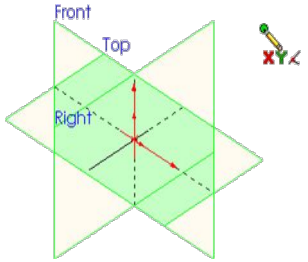
# Example 2: Handlebars

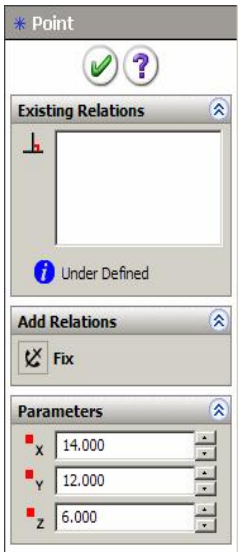
- Working with Sketch Points.
- Adding dimensions to reference planes.
- Working with rectangles in 3D.
- Creating 3D constructions to capture design intent.
- Dimensioning to Sketch Fillets in a 3D Sketch.
- Using the Selection Manager for choosing a sweep path.



# Handlebars – Step 1

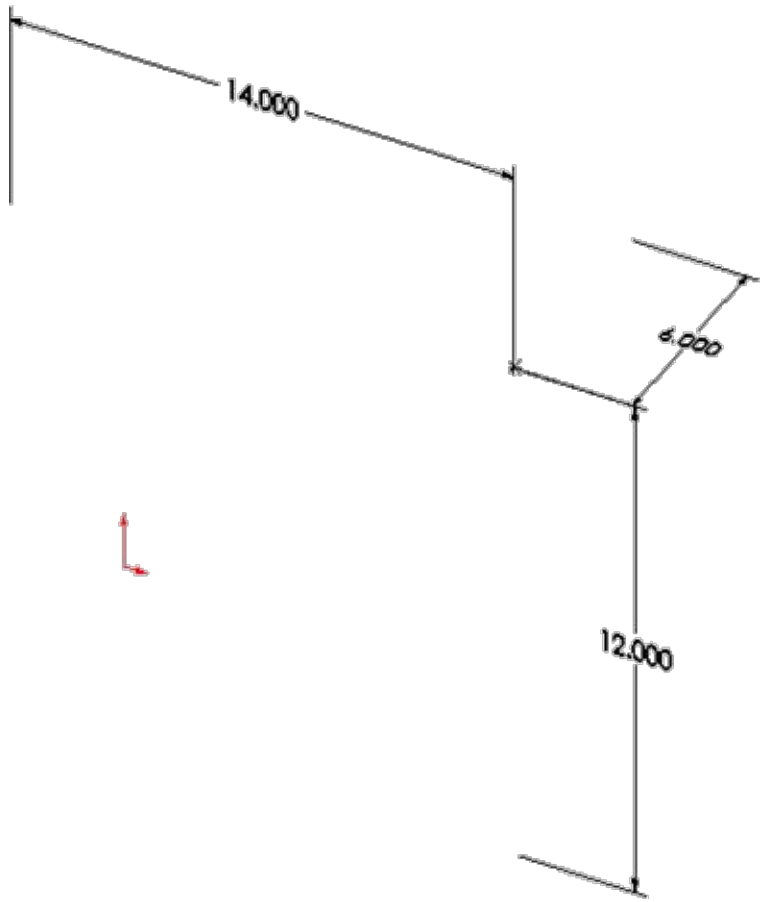
- Create a new part.
- Insert a new 3D Sketch.
- Switch to Isometric view.
- Create a Sketch Point at any random location.
- Edit the parameters in the PropertyManager to position the point precisely:
  - X = 14 in.
  - Y = 12 in.
  - Z = 6 in.



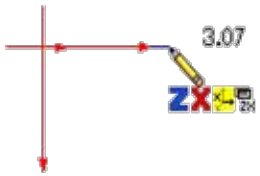
# Handlebars – Step 2

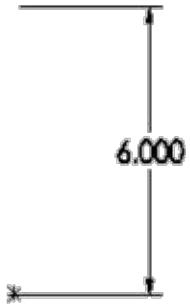
- To properly constrain the point, add dimensions.
- The 3D Sketch allows you to dimension from reference planes to sketch points.
- This method locks in the desired orientation.
- Create three dimensions, one to each of the three default reference planes (Front, Top, Right).
- Keep the values that appear, since they are correct.



# Handlebars – Step 3

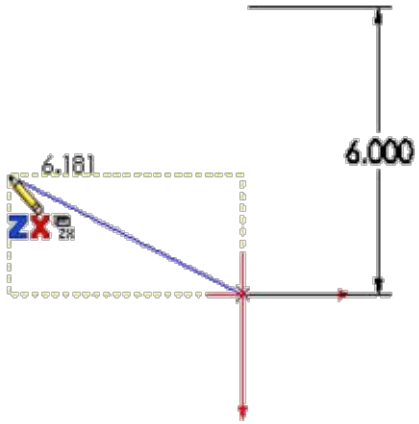
- Switch to a “Top” view.
- Notice the workplane automatically switches to XZ to match the view.
- Sketch a line ALONG X, about 3 inches long from the origin.



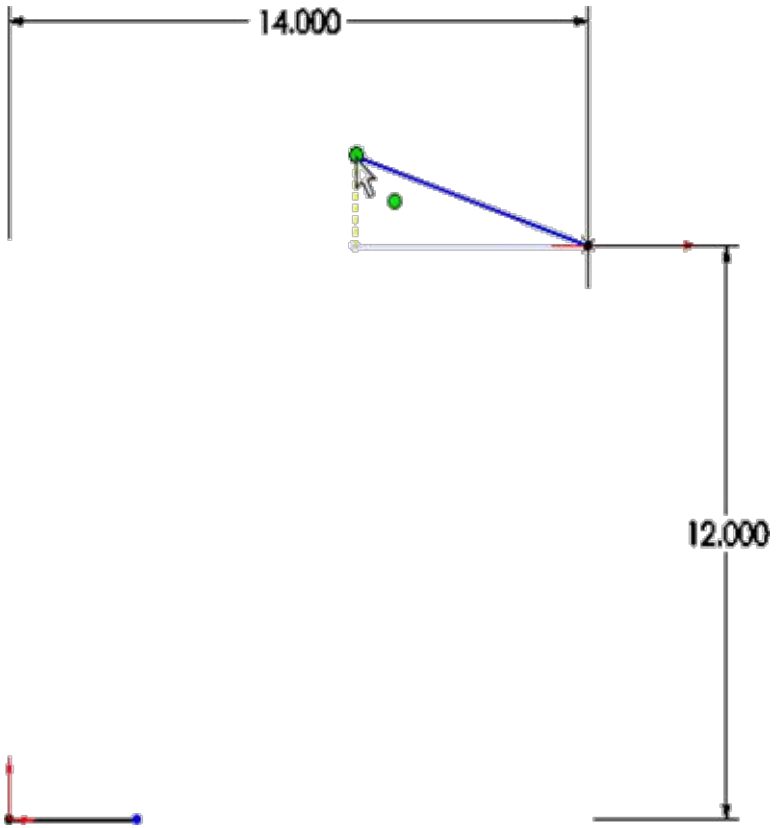
# Handlebars – Step 4

- Sketch another line while in the “Top” view.
- Begin at the 3D Sketch Point.
- Angle the line slightly toward the origin.



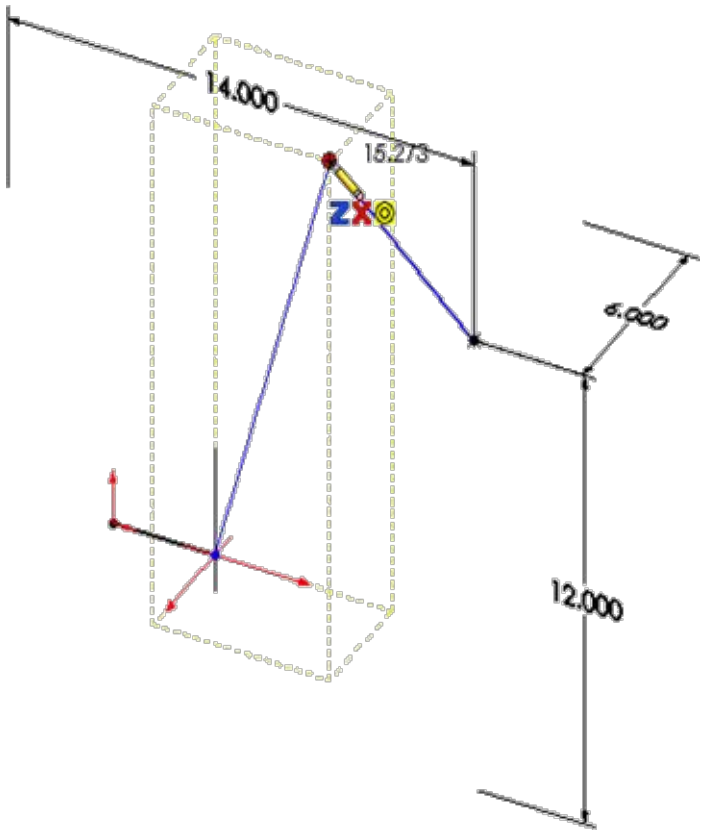
# Handlebars – Step 5

- Switch to a “Front” view.
- Grab the end of the line you just drew, and drag it upward a bit.
- Next we will have to figure out how to constrain the compound angle of this line.



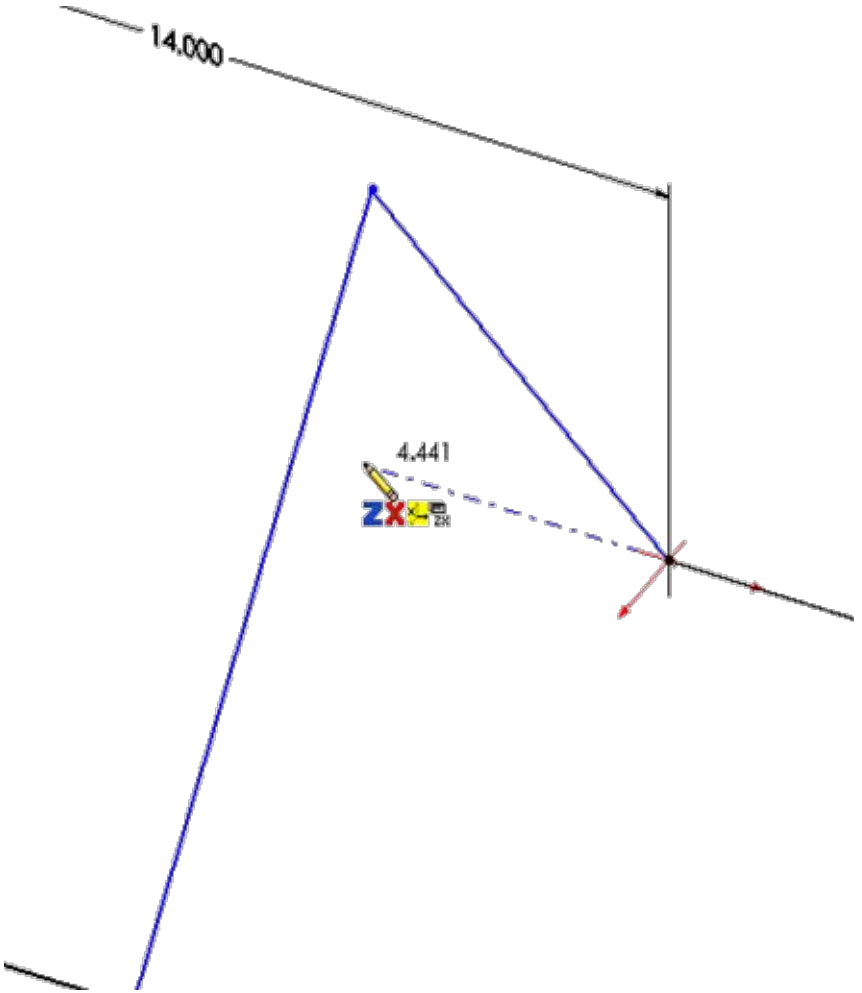
# Handlebars – Step 6

- Add one more line to connect the endpoints of the two existing lines.
- Notice the yellow wireframe box that helps indicate that this line is at a compound angle through 3D space.



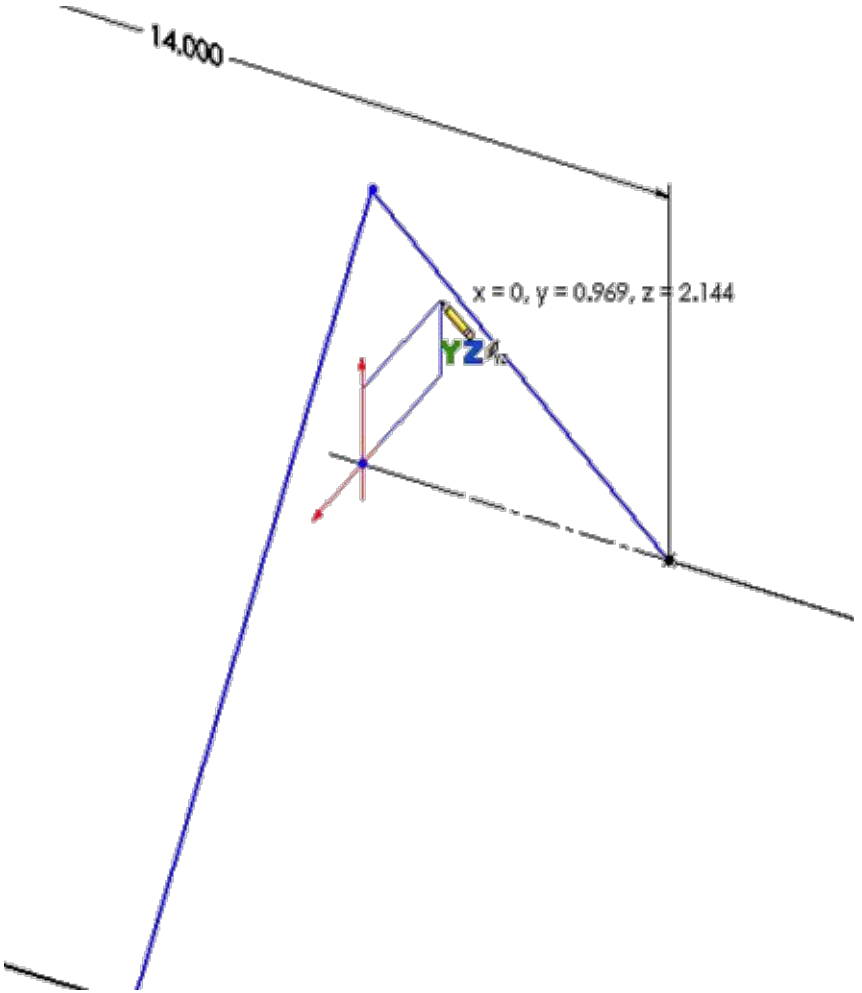
# Handlebars – Step 7

- To constrain the handlegrip, we will make a construction box.
- Begin with a line ALONG X.



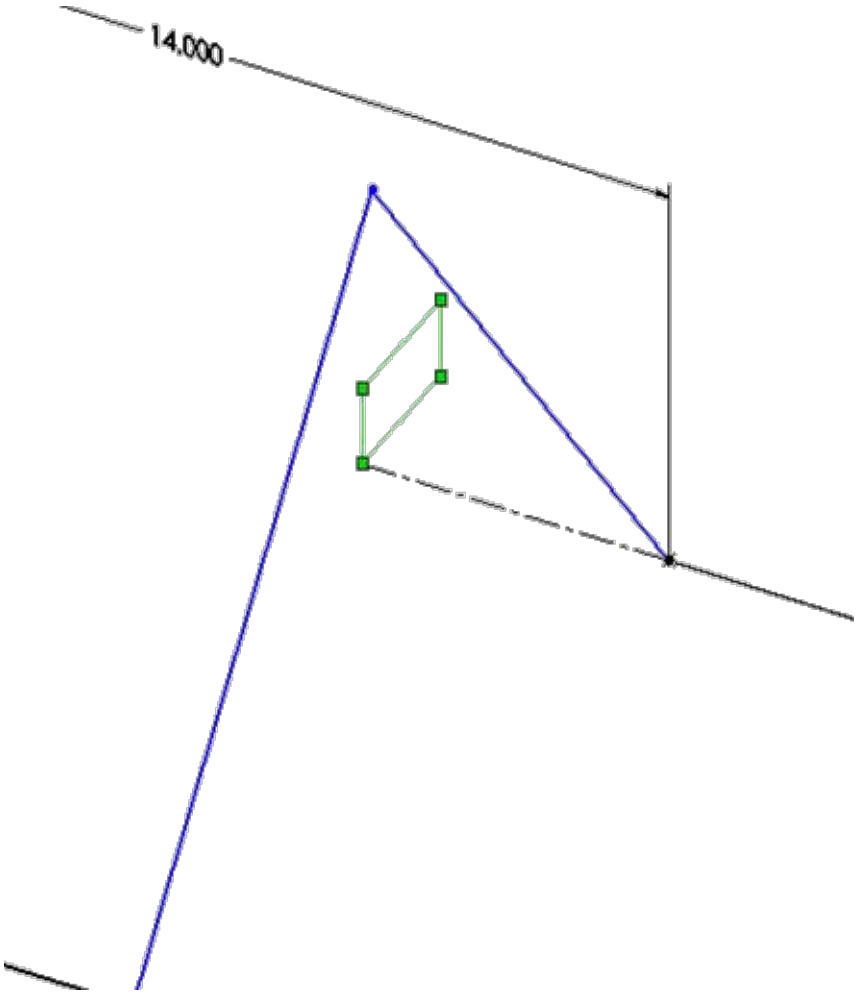
# Handlebars – Step 8

- Turn on the Rectangle tool.
- Start the rectangle at the endpoint of the line you just drew.
- As you create the rectangle, you can toggle the TAB key until you are working in the YZ plane.



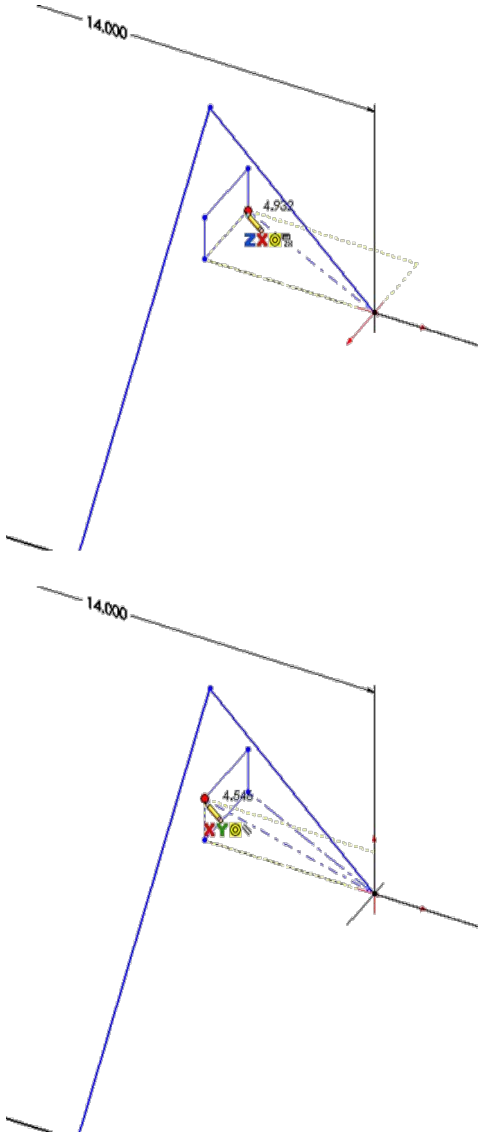
# Handlebars – Step 9

- Select the four lines of the rectangle and make them construction geometry.



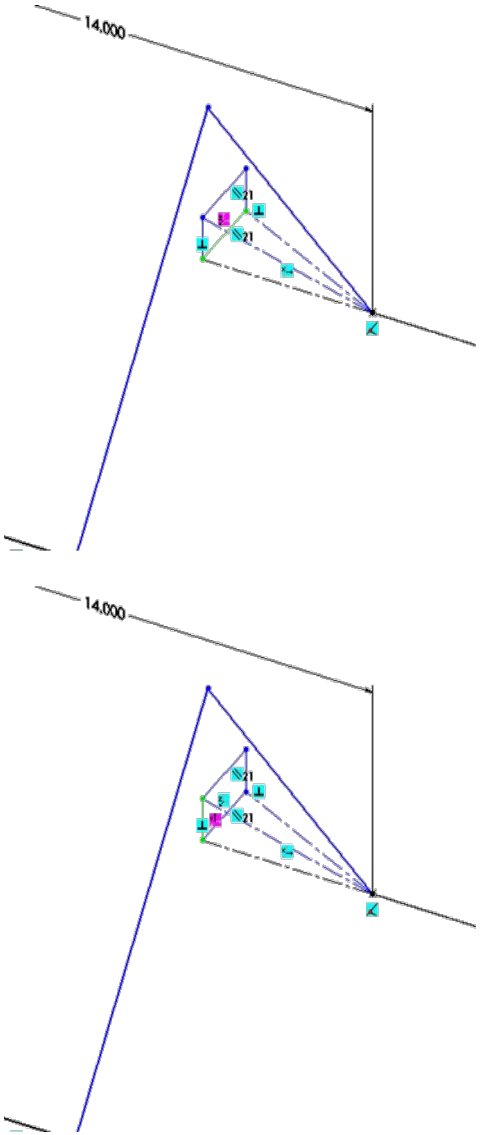
# Handlebars – Step 10

- Create two more construction lines, to form the diagonals of the construction box.
- One line is in the XZ plane.
- The other line is in the XY plane.
- We will be able to dimension the angles these lines form, in the correct 3D orientation.



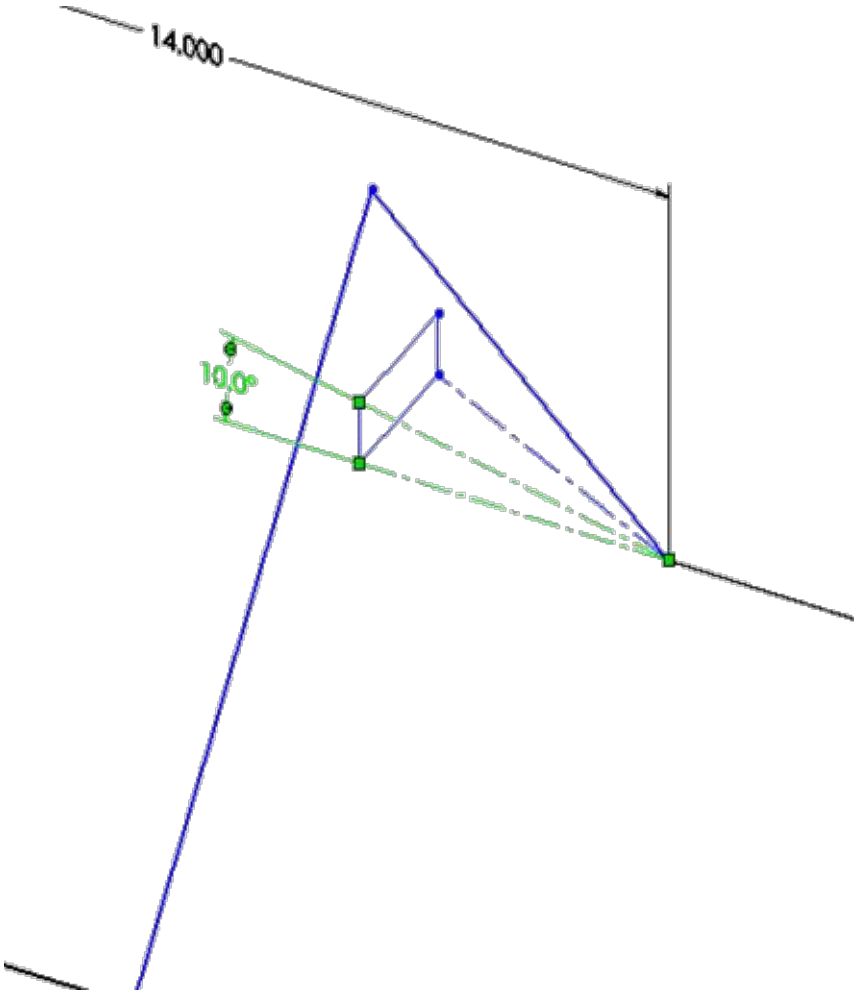
# Handlebars – Step 11

- Notice that the rectangle is made with Perpendicular and Parallel relationships.
- The location in the XZ plane was not captured automatically by SolidWorks.
- Select the bottom side of the rectangle and set it ALONG Z.
- Select another side and set it to be ALONG Y.



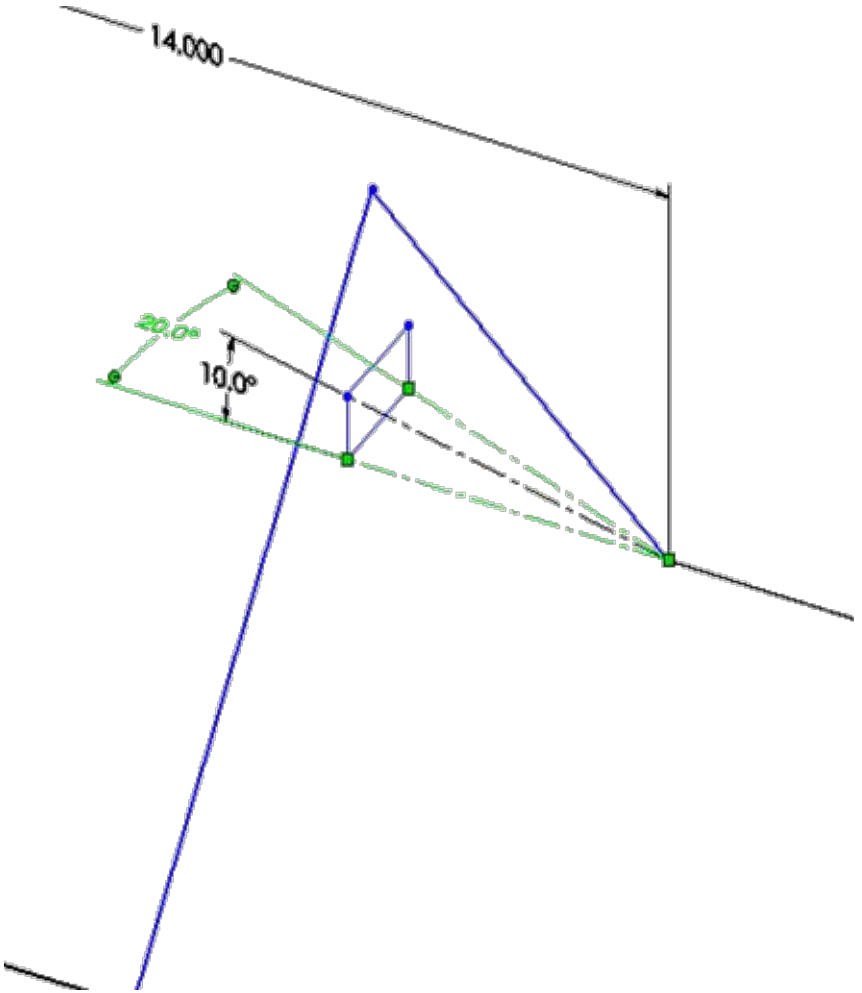
# Handlebars – Step 12

- Now we can add dimensions.
- Constrain the angle in the XY plane to be 10 degrees.



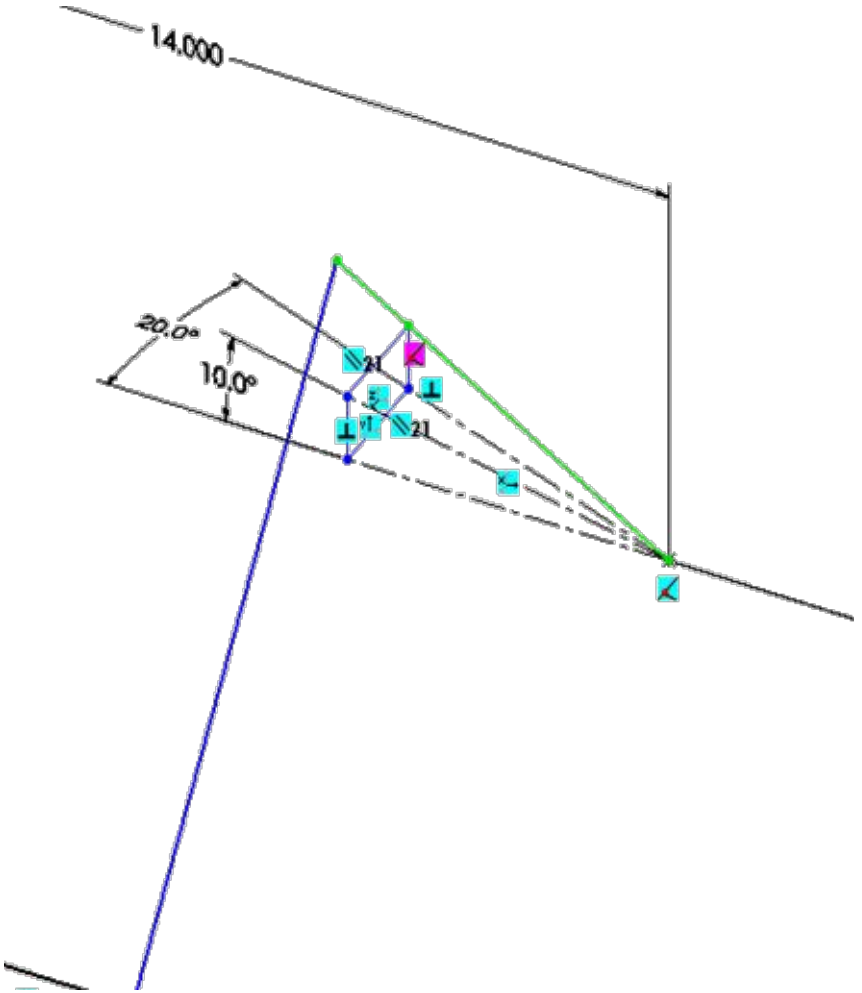
# Handlebars – Step 13

- Constrain the angle in the XZ plane to be 20 degrees.



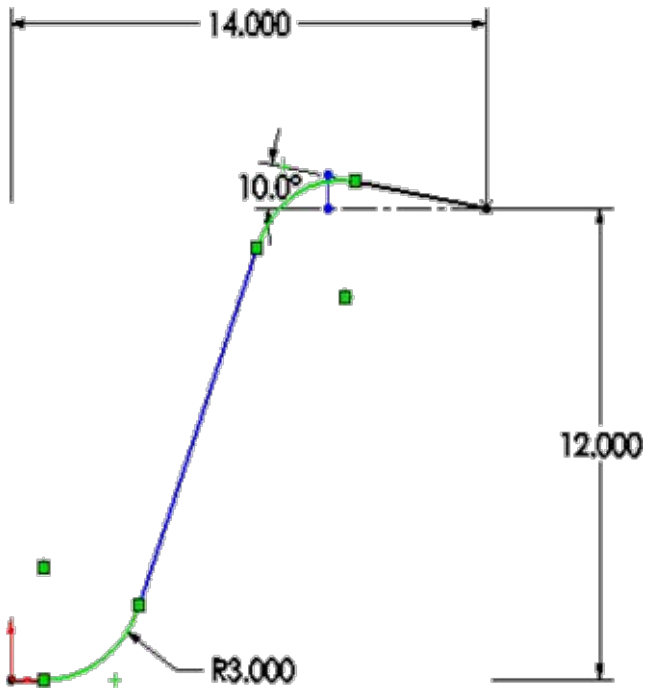
# Handlebars – Step 14

- Select the far corner of the construction box and make it COINCIDENT to the angled line of the handlegrip.
- This captures the desired design intent of the handlegrip's compound angle.



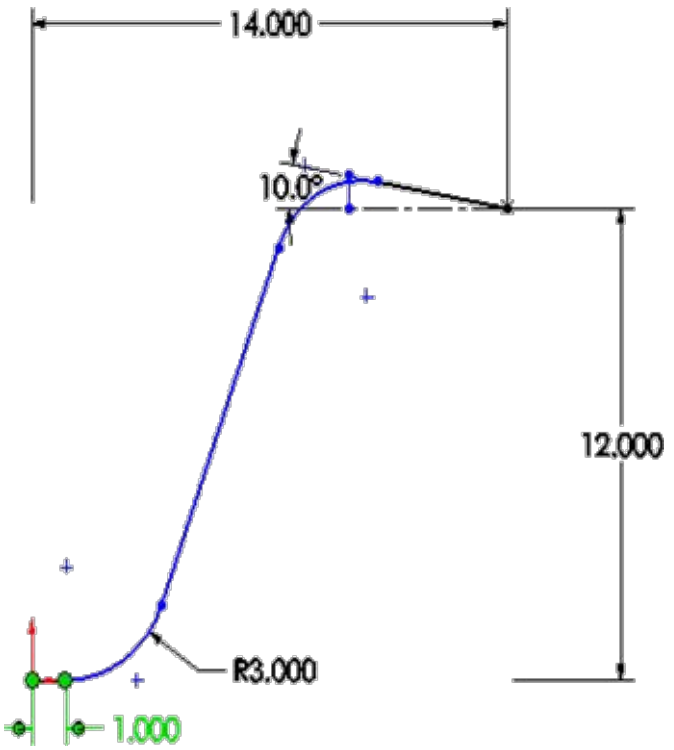
# Handlebars – Step 15

- Add R3.0” fillets to the two sharp corners of the handlebar.
- Switch to a “Front” view.



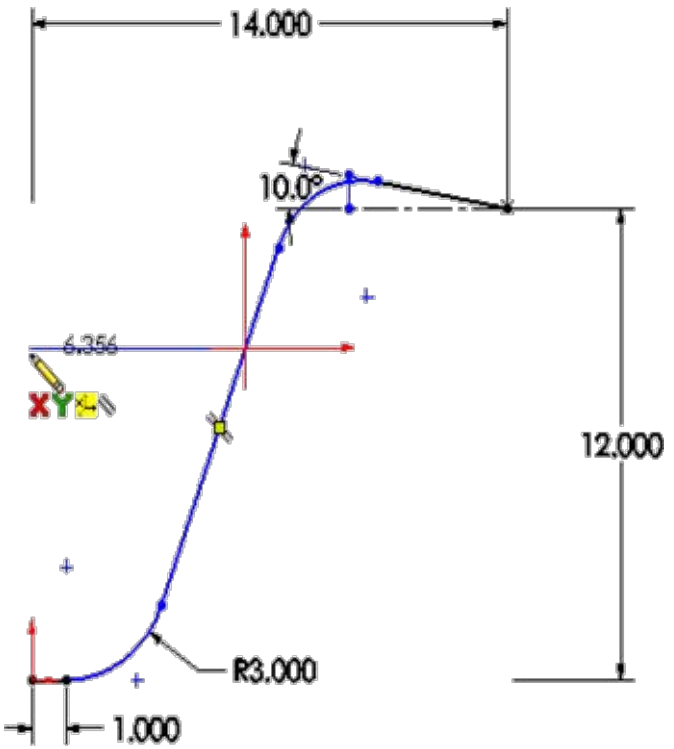
# Handlebars – Step 16

- To dimension the straight portion near the origin, you must select the endpoints of the line.
- If you select the line, the dimension would be created to the virtual sharp, which is not what we want in this case.
- Make the straight portion 1.00” long.



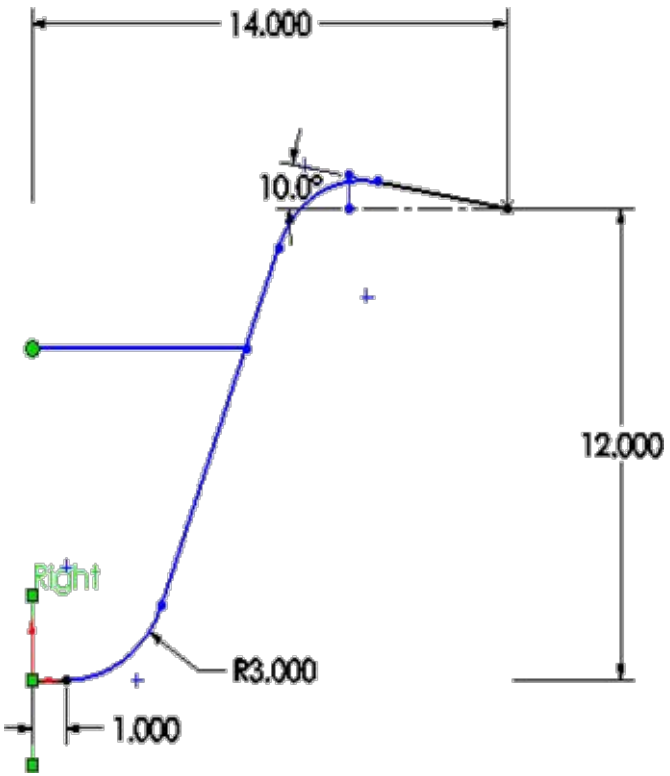
# Handlebars – Step 17

- Using a 3D Sketch, it's easy to determine the location of a 12 in. crossbar.
- Create another line, beginning at the large slanted line of the handlebar, snapping ALONG X.
- End the line above the origin.



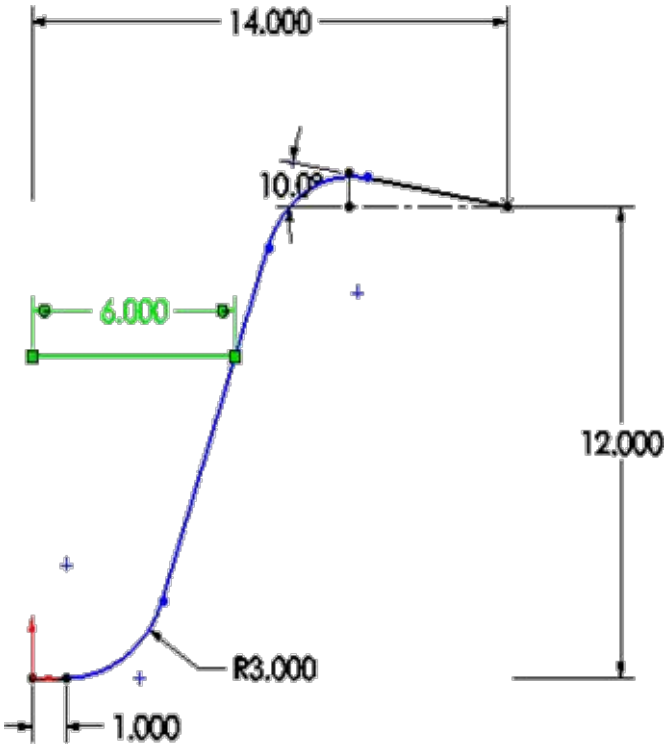
# Handlebars – Step 18

- Select the endpoint of the line and the Right reference plane.
- Add an ON SURFACE relation.
- This is similar to a coincident relationship. It is used to keep 3D sketch entities constrained to a plane or model face.



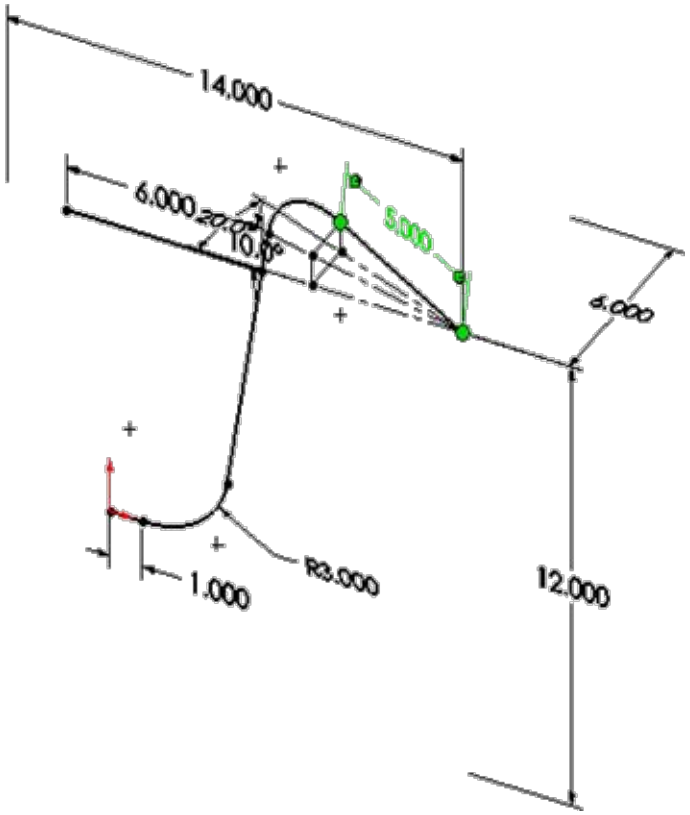
# Handlebars – Step 19

- Add a dimension to define the line to be 6 inches long.
- Notice the line moves up or down as needed to achieve the correct length.



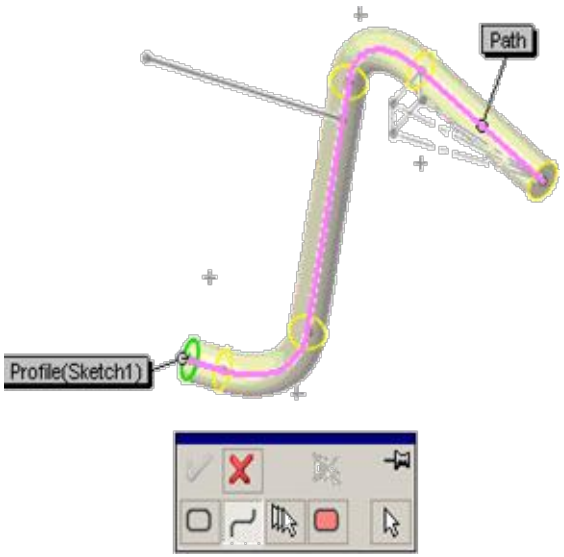
# Handlebars – Step 20

- Complete the sketch by adding a final dimension to the endpoints of the handlegrip region.
- Set the length to 5 inches.



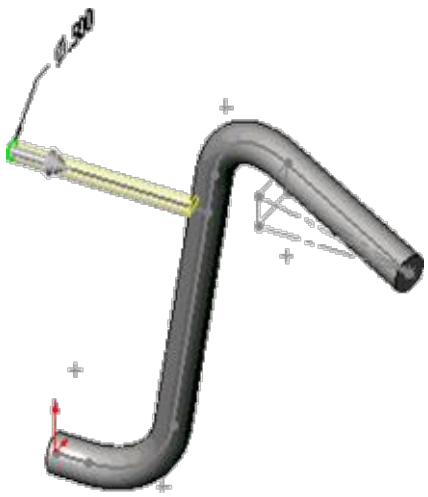
# Handlebars – Step 21

- To create a solid requires just a few more steps:
  - Exit the 3D Sketch.
  - Create a Ø1.0” circle on the Right plane to use as the sweep profile.
- Insert a Boss-Sweep.
- The Selection Manager will appear to confirm the portion of the 3D Sketch that is desired for the sweep path.



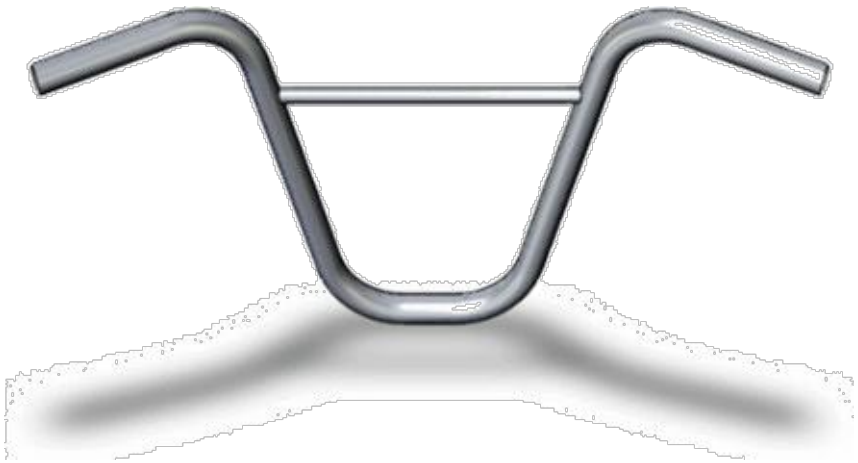
# Handlebars – Step 22

- Extrude a Ø0.50” circle UP TO NEXT to complete the design.
- Mirror the solid, Shell it, and you’re done.



# Handlebars – Done

- Mirror the solid, Shell it, and you're done.

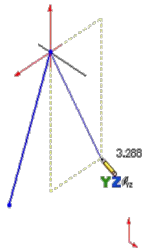


# Example 3: Swing Frame

- Sketching technique with more TAB key.
- Capturing symmetry in 3D Sketch:
  - The EQUAL relationship.
  - Using MIDPOINT and ALONG X (or Y or Z) to capture symmetry.
- 3D Sketch Planes.

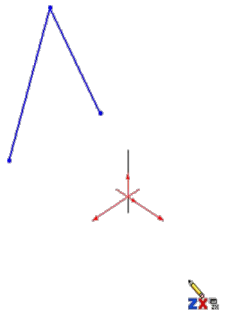
# Swing Frame – Step 1

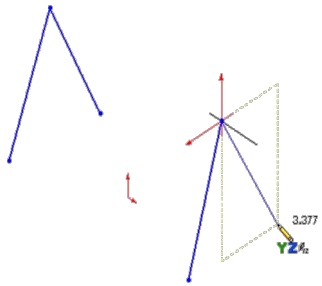
- Start with a line.
- Use the TAB key to begin the 3D Sketch in the ZX plane.
- Click once to begin the line, and then TAB to the YZ plane to complete the line.
- Draw a second line, also in YZ, to form an “A” shape.

# Swing Frame – Step 2

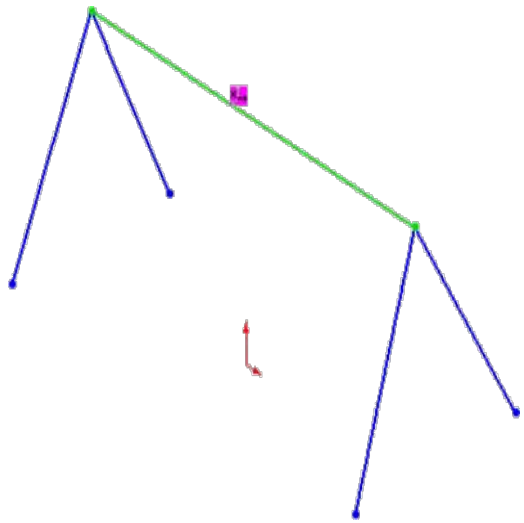
- Repeat the previous procedure to create a second “A” shape.
- Use the TAB key to switch to the ZX plane.
- Begin the line, then TAB to YZ to complete the line.
- Draw another line also in YZ.



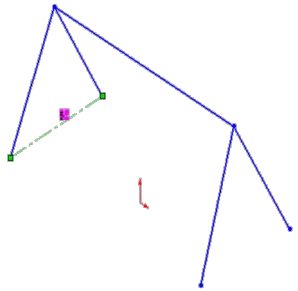
# Swing Frame – Step 3

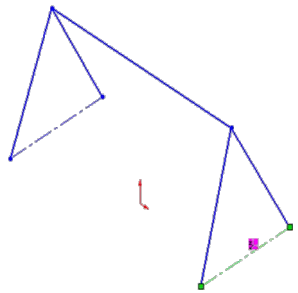
- Draw a line connecting the two corners of the “A” shapes.
- Set this line to be ALONG X.



# Swing Frame – Step 4

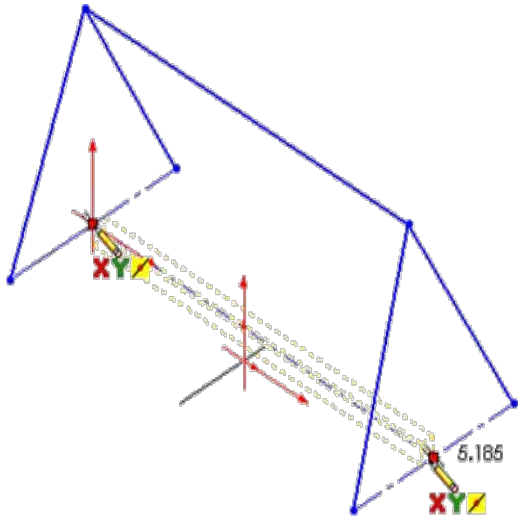
- Create a construction line connecting the endpoints of one of the “A” shapes.
- Set that line to be ALONG Z.
- Repeat for the other “A” shape.





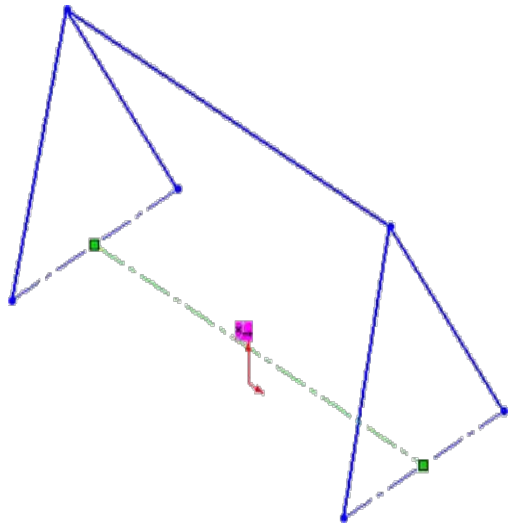
# Swing Frame – Step 5

- Draw a line.
- Begin by snapping to the midpoint of one construction line.
- End by snapping to the midpoint of the other construction line.



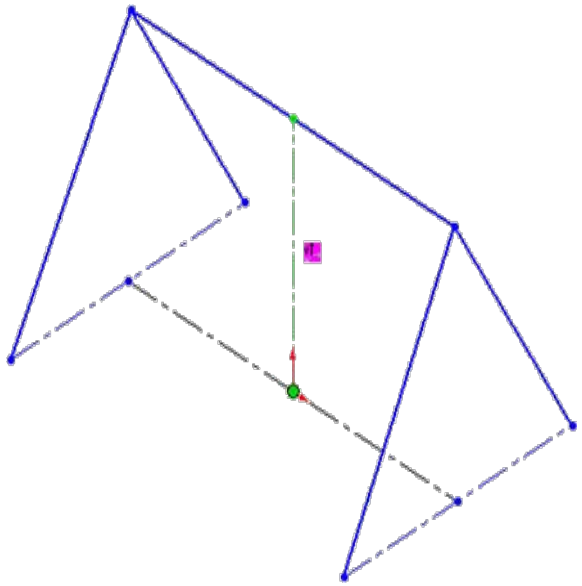
# Swing Frame – Step 6

- Set the long construction line to be ALONG X.
- This has the effect of constraining front-to-back symmetry for the A-frame.



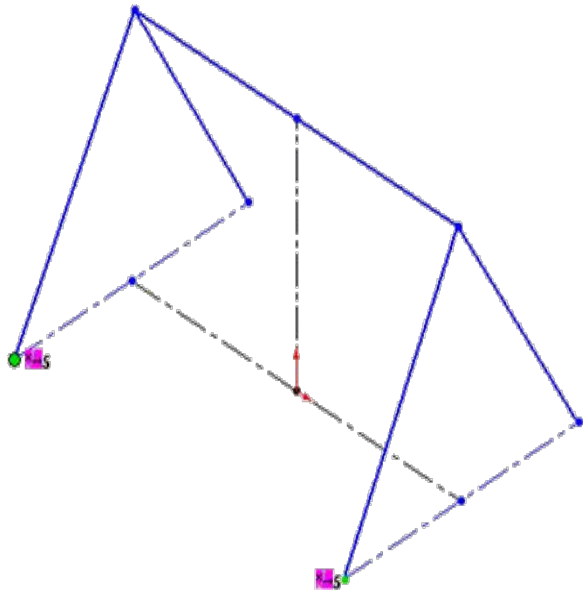
# Swing Frame – Step 7

- Repeat the centerline procedure to capture left-to-right symmetry:
- Create a new construction line
- Begin at the midpoint of the long construction line.
- End at the midpoint of the long regular line.
- Set this new construction line to be ALONG Y.
- Finally, grab the endpoint of the new centerline and drop it onto the Origin.



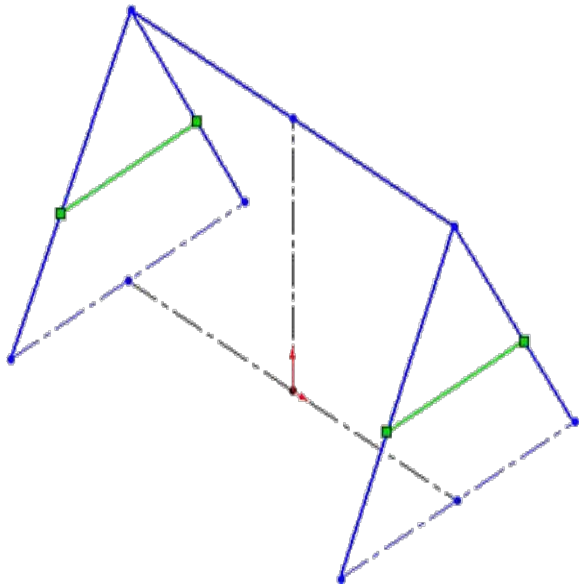
# Swing Frame – Step 8

- Select the two front endpoints of the A-frames and set them ALONG X.
- This constrains the two A-frames to be the same size.



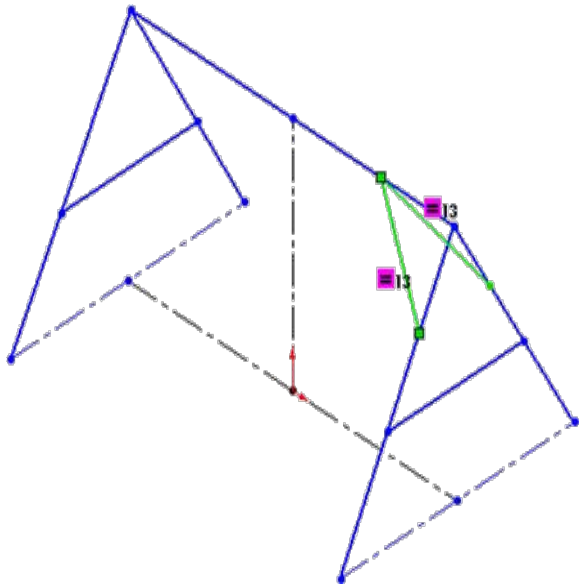
# Swing Frame – Step 9

- Draw two more lines ALONG Z.
- Snap the lines to the existing A-frames to form two crossbars.



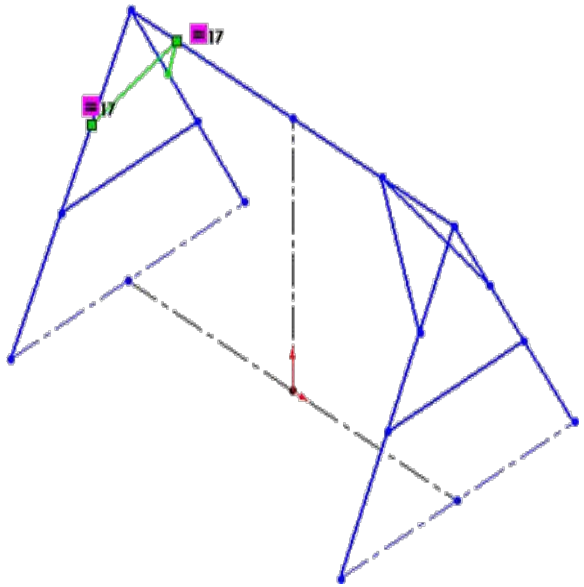
# Swing Frame – Step 10

- Draw two lines to represent struts to strengthen the corner of the A-frame.
- Set these two lines to be EQUAL to each other.
- This captures front-to-back symmetry of the design.



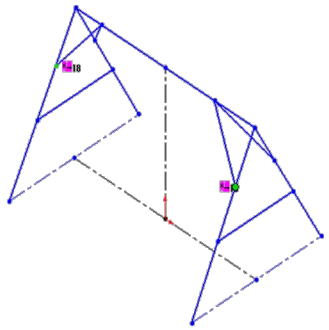
# Swing Frame – Step 11

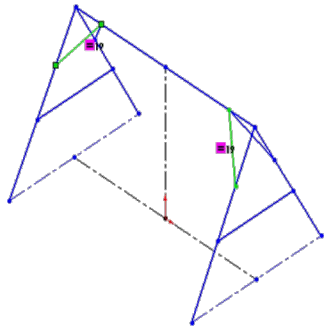
- Repeat on the other side.
- Draw two more lines representing two more struts.
- Also set these EQUAL to each other.



# Swing Frame – Step 12

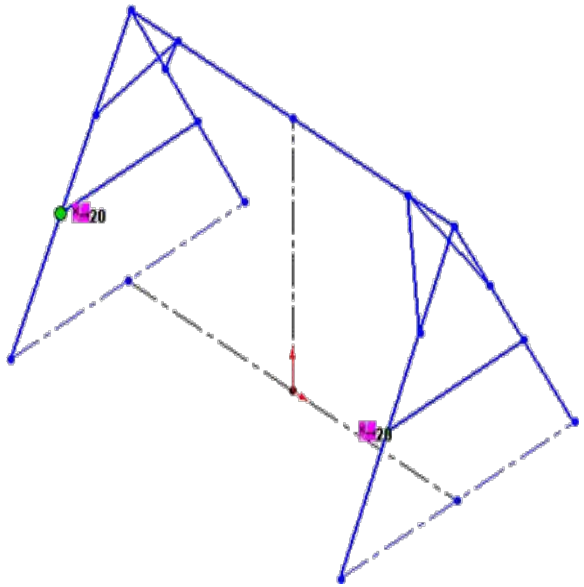
- Complete the left-to-right symmetry of the struts with two more relations.
- Make the two front struts (one on the left and one on the right) EQUAL to each other.
- Also, make the endpoints where the struts meet the A-frame ALONG X.
- Thus the struts are now the same length, *and* they are in the same positions on the A-frames.





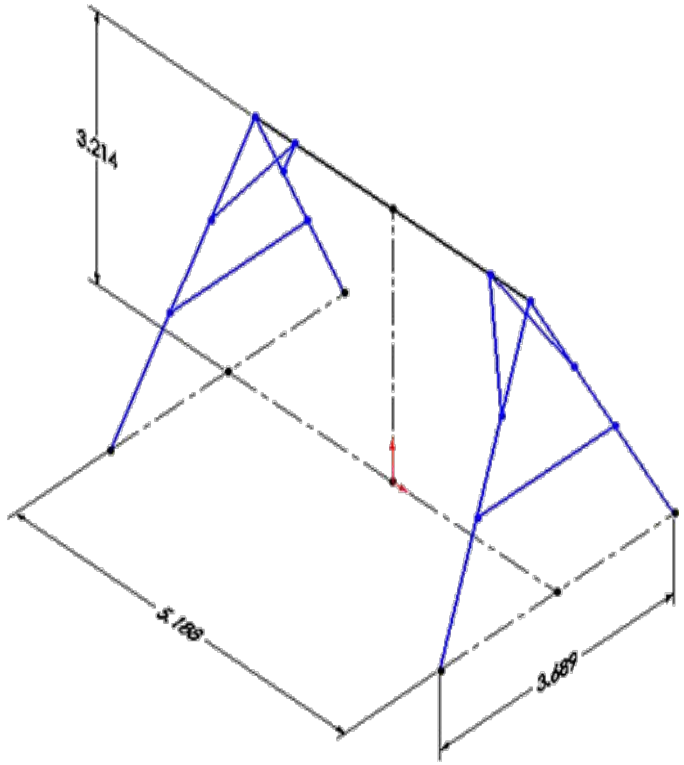
# Swing Frame – Step 13

- Complete the design intent for the shape using one more ALONG X relationship to keep the two crossbars at the same height.
- Now the challenge is to apply parametric dimesions to control the size of the frame as desired.
- Drag it around some more to test.



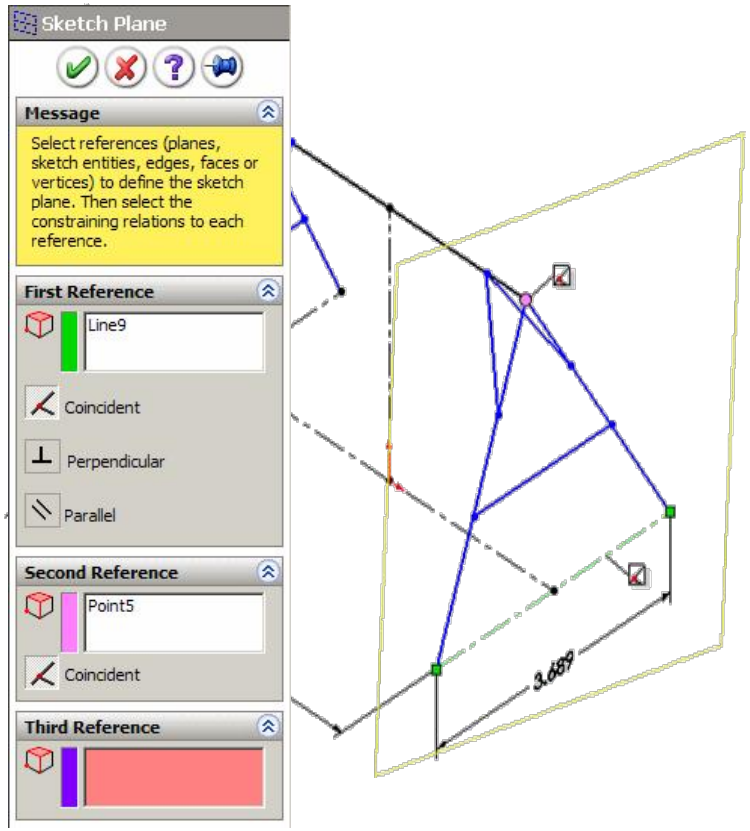
# Swing Frame – Step 14

- Add dimensions for the overall height, length, and width of the swing frame.
- Do not input any new values yet. Just take the default numbers that appear.
- [Optional: Open the file *"3 swing step14.sldprt"*]



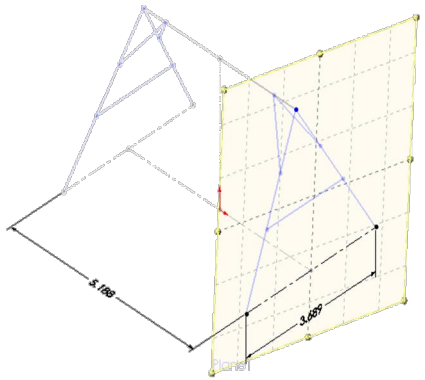
# Swing Frame – Step 15

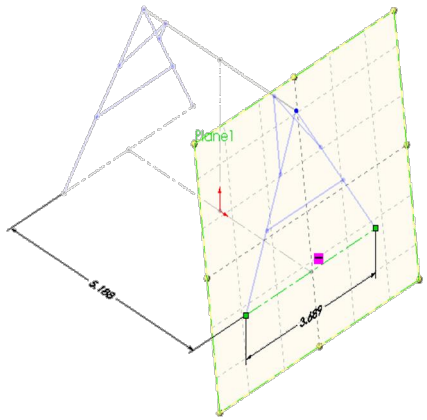
- The A-frame design requires an inward tilt of 5 degrees. This is difficult to capture with a dimension since the lines are at compound angles.
- We will solve this conundrum with a 3D Sketch Plane.
- Use Tools > Sketch Entities > Plane.
- Select the bottom construction line and the top corner of the A-frame.



# Swing Frame – Step 16

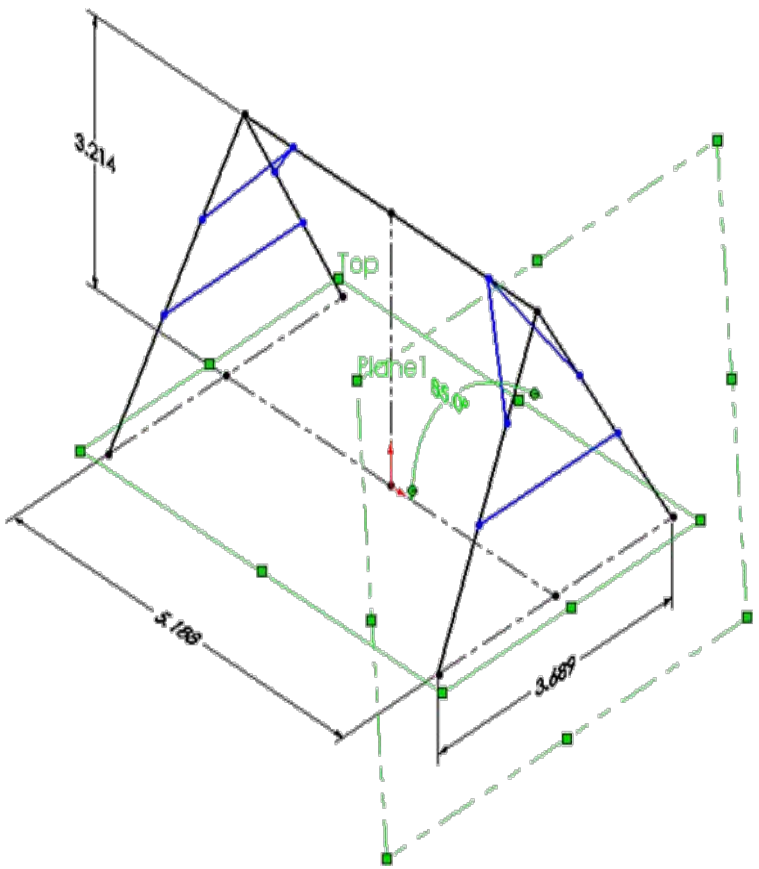
- You are now working in 2D, in 3D!
- To better align this new plane, we can add a relationship.
- Select the construction line and make it HORIZONTAL. This relationship is added with respect to the current 3D Sketch Plane.
- Double-click in space to get out of the 2D mode and get back to regular 3D sketching.



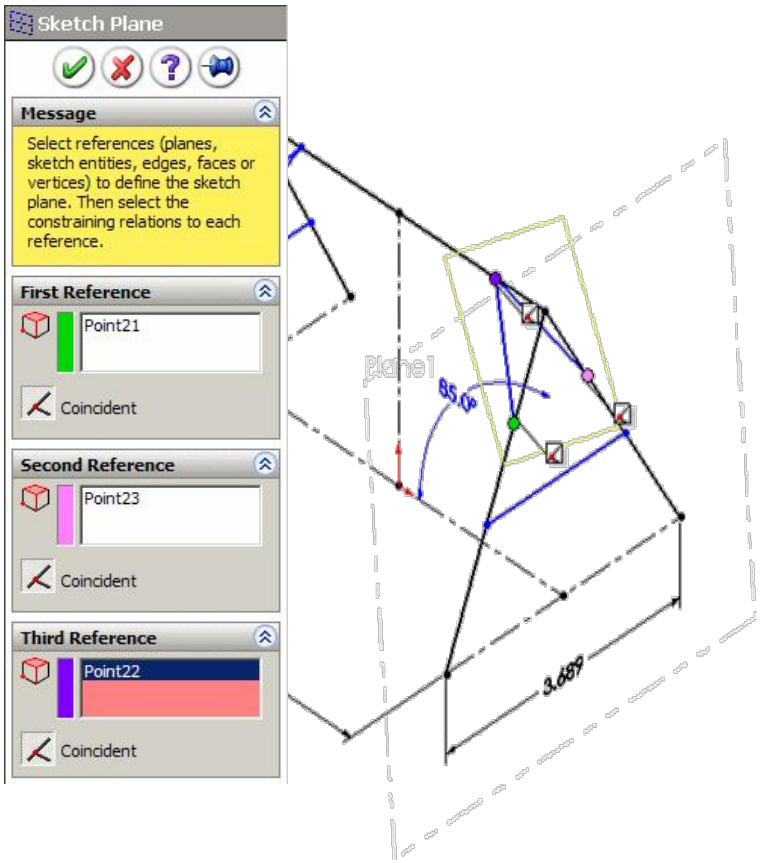
# Swing Frame – Step 17

- With the 3D Sketch Plane now attached to the A-frame, we can add the dimension we want.
- Create an angular dimension of 85 degrees between the Top Plane and the 3D Sketch Plane.



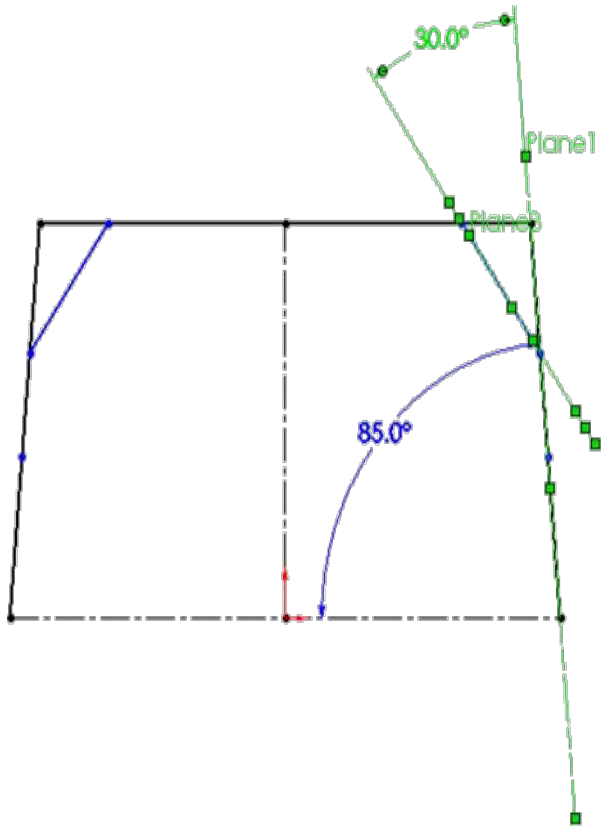
# Swing Frame – Step 18

- Repeat the process to create another 3D Sketch Plane
- Using the three endpoints of the two struts.



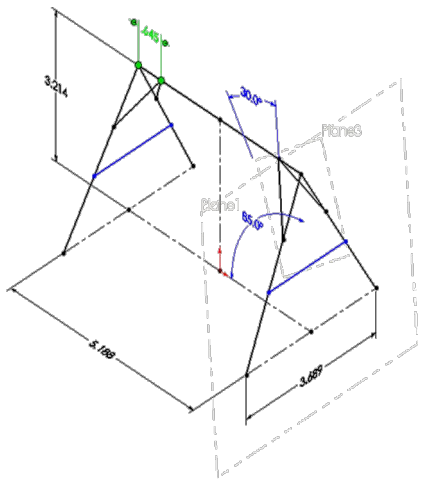
# Swing Frame – Step 19

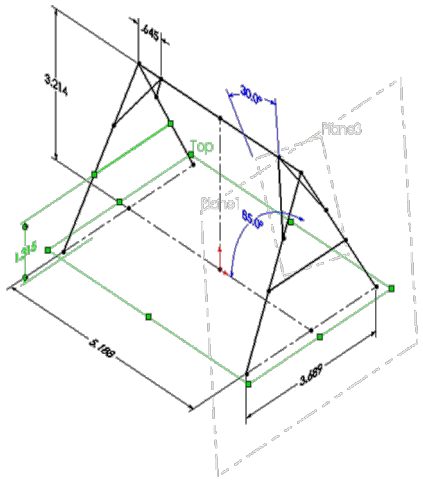
- Add a dimension between the two 3D Sketch Planes.
- Constrain the struts to be at a 30 degree angle to the A-frame.



# Swing Frame – Step 20

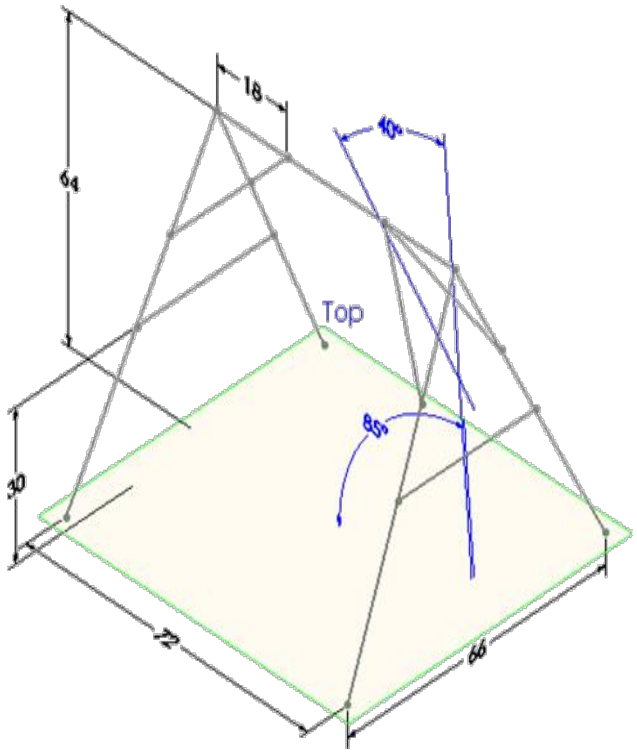
- The last few dimensions will fully define the sketch:
- Dimension the distance from the top corner of the struts to the top corner of the A-frame.
- Dimension the distance from the floor (Top Plane) to the crossbar.
- Now a trick to get it the correct size...



# Swing Frame – Step 21

- If you try to change the dimension values one at a time, the sketch re-solves automatically.
- Sometimes this is not good.
- EXIT the sketch.
- Change all the dimensions by double-clicking them.
- When they’re all changed...
- ...REBUILD.



# Swing Frame – Done

- Now it’s WELDMENT time!
- That can be your homework. ☺

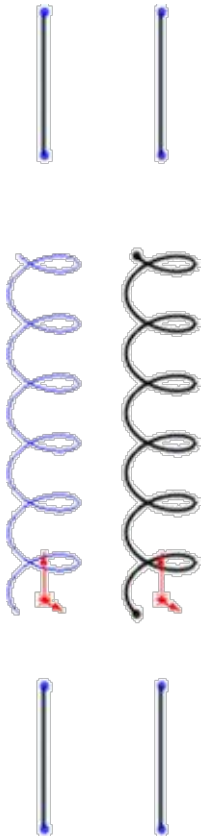
# Example 4: Filament

- Using a 3D Sketch and Convert Entities instead of the old Composite Curve
- Fit Spline as an easy way to create splines that curve in 3D.

# Filament – Step 1

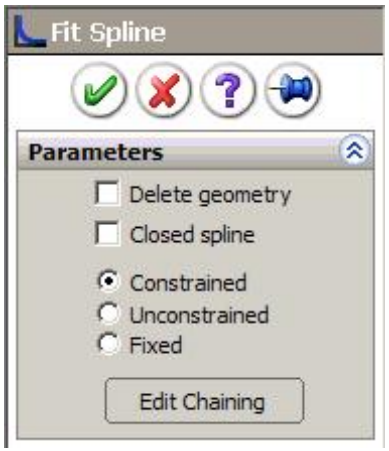
- Open "*4 filament begin.sldprt*".
- Select the helix (it might not highlight) and use Convert Entities.

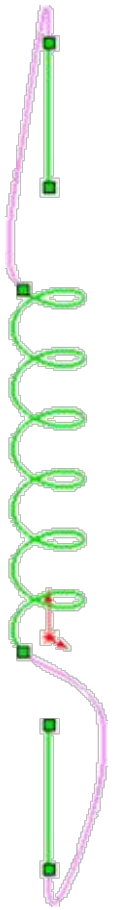

- This creates an associative copy of any edge or curve.
- It is now much easier to design other sketch objects to this curve with tangency, curvature, fillets, trim/extend, etc.



# Filament – Step 2

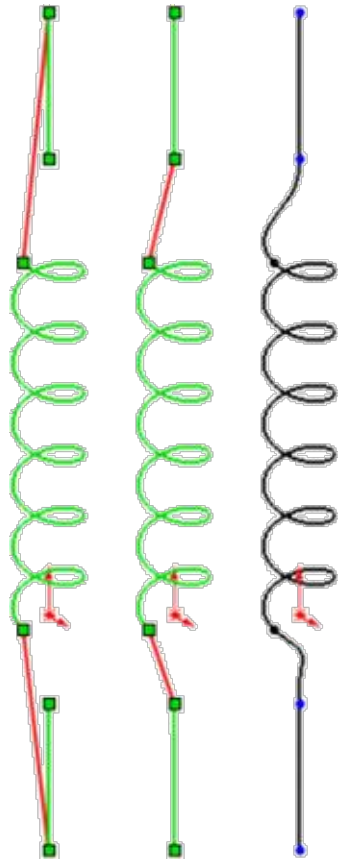
- To create the transition from helix to straight, use a Fit Spline.
- Tools > Spline Tools > Fit Spline
- DISABLE the option to create a Closed spline.
- Select one of the lines, the helix, and then the other line.
- But... the Fit Spline doesn't connect the entities the way we want it to... yet...



# Filament – Step 3

- Click the “Edit Chaining” button.
- In order, click two endpoint that you want connected (end of one line to the end of the helix).
- Click two more endpoints you want connected (the other end of the helix to the end of the other line).
- Create the fit spline.

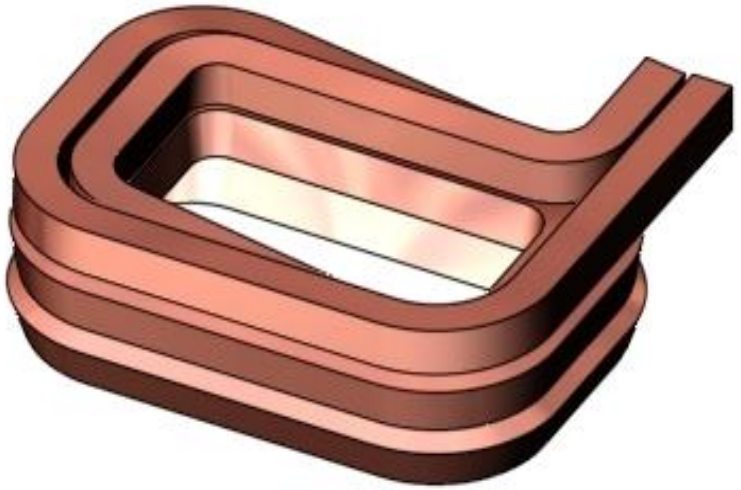


# Filament – Done

- Use the 3D Sketch as a sweep path.
- This technique is also good for wires, cooling lines, heat exchangers, and other tubing with coils in them.

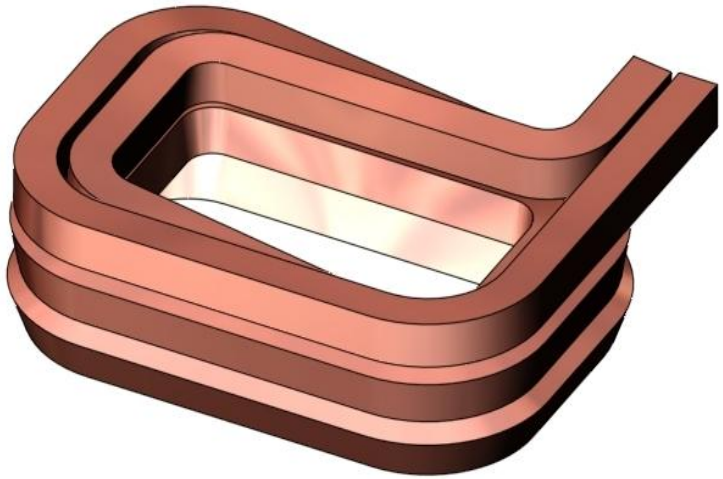
# Example 5: Square Coil

- Trying to use a Sweep does not give the desired result.
- There is a sweep option that solves this issue (sometimes) but that's a different talk for a different day...



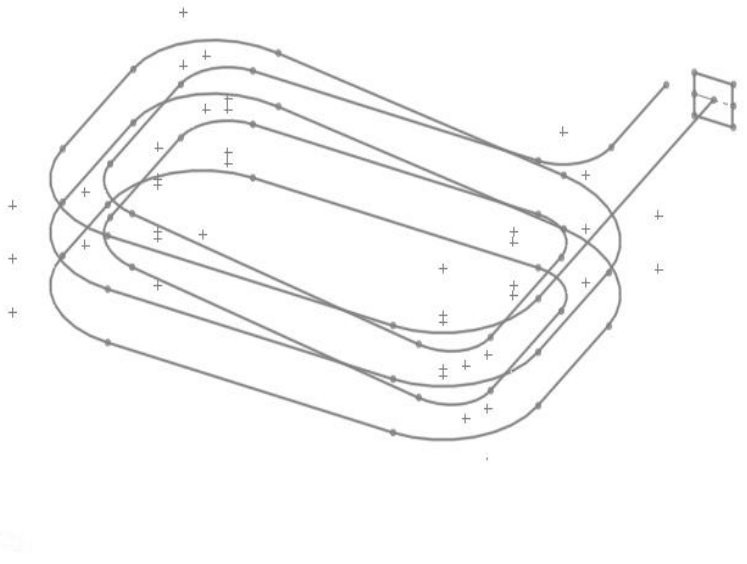
# Square Coil – Step 1

- Open the file "*5 coil begin.sldprt*".



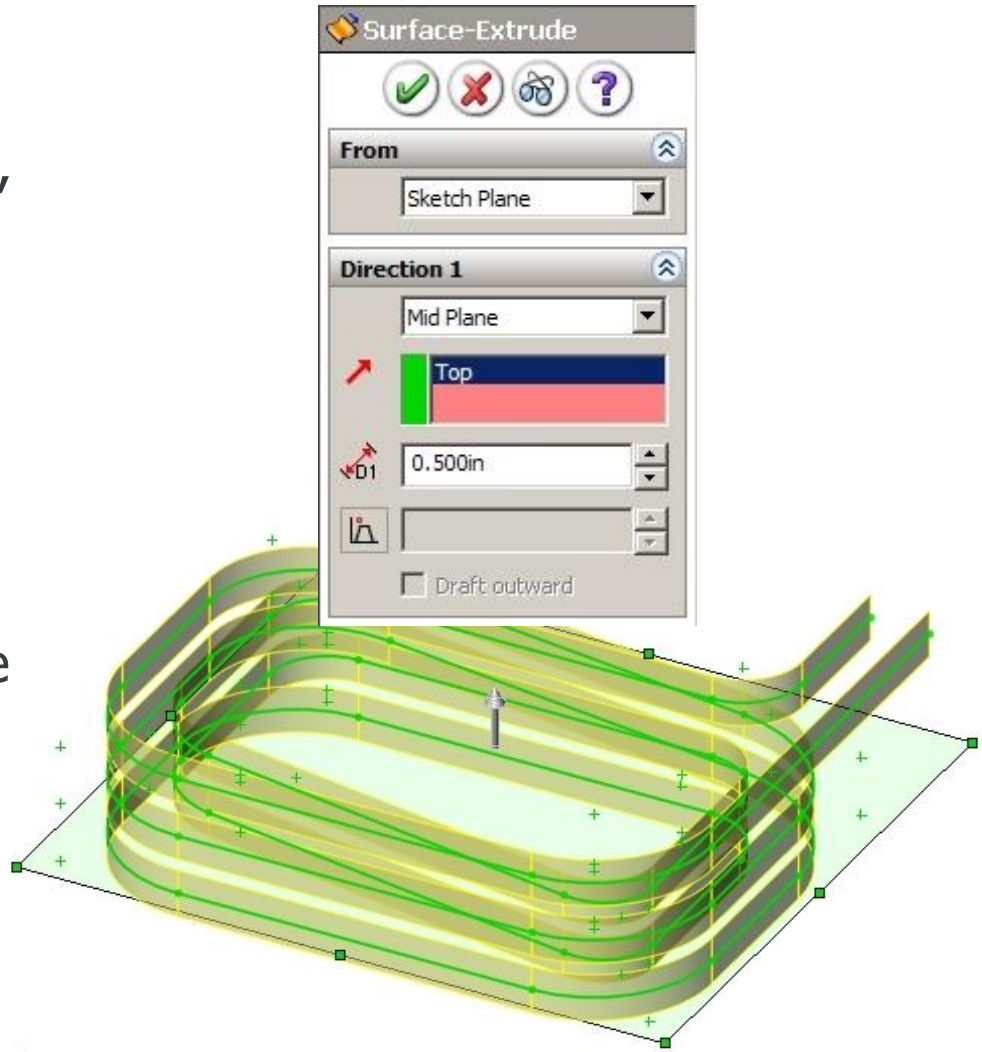
# Square Coil – Step 2

- Delete the Sweep feature.
- Notice the 3D Sketch used as the sweep path.
- This used a combination of Convert Entities + drawing lines + Sketch Fillet.
- The goal is a constrant cross section 0.5 inches square using the 3D Sketch as the centerline.



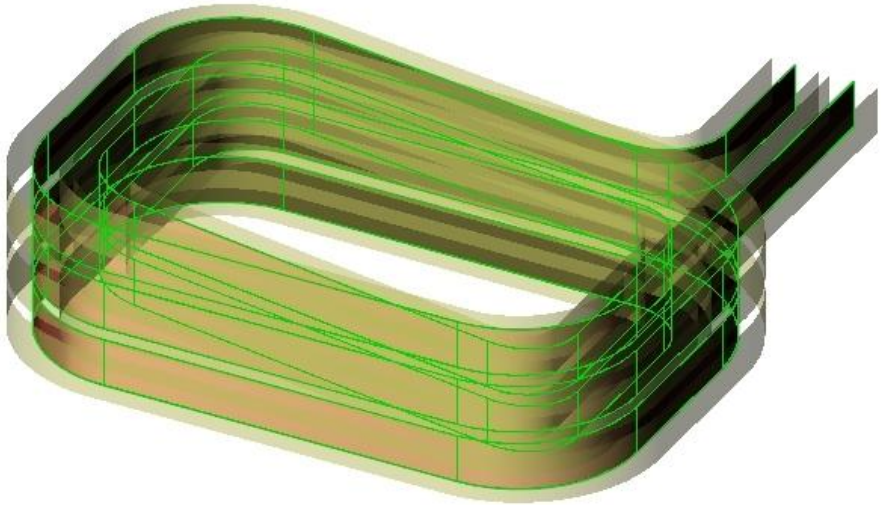
# Square Coil – Step 3

- You can Extrude a 3D Sketch!
- If you have closed regions, you can create a solid.
- Or you can Extrude a Surface.
- Insert > Surface > Extrude.
- Select the 3D Sketch.
- Select the Top Plane of the part to define the vector direction for the extrude.
- Extrude Midplane 0.5 inches.



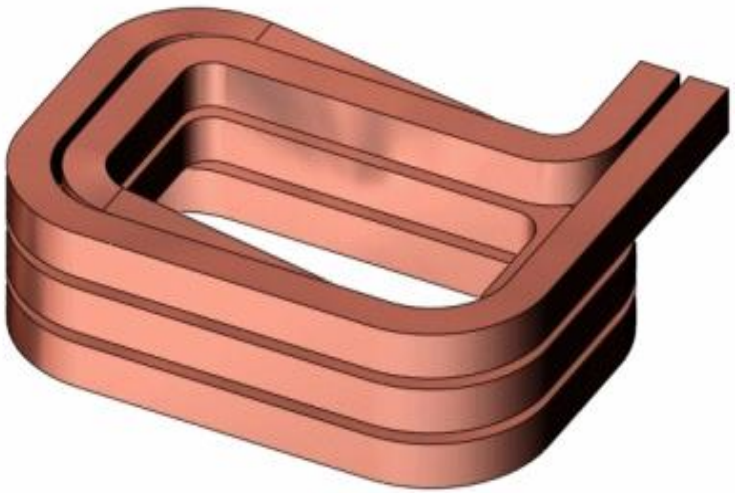
# Square Coil – Step 4

- To create the solid, we will thicken the surface.
- Insert > Boss > Thicken
- Select the surface body.
- Thicken to both sides, 0.25 inches.

# Square Coil – Done

- A nice analytical shape is produced.
- There could be other interesting uses for extruding a 3D Sketch (or a portion of it). This is one example.

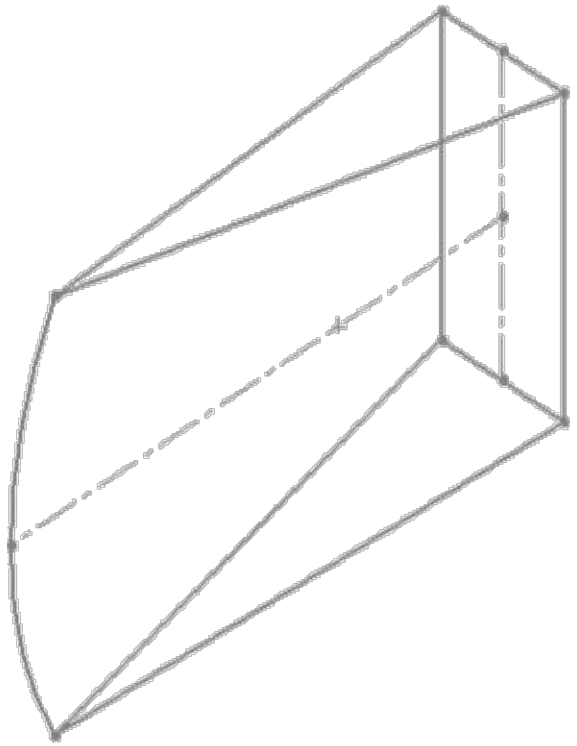


# Example 6: Axe head

- Using 3D Sketches to define Loft profiles and guide curves.
- Use of the Selection Manager.

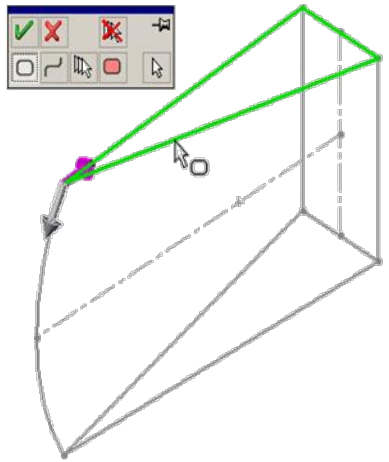
# Axe head – Step 1

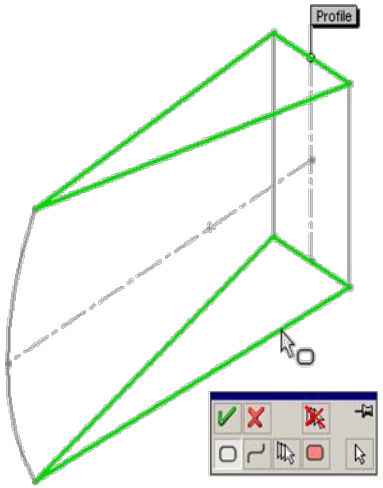
- Open the file "6 *axe begin.sldprt*".
- This is one 3D Sketch!
- Putting all these entities into one sketch makes the definition of this shape simpler to achieve.
- If you try to Loft from the rectangle to the arc, it will not work. SolidWorks cannot do it.
- Instead, we will Loft from top to bottom.
- The SelectionManager helps us try both methods quickly.



# Axe head – Step 2

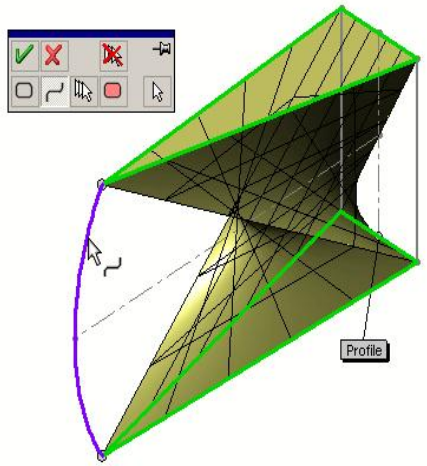
- Select one of the edges of the triangular top.
- The SelectionManager appears.
- Toggle the icon for a closed loop and confirm.
- Repeat for the bottom triangle.

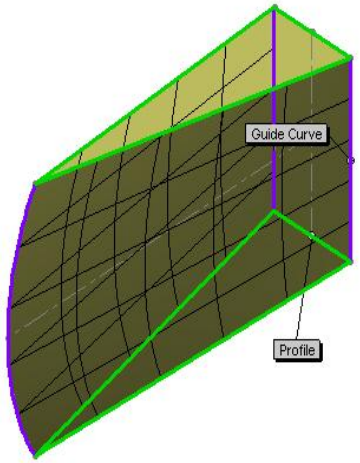




# Axe head – Step

- Now select the arc.
- Again the SelectionManager appears.
- Accept the option for the open loop.
- Repeat for the other two edges of the rectangle.



# Axe head – Done

- This is a very simple example of the powerful ability to define many profiles and guide curves in a single 3D Sketch.
- This would have required 7 features to set up in earlier releases.
- You can also make use of 3D Sketch Planes with this method.

# Example 7: Fly-through

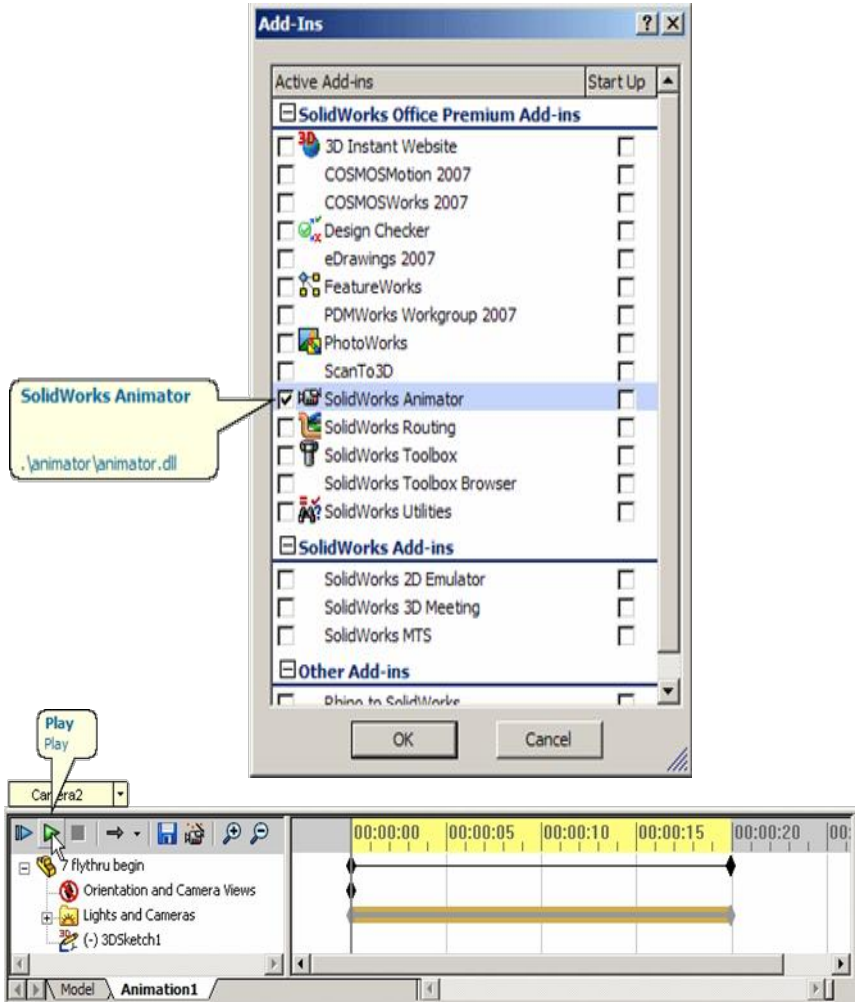
- Using the Triad to move entities in 3D space.
- Working with 3-dimensional spline curves.
- Animating a camera to follow a path in 3D.

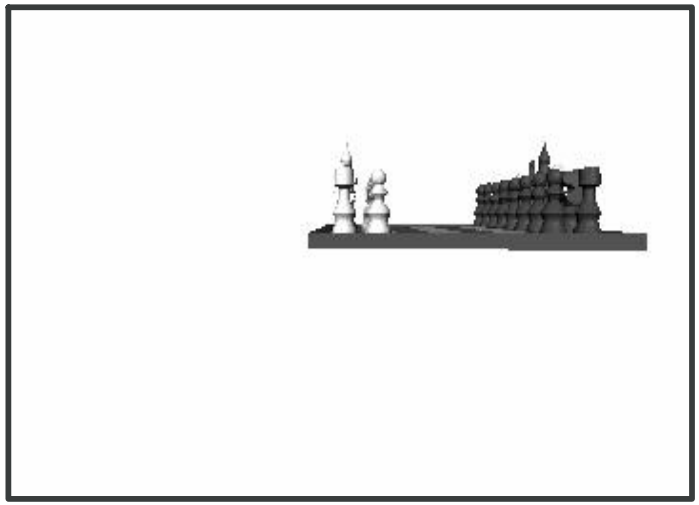
# Fly-through – Step 1

- Open the file:
  *"7 flythru begin.sldprt"*.
- This assembly contains a 3D Sketch with a 3D spline used as the path for a camera.

# Fly-through – Step 2

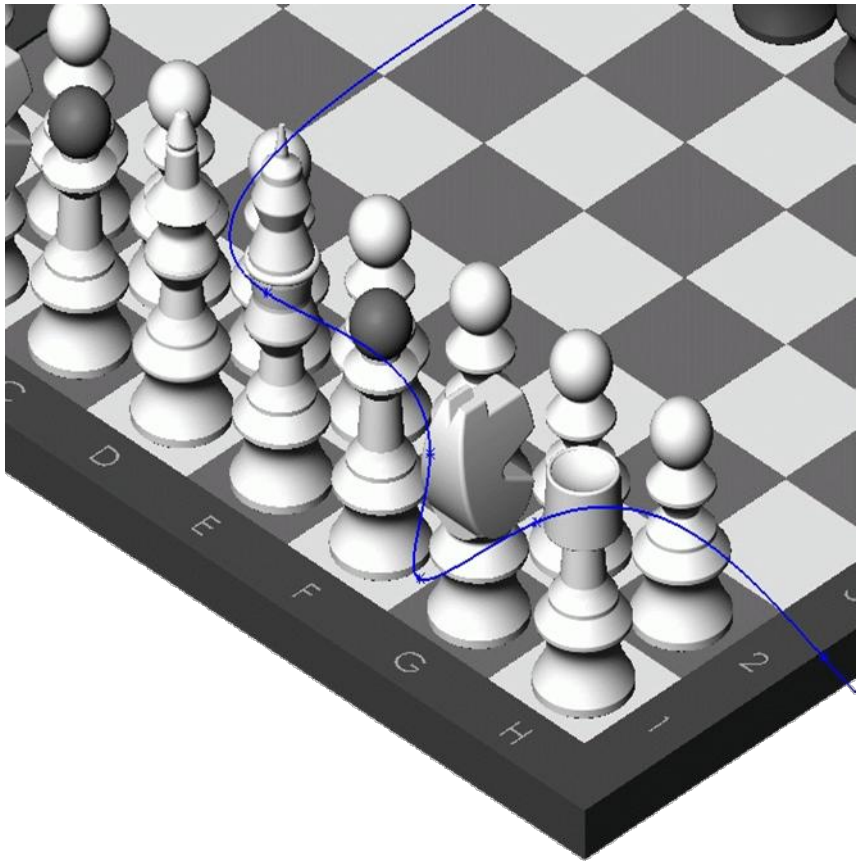
- Turn on Tools > Add-ins > SolidWorks Animator.
- Click the *Animation1* tab at the bottom of the window.
- Press the green *Play* icon.



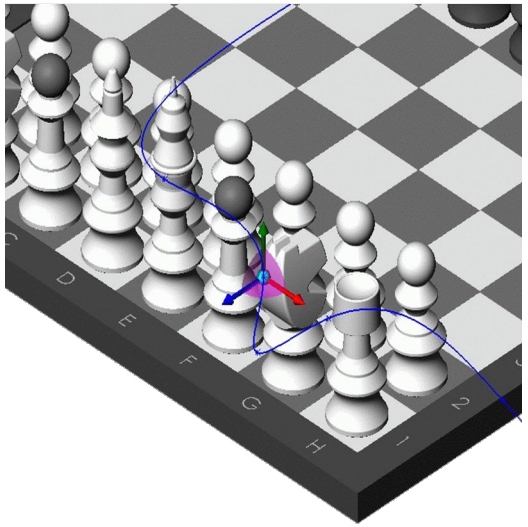
# Fly-through – Step 3

- Click on the *Model* tab to exit the Animator interface.
- Find the “3DSketch1” at the bottom of theFeatureManager.
- Right-click and Edit Sketch.
- Zoom in on the area shown.
- We will alter the flight path of the camera.



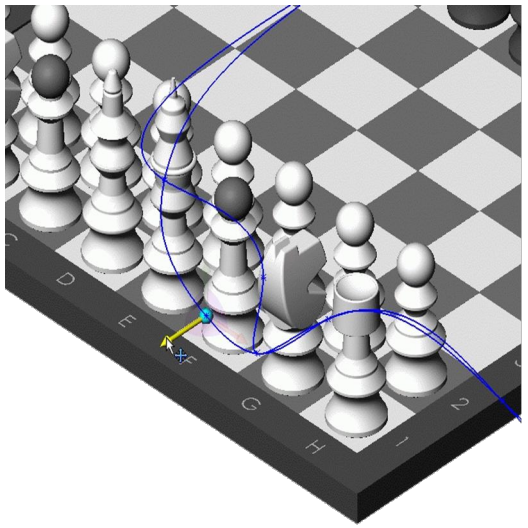
# Fly-through – Step 4

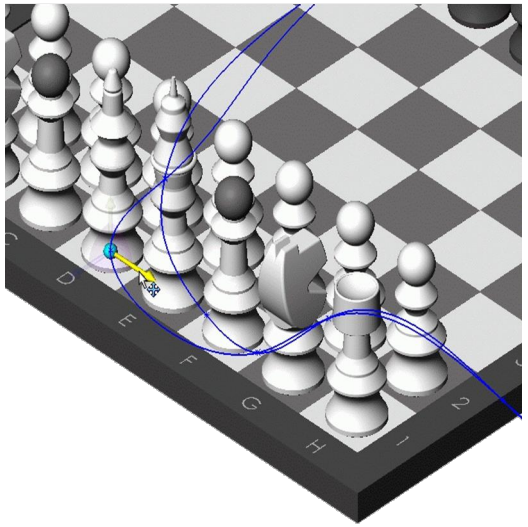
- The Triad is another way to manipulate objects in 3D.
- Right-click on a spline point and choose Show Sketcher Triad.
- The Triad then appears, aligned to the spline point.
- You can use the Triad to move the point through 3D space.

# Fly-through – Step 4

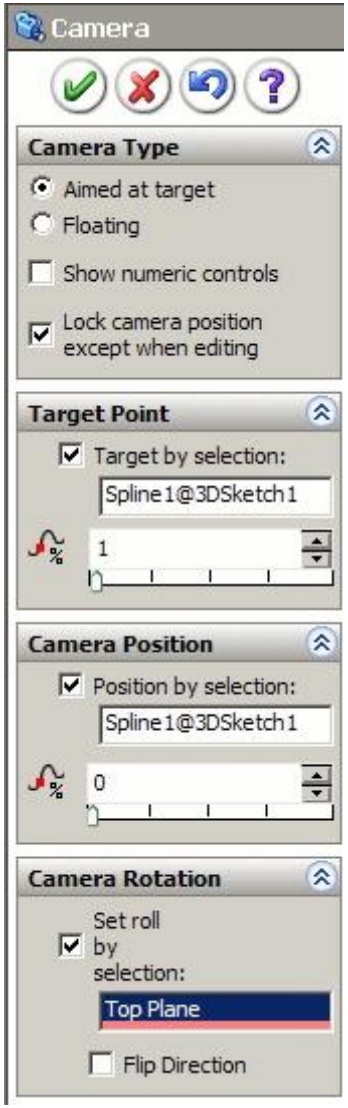
- Drag the blue arrow to move the point in the Z direction.
- Drag the red arrow to move the point in the X direction.
- You could also add or delete spline points as desired.
- Rebuild or Exit the sketch.
- Play the animation again.



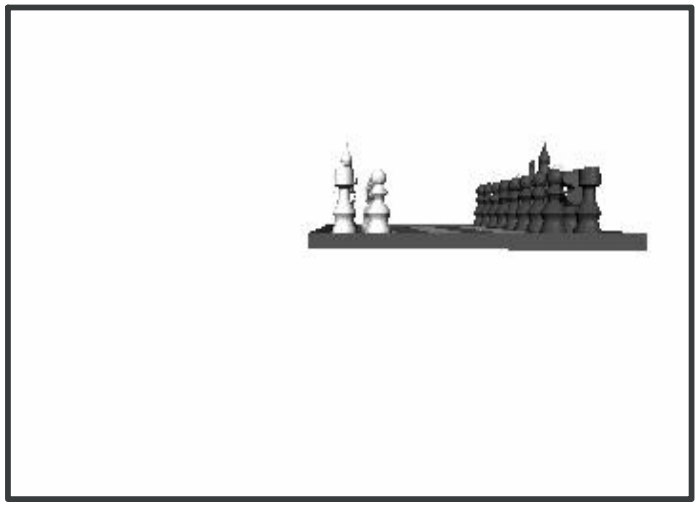


# Fly-through – Step 5

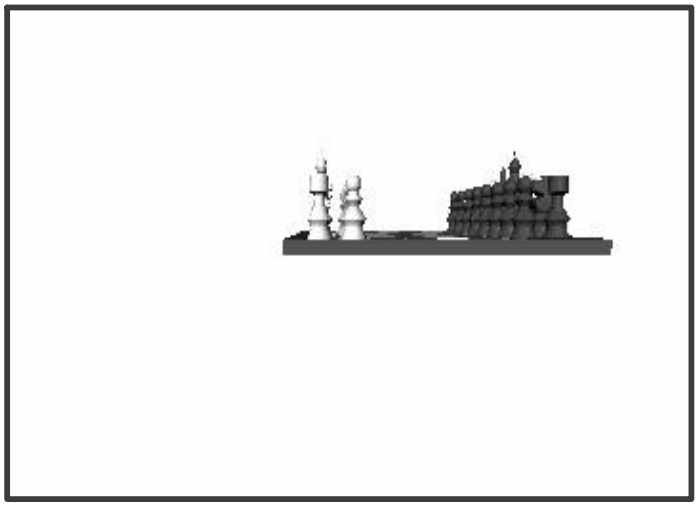
- The animated flight along the spline is achieved by editing the Camera properties.
- The spline is selected as the Target of the camera.
- The spline is also selected for the Position of the camera.
- Animator is used to capture:
  - Starting condition 0% along the spline, targeted 1% ahead
  - Ending 99% along the spline, targeted at 100%
- Rotation control is locked to the Top plane to prevent



# Fly-through – Done

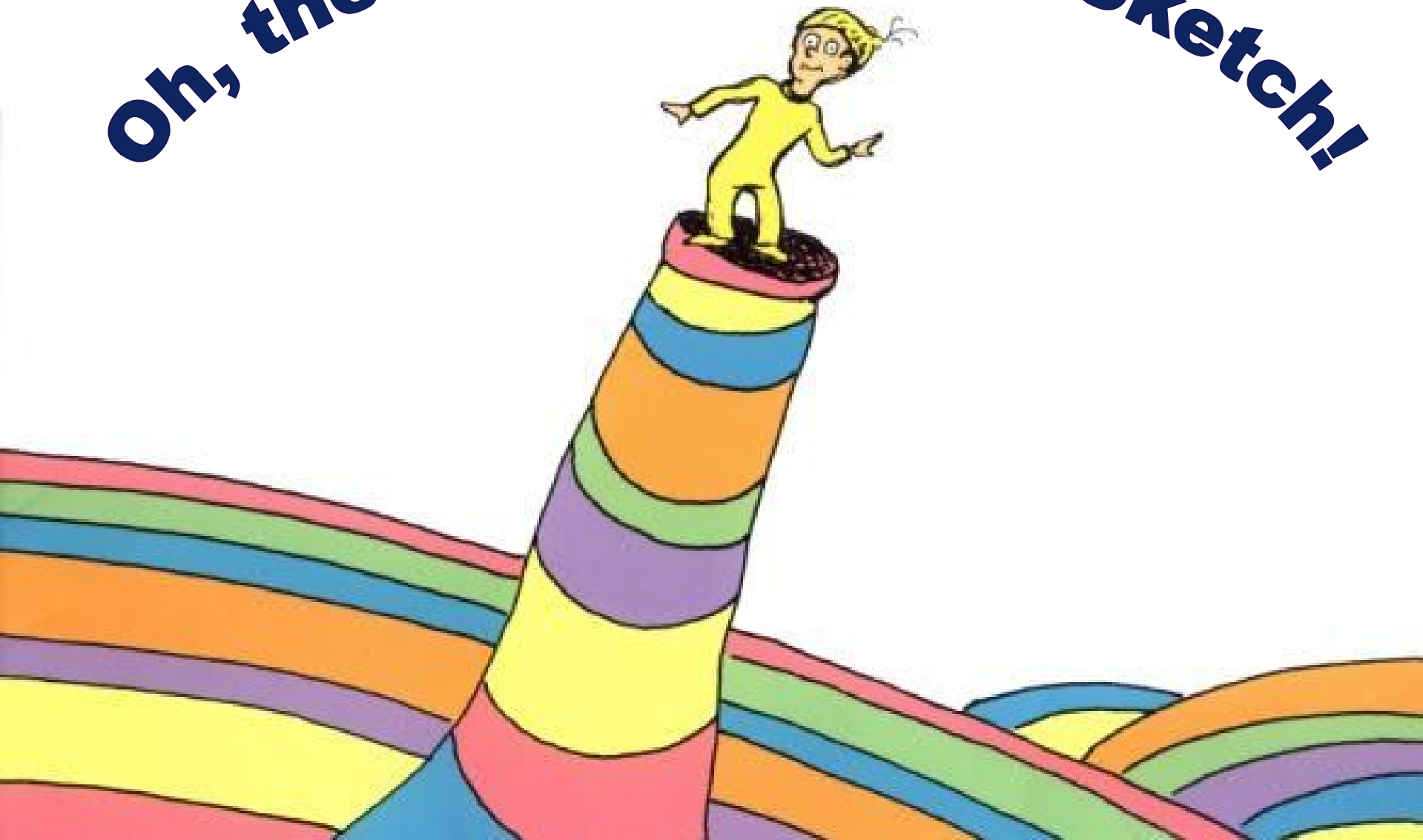


Before



After

Oh, the Sketches you'll Sketch!



Oh the places you'll go! Oh the things you will do!

You can sketch in 3D!

I'm so happy for you!

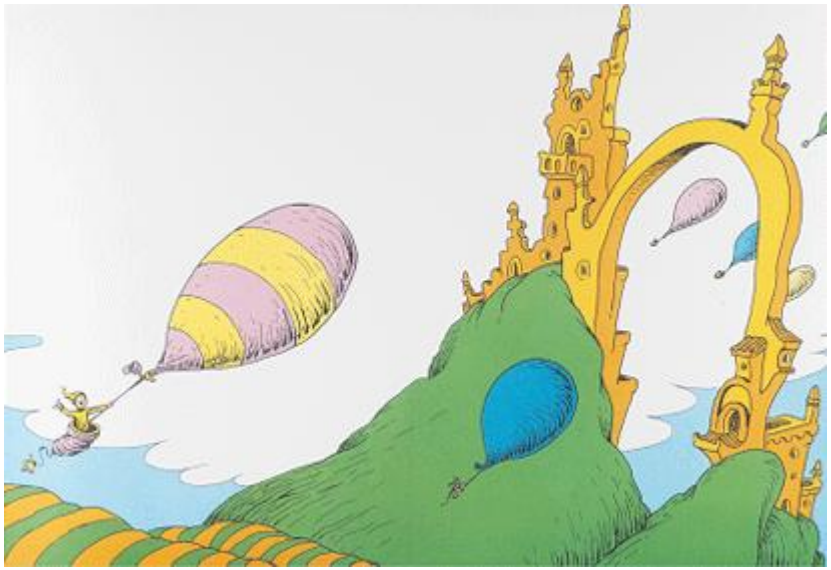
Remember to drag it! See what it can do.

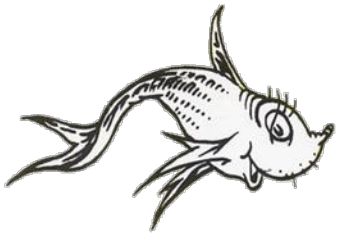
Keep adding constraints 'til it's no longer blue!

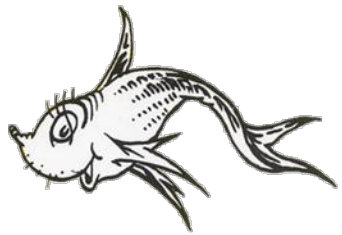
If X,Y is not where you wanted to be,

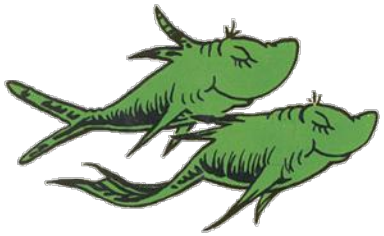
Your TAB key will switch

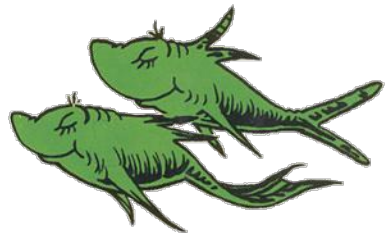
from XY to YZ!





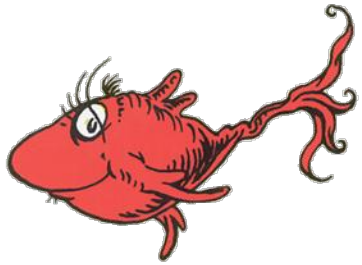


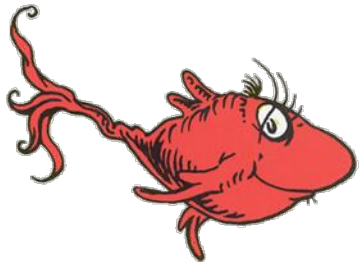




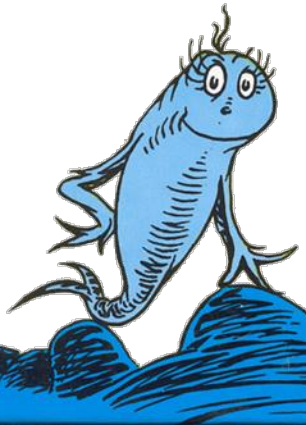
Now you can sketch whatever you like!
You can sketch handlebars for a bike.

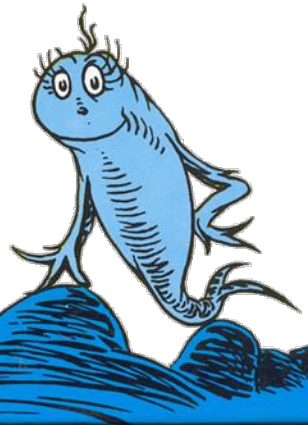
You can sketch a 3D line.
You can sketch a 3D spline.
You can LOFT them to a circle.
The profiles are green,
the guide curves are purple!





Sketch a wire, sketch a pipe,
Sketch a ROUTE of any type!





Sketch a coil, sketch a spring,
Sketch an A-frame for a swing!

Yes, 3DSketch
Is tough at times,

Made EASIER
with help
from DR.SEUSS rhymes.

# Thank you!

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