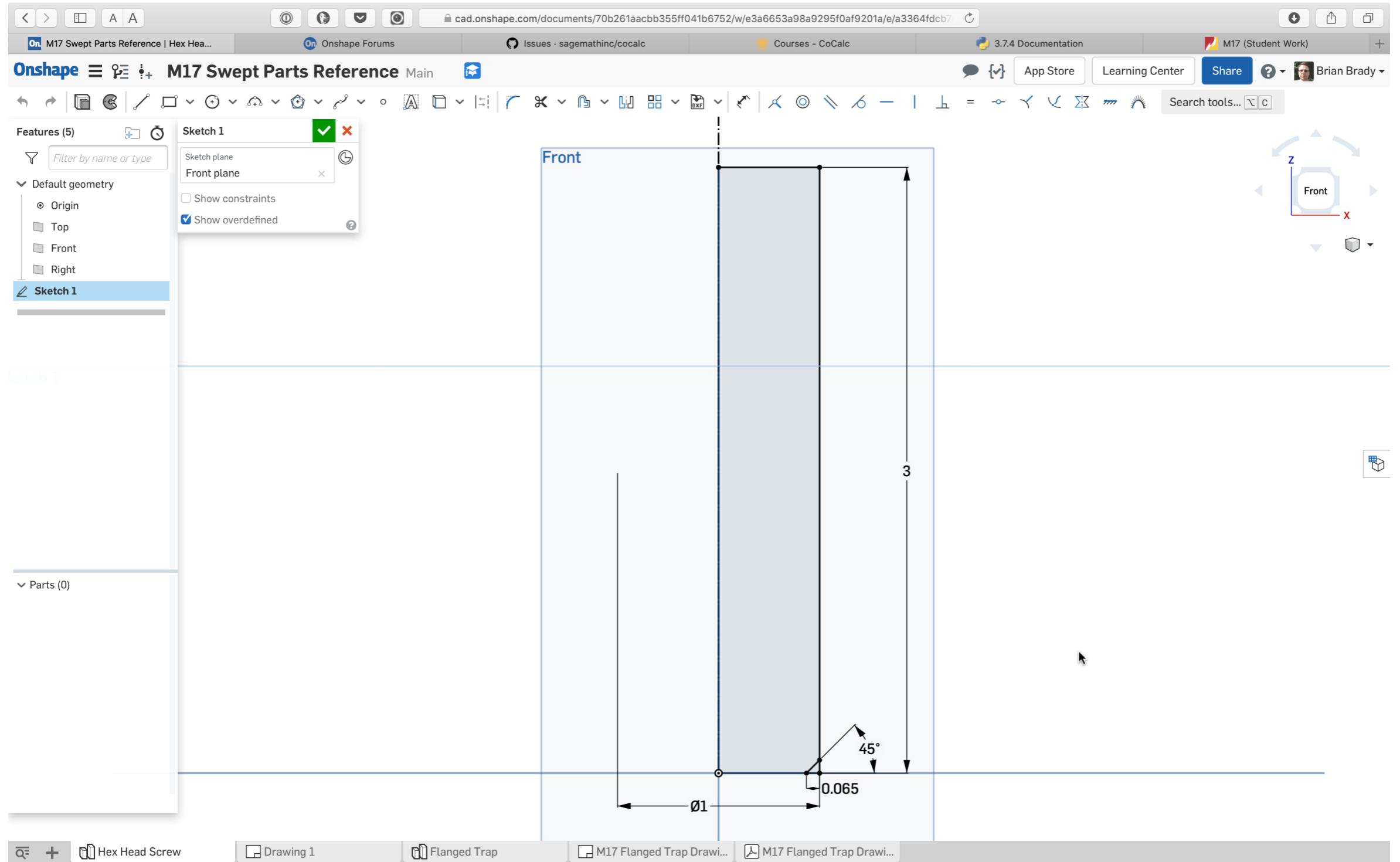


Modeling a Screw or Bolt with Onshape

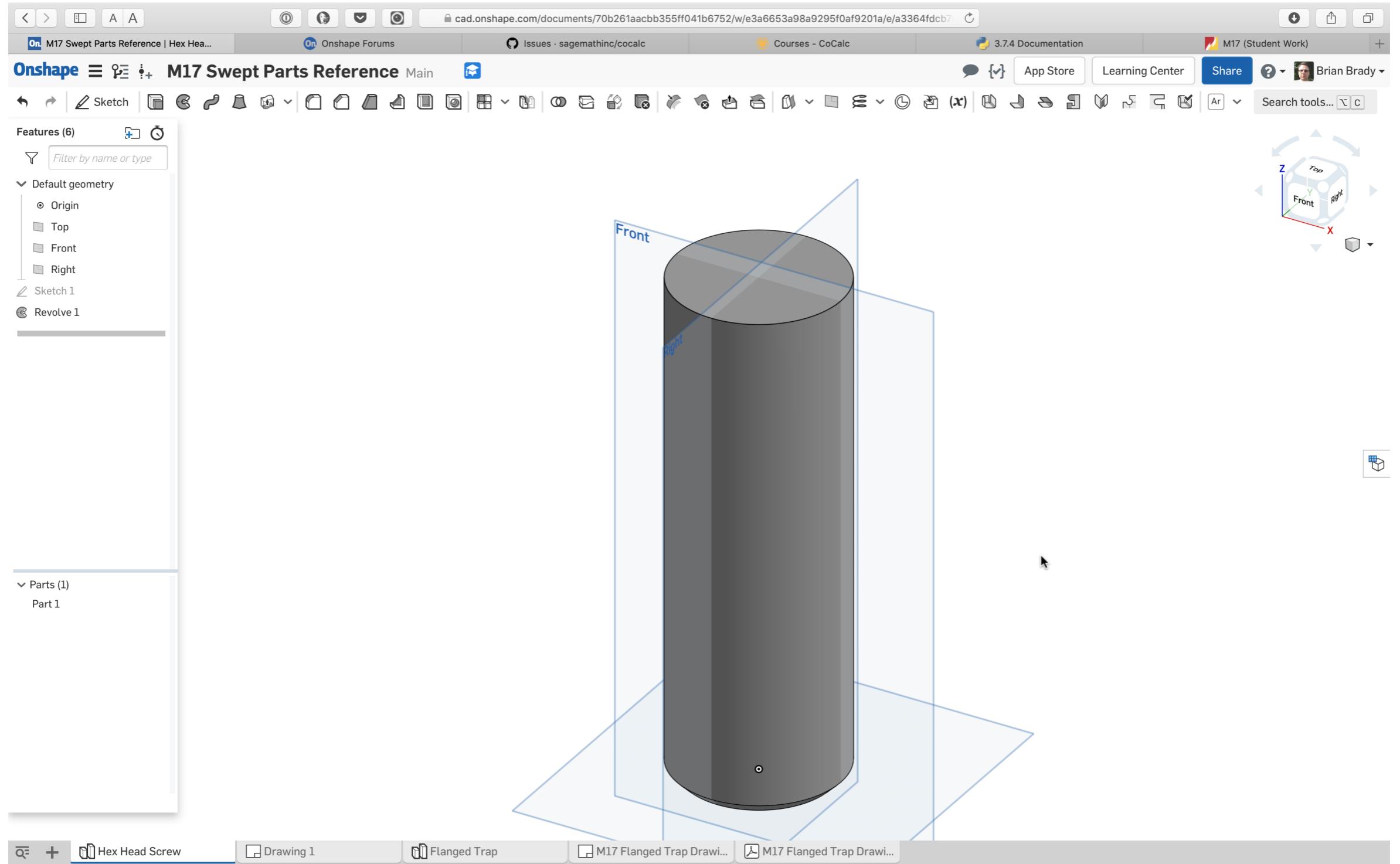
An exercise in using the Helix and Sweep tools

Screw Specs

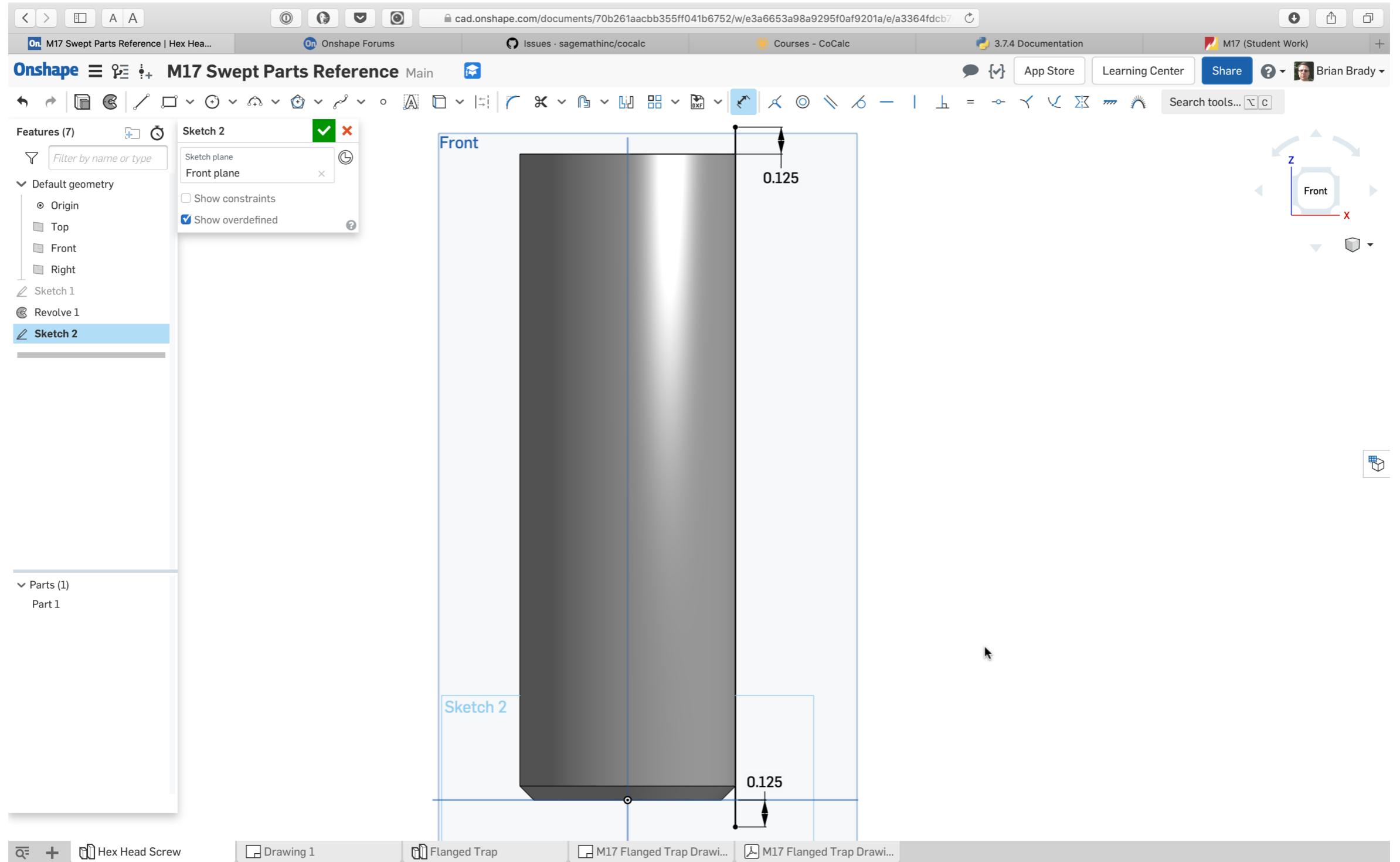
- 1-8 x 4.5 hex head screw
- Thread size is 1-8 UNC, meaning 1 inch diameter and 8 threads per inch (tpi)
- Threaded portion is 3 inches long
- Overall length (below the head) is 4.5 inches
- Hex head is 1.5 inches across the flats and 0.75 thick



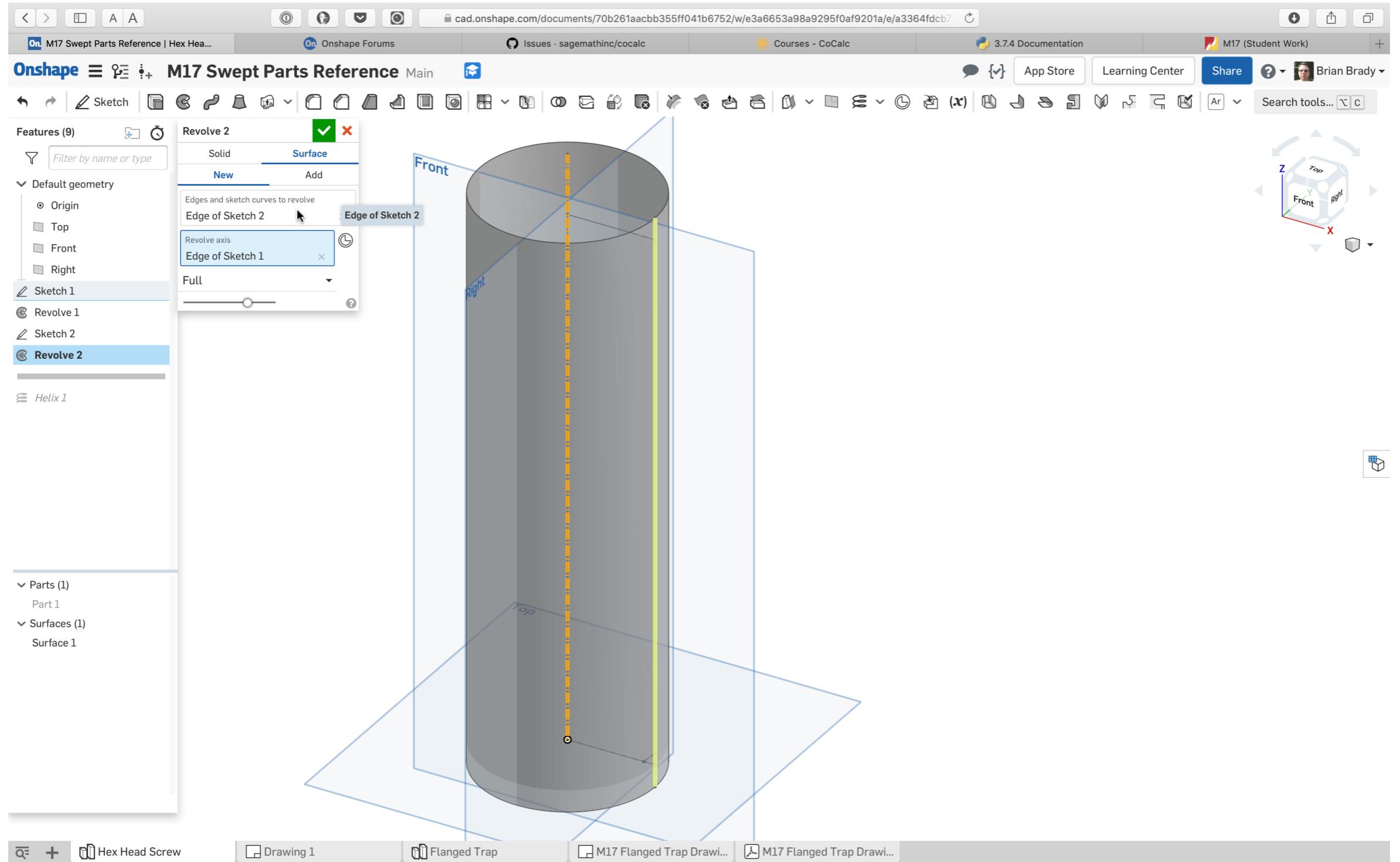
Sketch the shape that will be revolved to make the threaded portion of the screw on the front plane.
Notice the use of diameter dimensioning.



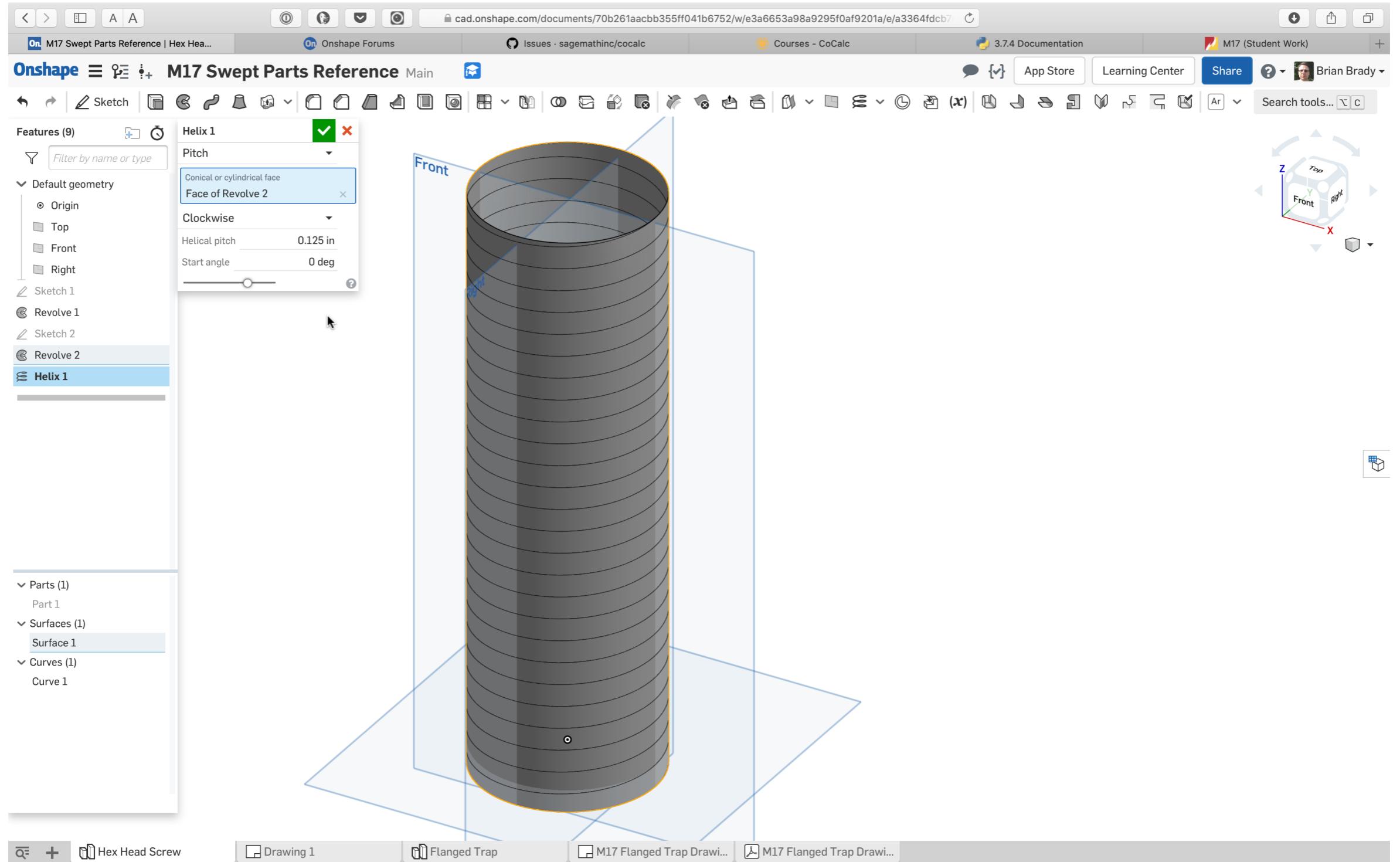
Revolve the initial sketch



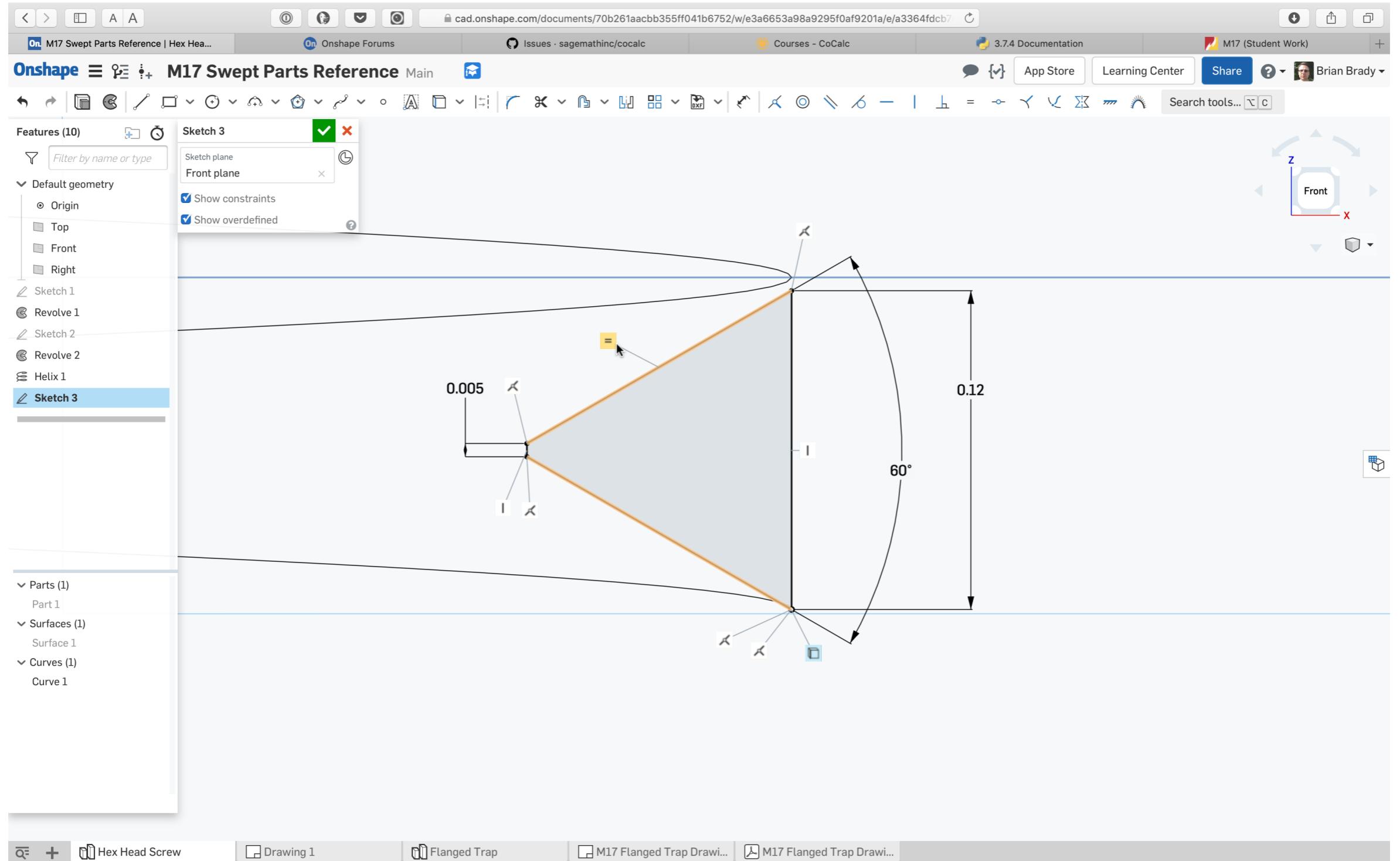
On the front plane sketch a line that extends 0.125 above and below the revolved part. “Use” the right edge of the revolve so the line aligns with the outer limit of the revolve.



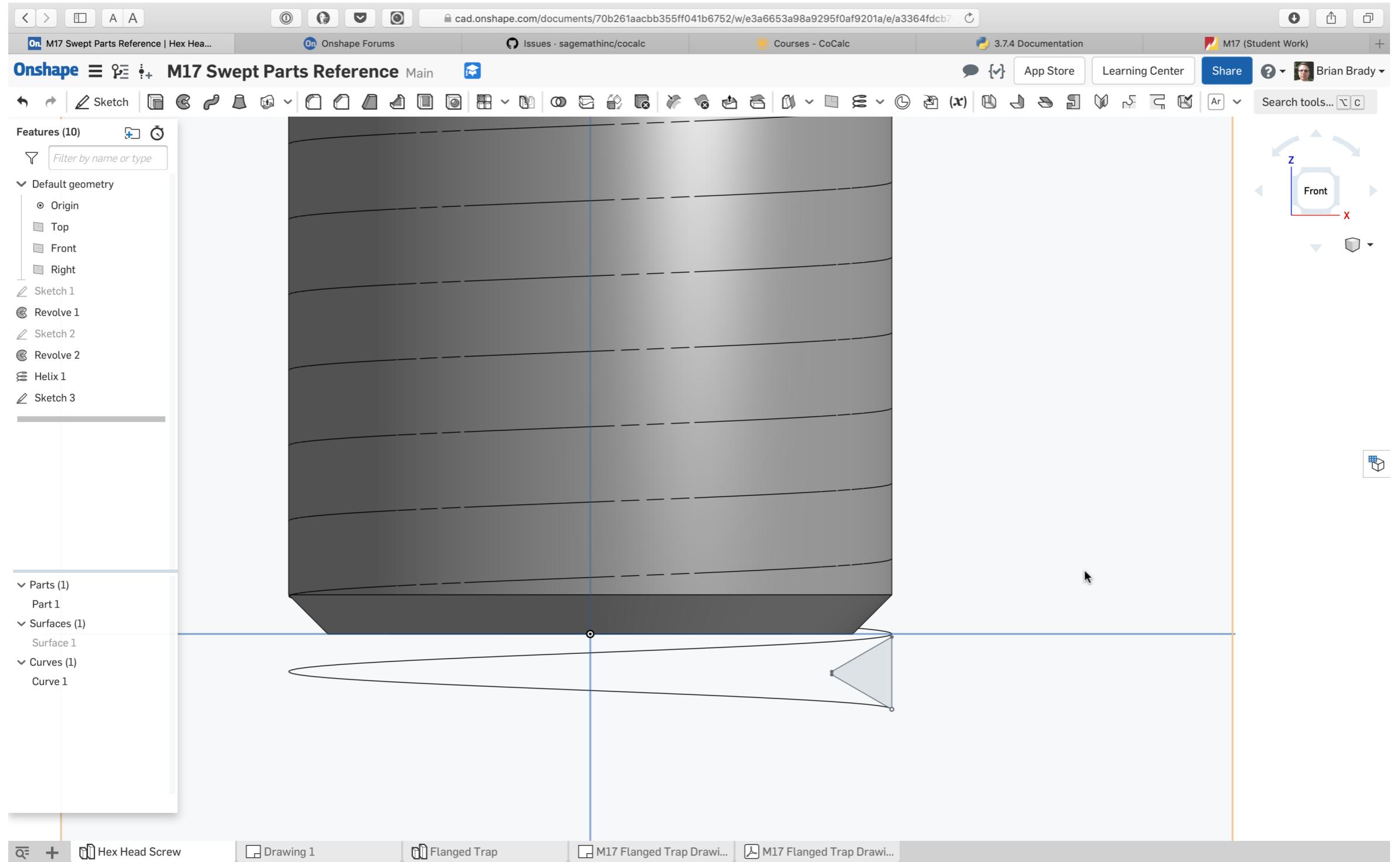
Revolve the line sketch to create a surface (not a solid). Use the centerline from the initial sketch (or the outer surface of the revolved part) as the axis of the revolve.



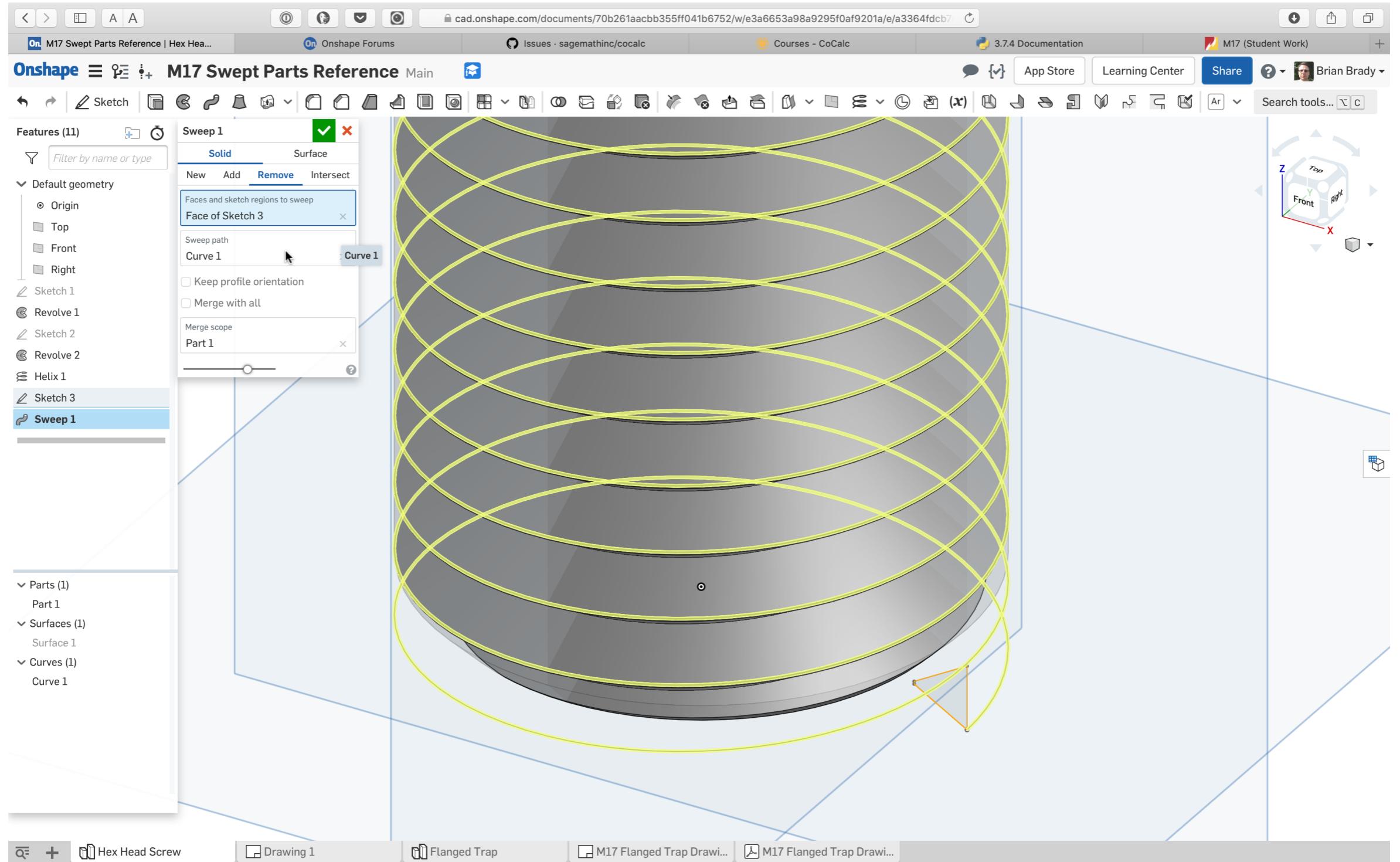
Use the surface to create a helix (notice that the part is hidden see the surface better). The helix has a pitch of 0.125 (1 divided by the threads per inch).



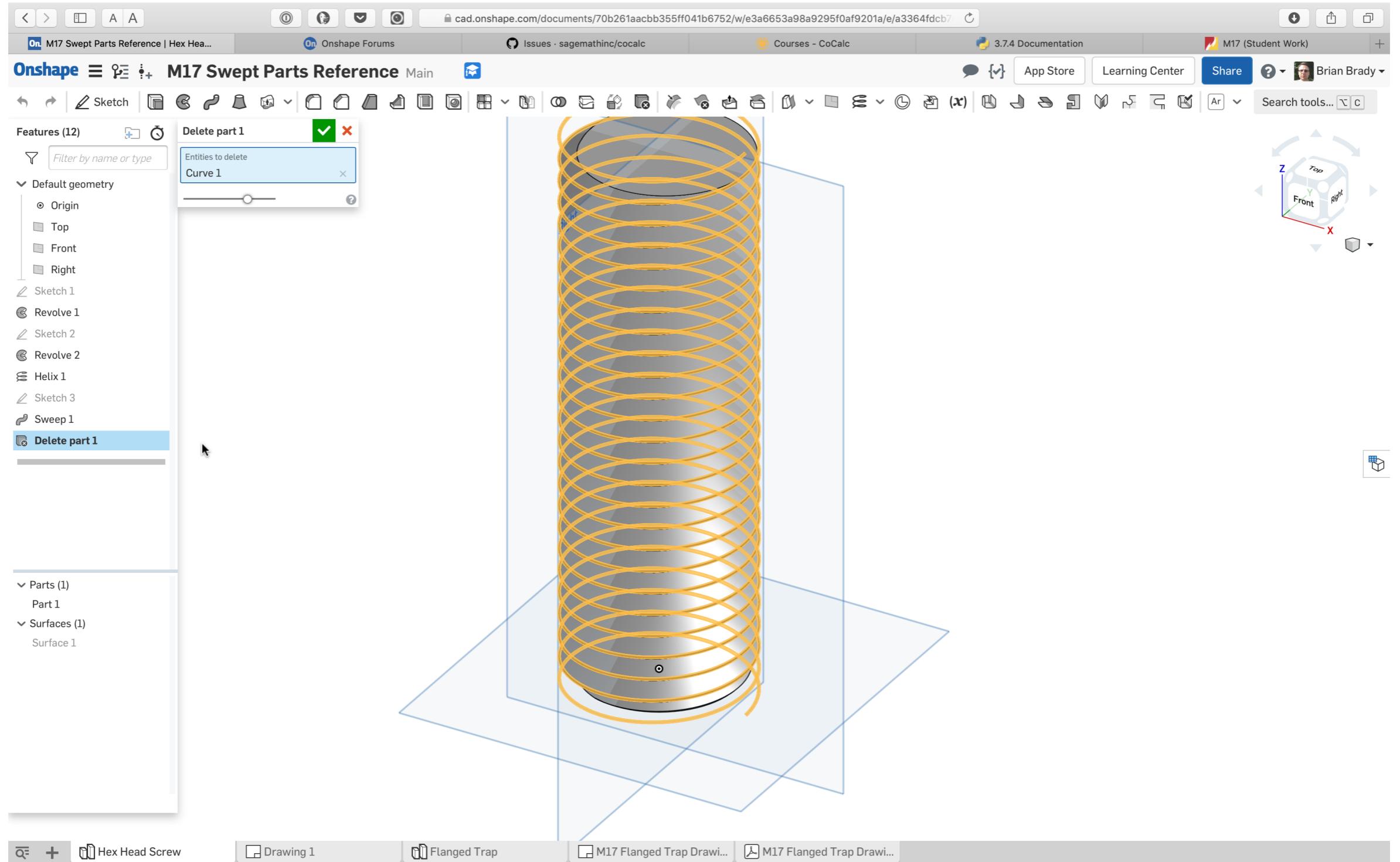
After hiding the surface and solid part, create a sketch on the front plane that will be used to “cut” the threads into the screw shank. “Use” the bottom end point of the helix (lower, right vertex in the sketch). Note the constraints shown in the sketch.



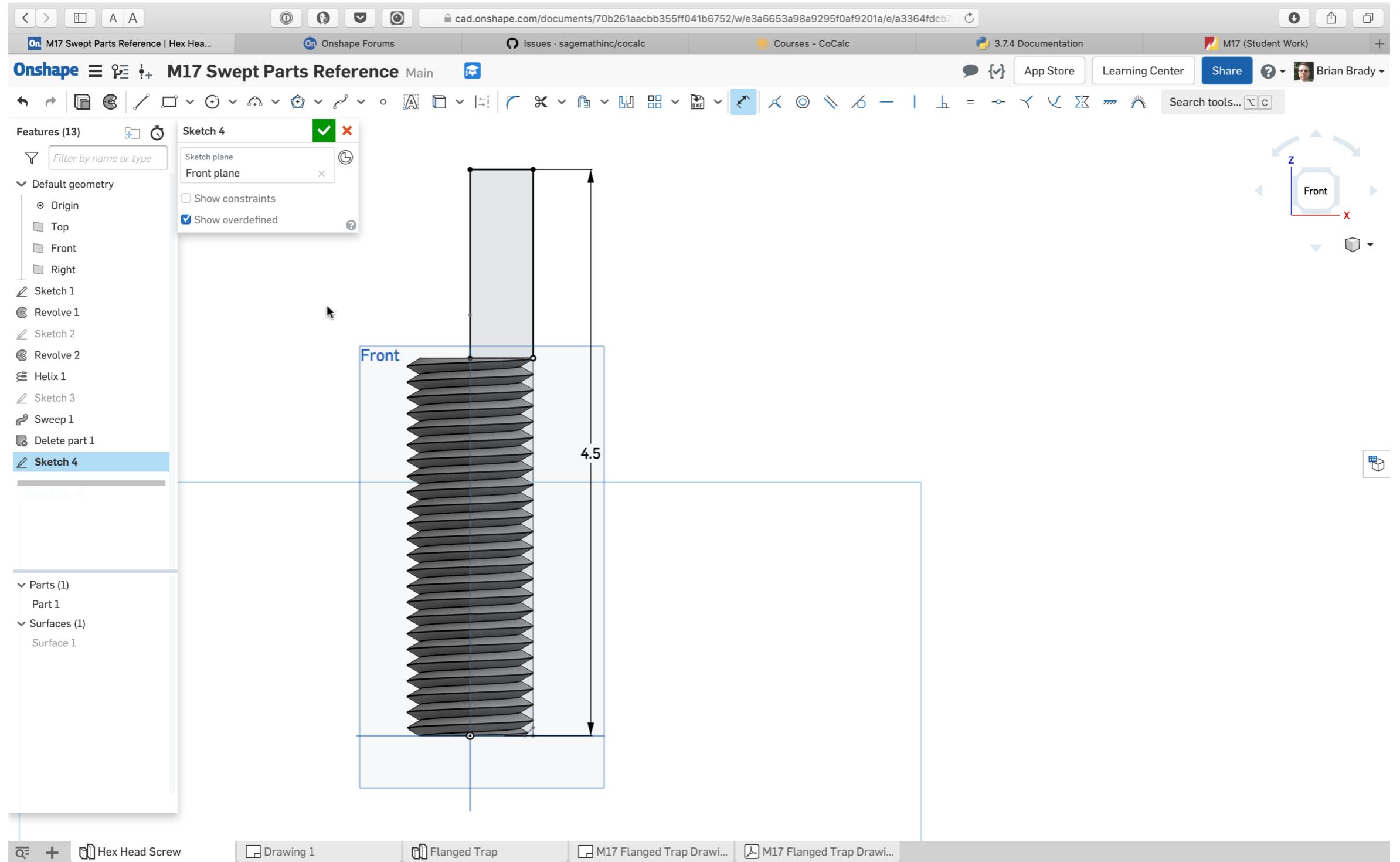
The thread cutting sketch shown with the solid part visible. Notice that it starts completely below the bottom of the solid part.



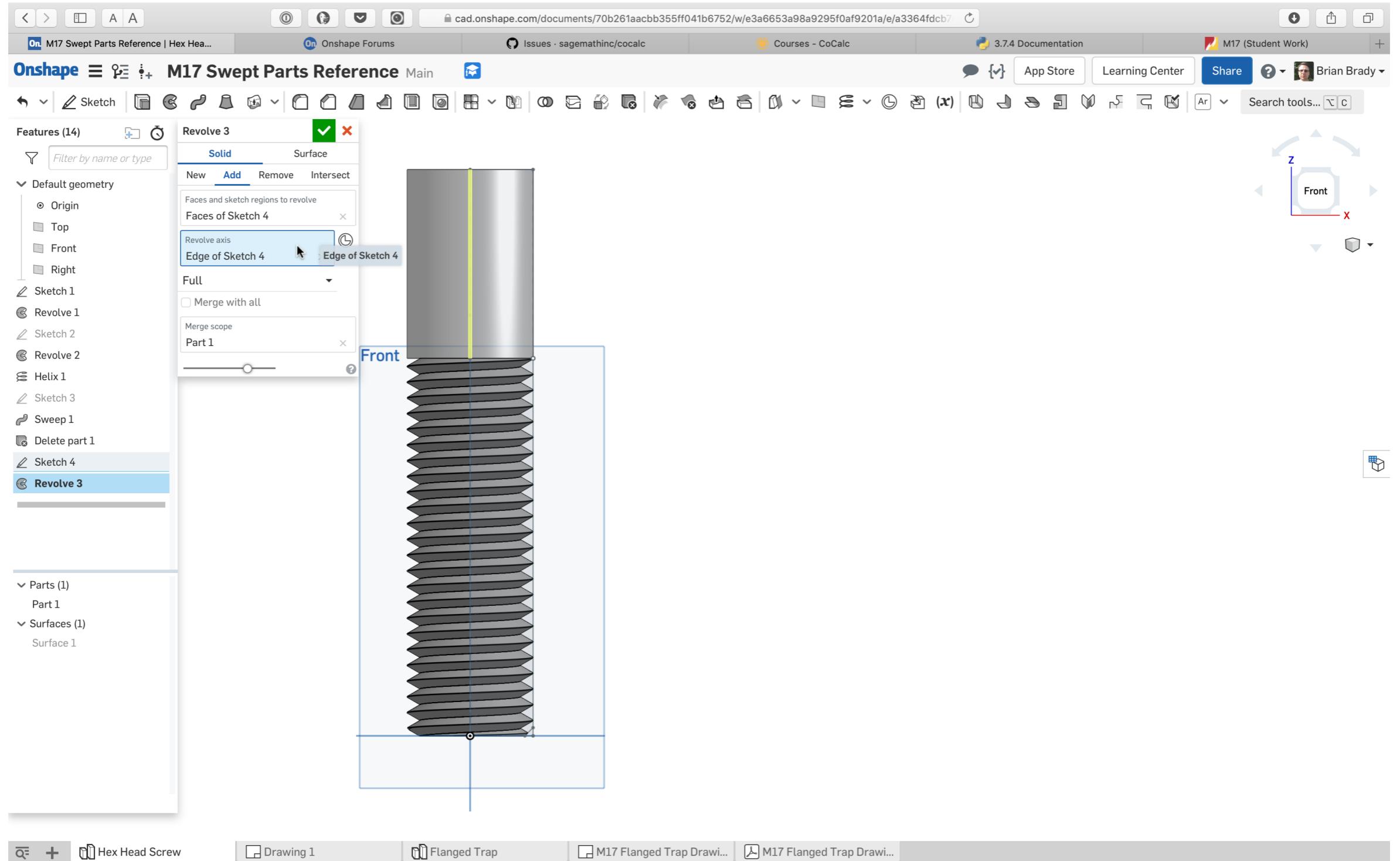
Use the Sweep tool with the helix as the path and the thread cutting sketch as the face/region to sweep. Make sure that Remove is selected instead of Add.



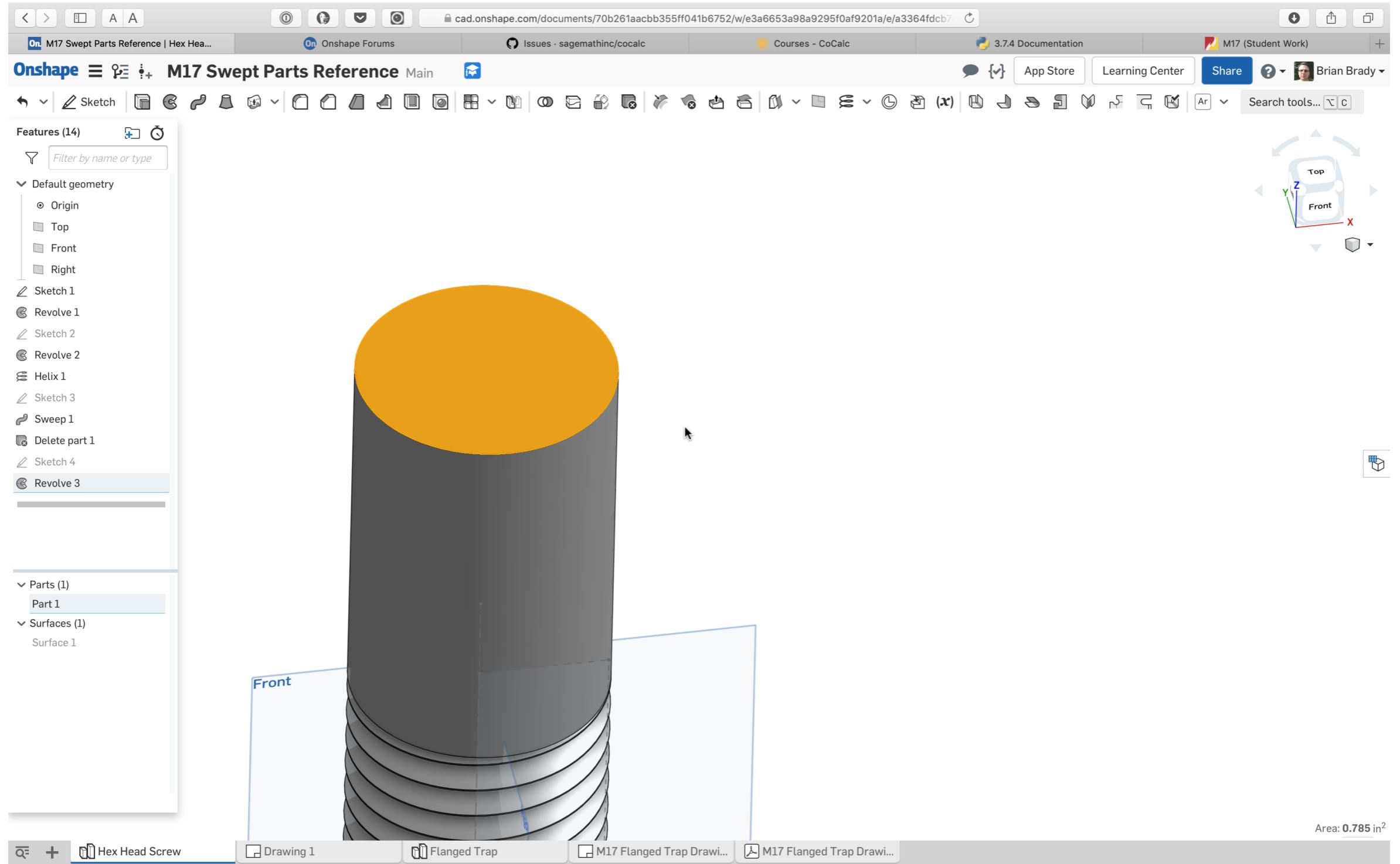
Right click on Curve 1 in the Curves list (you may have to expand the list first) and select Delete. This will add a deleted part feature in the feature list and the helix curve will disappear.



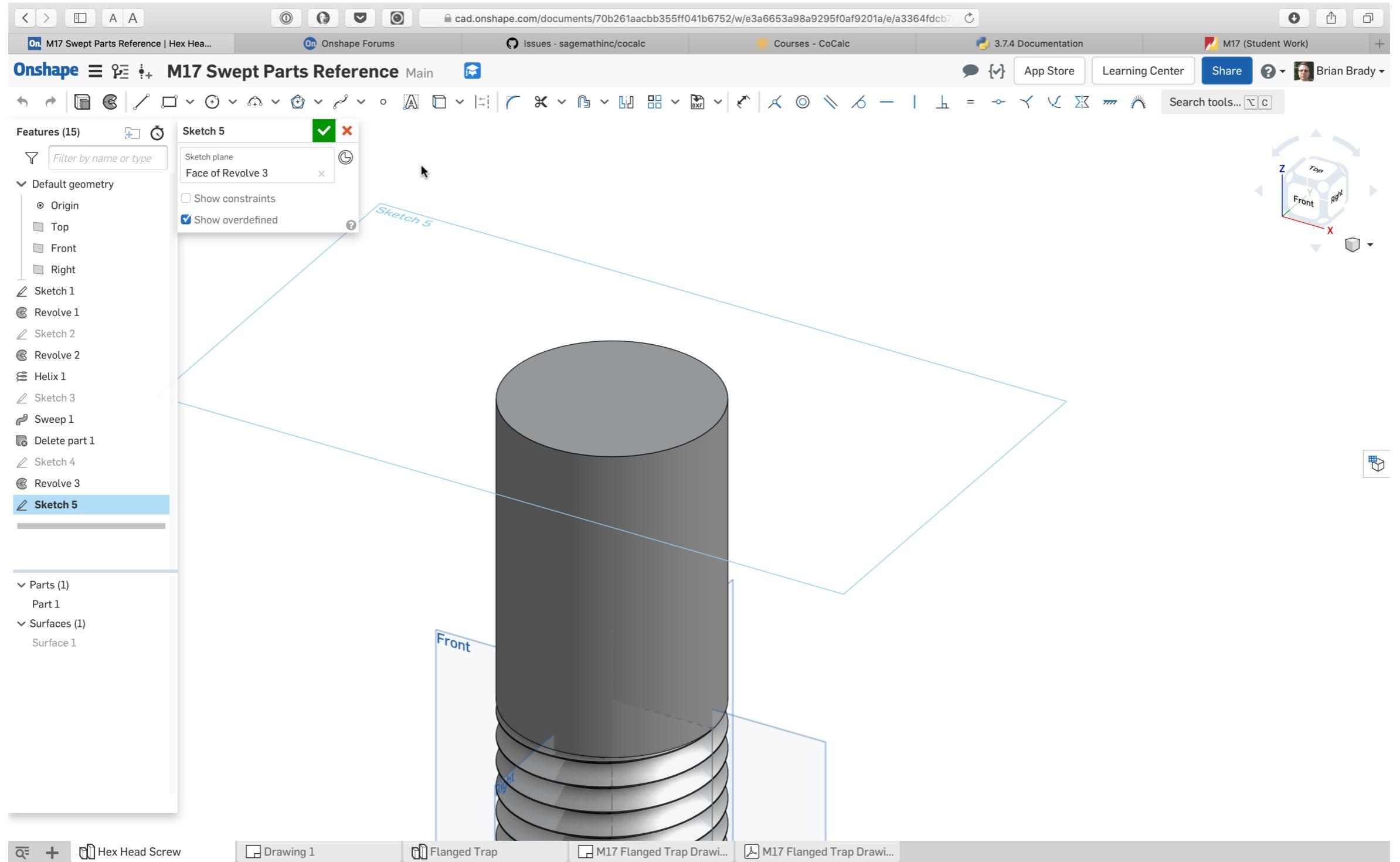
Create a sketch on the front plane for the upper portion of the shank. Notice that the dimension used is the overall length of the screw. This sketch does not require a diameter dimension because the top right corner of the very first sketch is used as a reference (the sketch will have to be visible to do this).



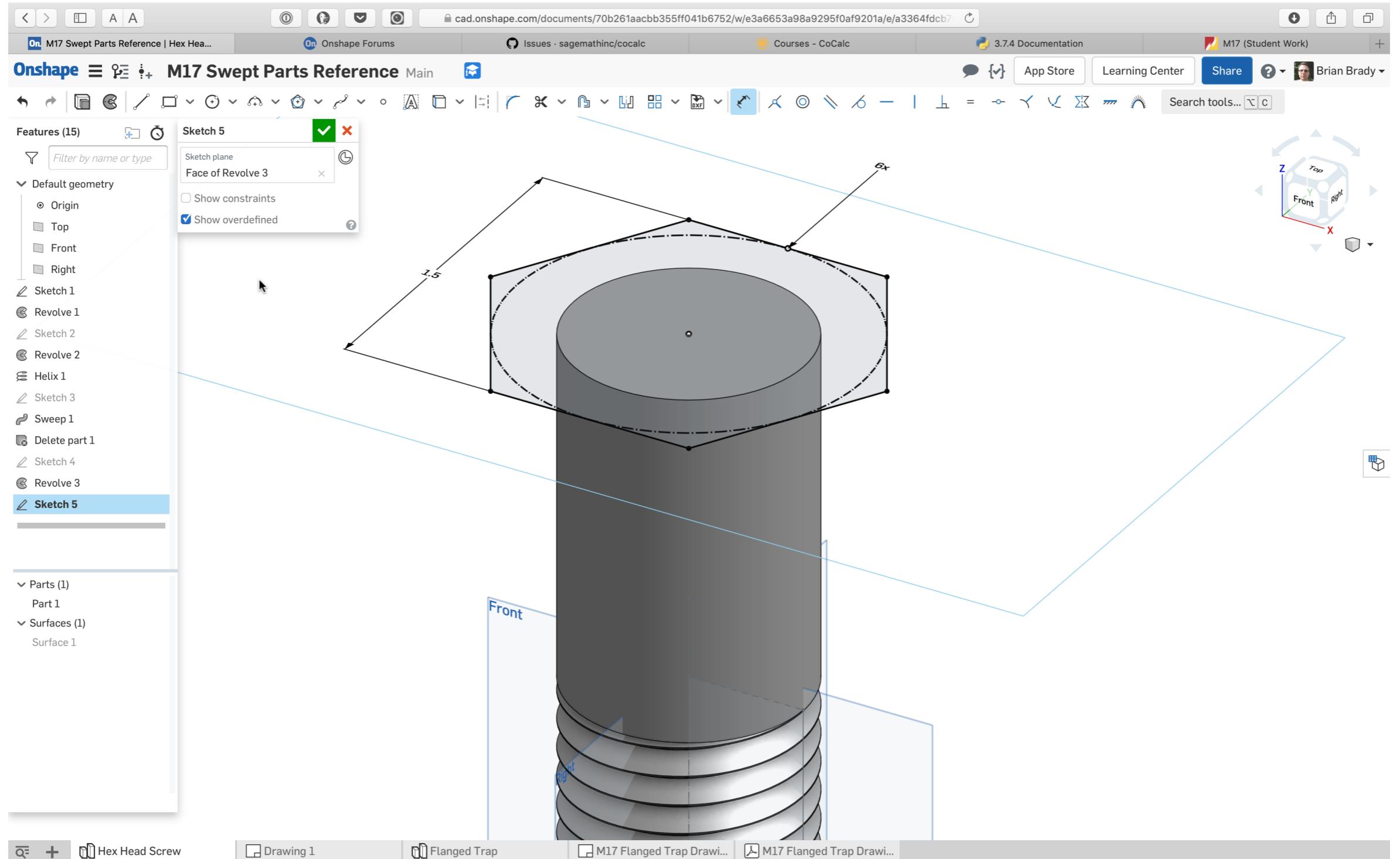
Revolve the previous sketch using the sketch edge that corresponded to the part centerline as the axis. The abrupt edge where the threaded portion meets the non-threaded portion is not exactly realistic, but fine for our needs since we don't generally model threads on parts.



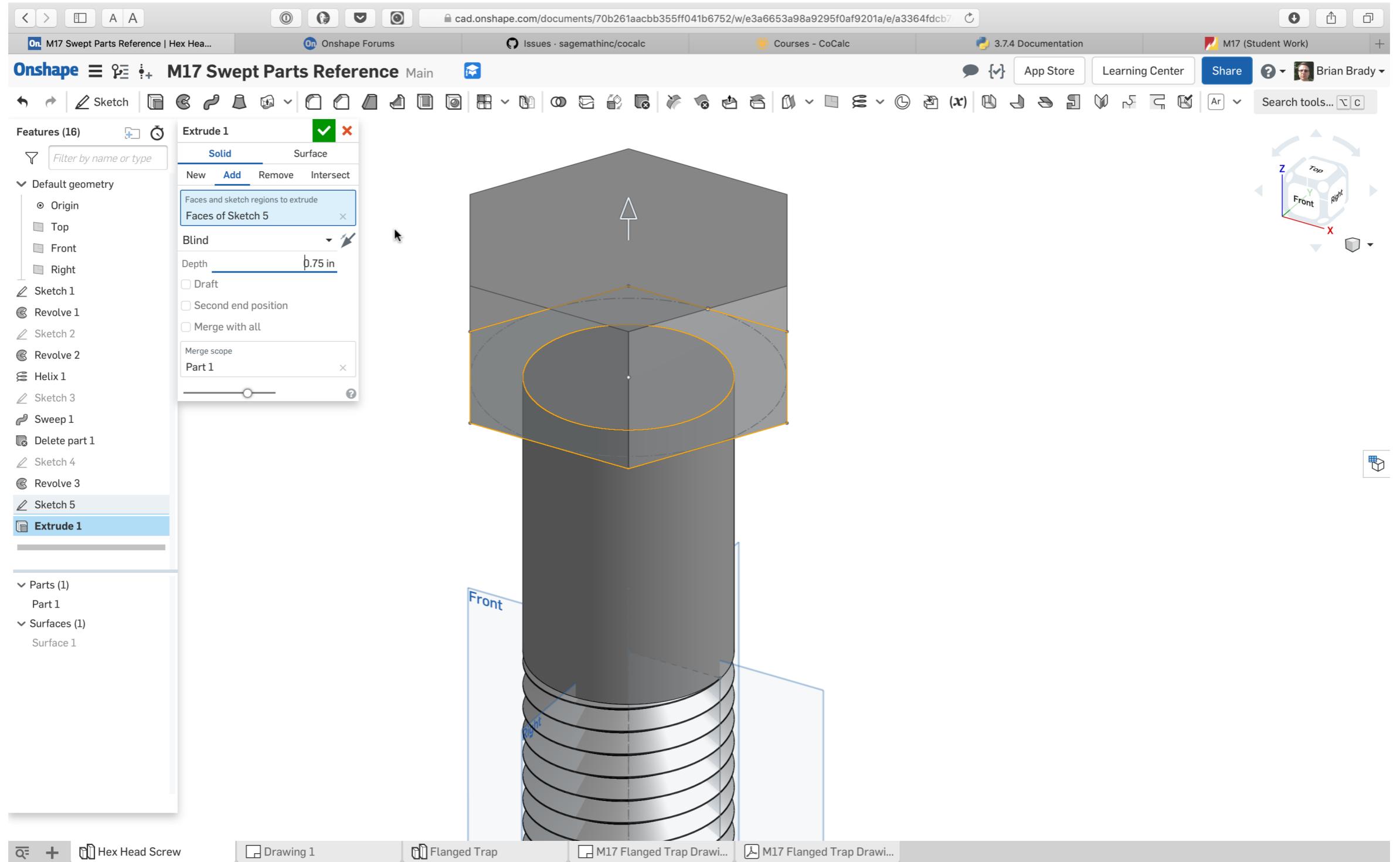
Select the top of the part ...



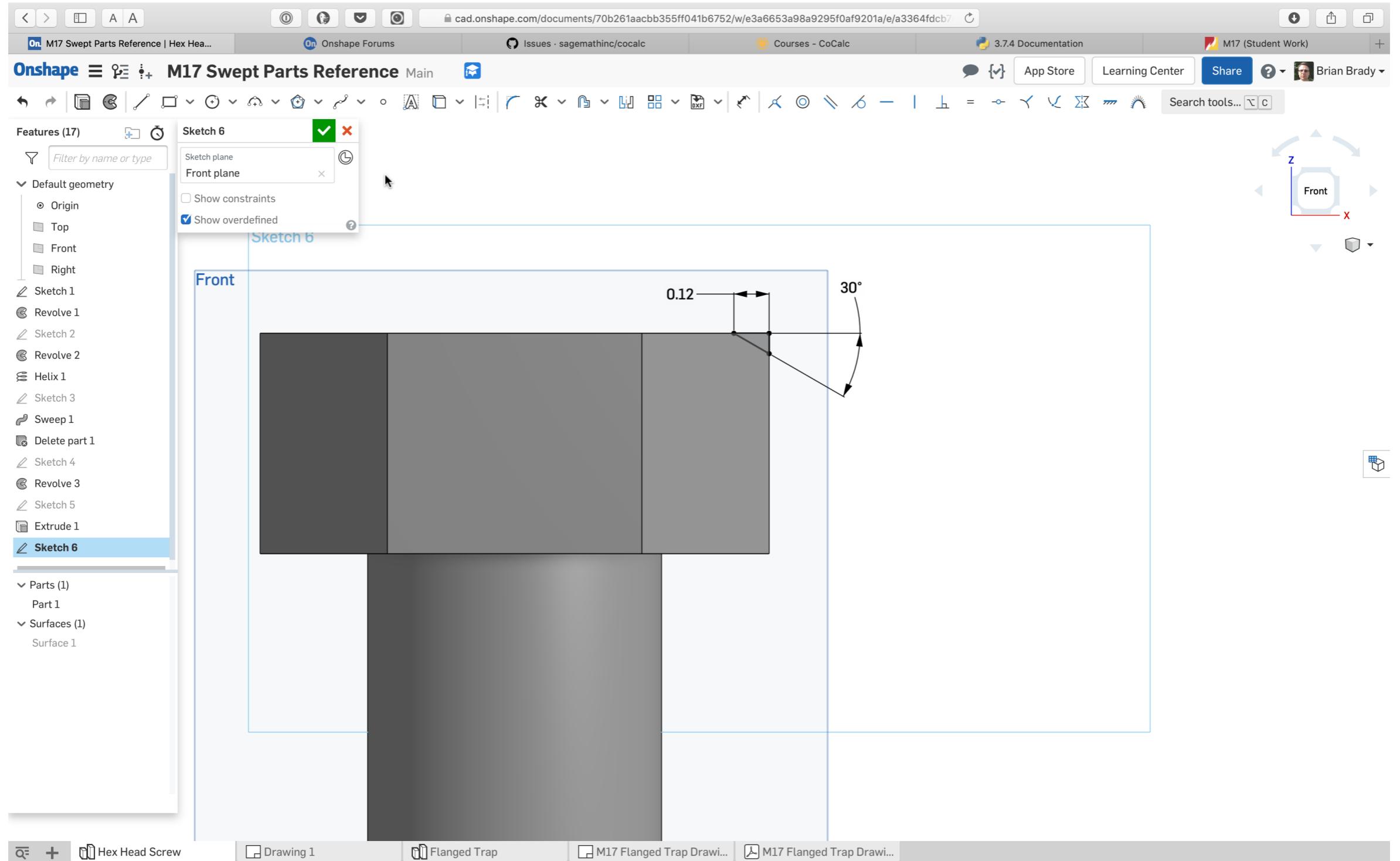
... to create a sketch



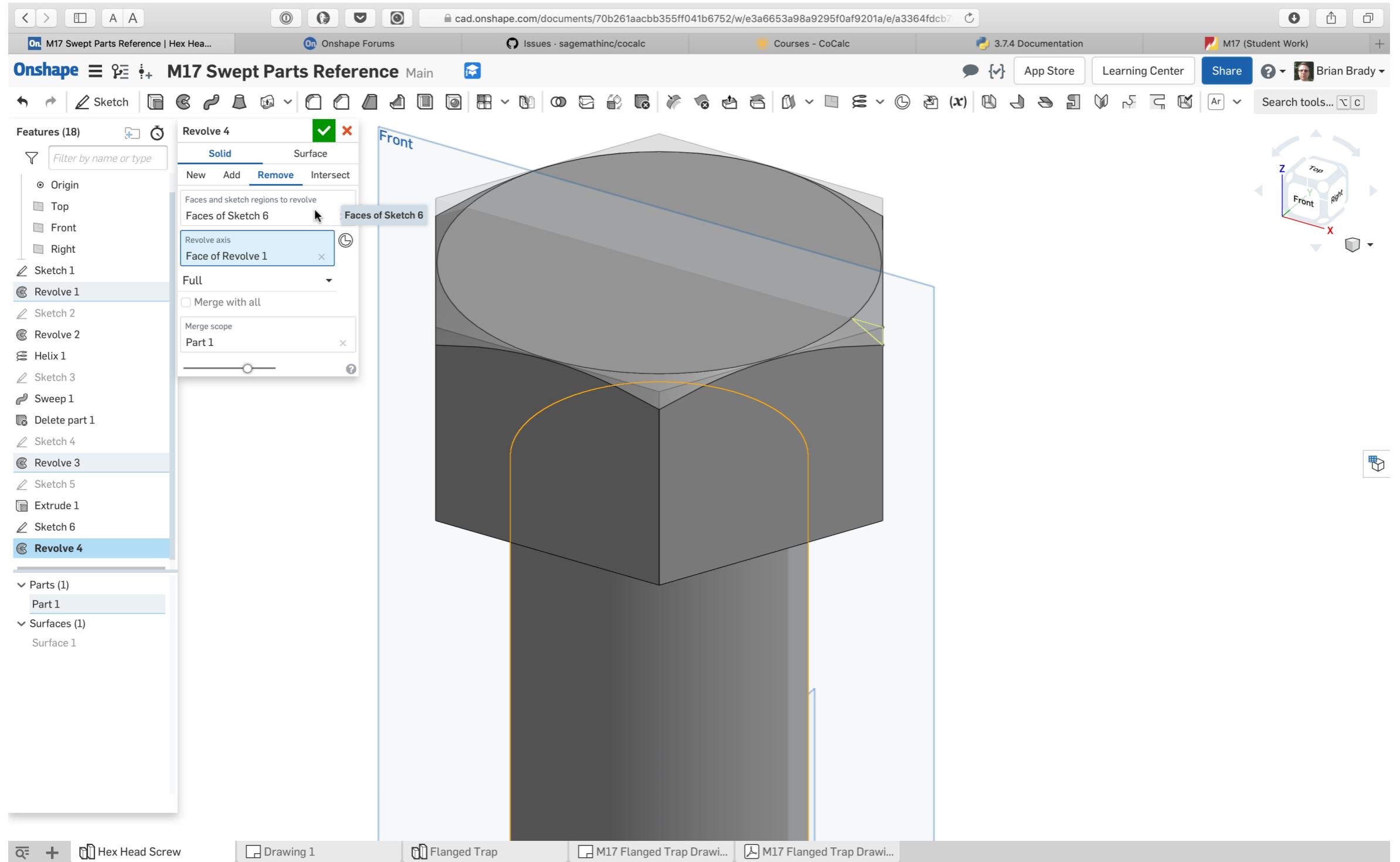
Use the circumscribed polygon tool to create a hexagon and dimension the distance across a pair of edges (the flats on the head of the screw).



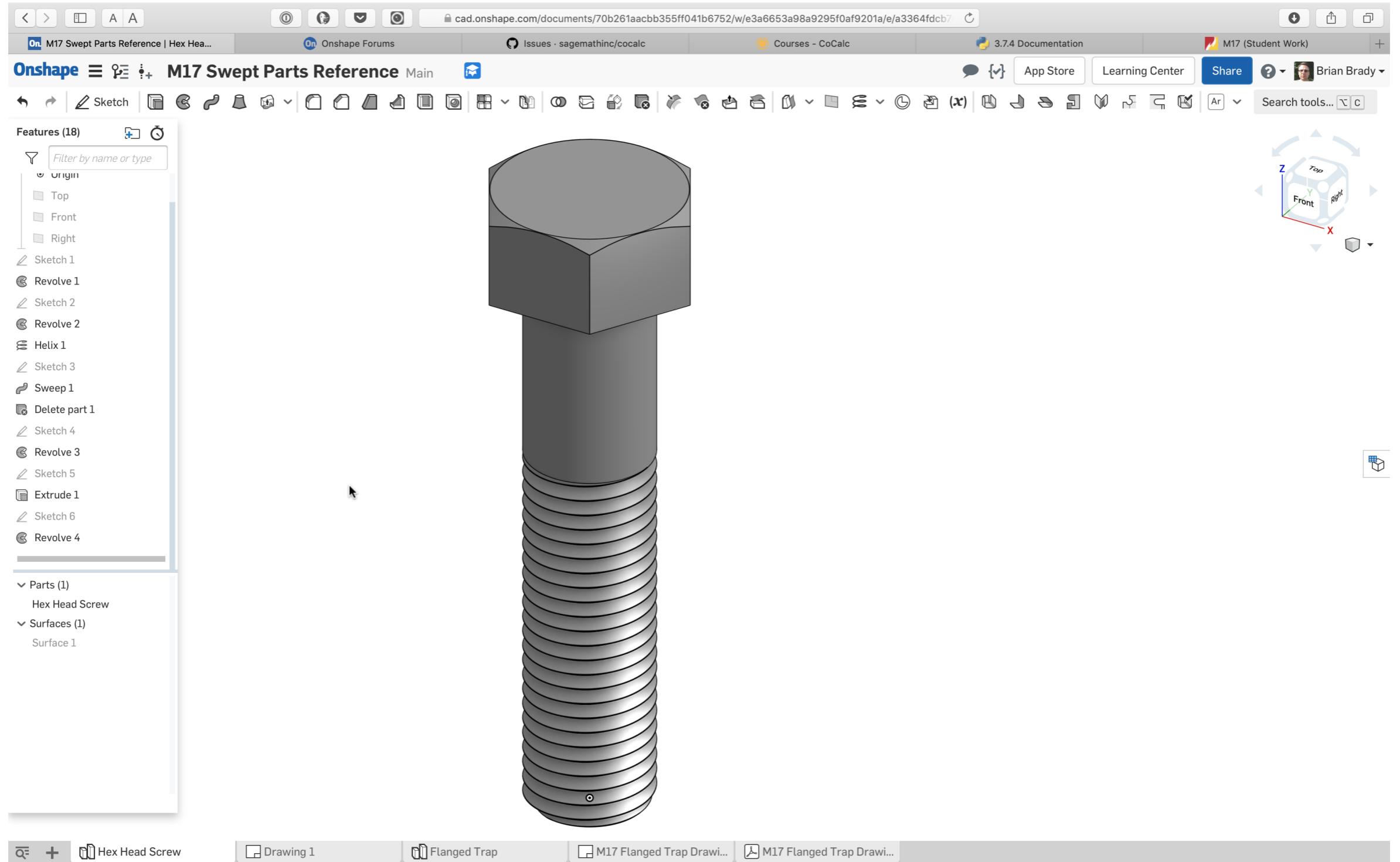
Extrude the hex sketch up by 0.75 inches to create the head.



Create a sketch on either the front or right plane, whichever intersects two of the hexagon vertices. This sketch will be revolved to create the clipped corners on the head (these cannot be created using a standard chamfer). “Use” the vertex of the extruded hexagon for placement of the sketch.



Revolve the previous sketch with the Remove option.



Your screw is finished, be sure to change the names of the part and the Part Studio tab.