A. Setting origin point (0:45)
    a. Select Z-axis/plane and X-axis. Make sure Z is pointing up and X is to the right
    b. Top Back Left corner of **model box point** for repeatability. Always pick a model box point so if you need to set up more than once you can. Select surface against fixed jaw of vice.
B. Setting rough stock size (3:03)
    a. Mode: **Relative Size Box**
    b. Stock Offset Mode: **Add stock to all sides**
    c. **Make sure to measure your rough stock accurately.** Keep as much on the bottom (-Z) as possible for work holding. Minimum of .200".
    d. Double check that the dimensions at the bottom of the stock tab are equal to your actual rough stock size.
C. Facing Operation (6:00)
    a. Select **2D Facing Operation**
    b. Tool: Press select button and follow the prompts to create a tool. I use a 25mm - 2 flute carbide end mill from Tormach. Use the largest diameter tool you have available. Diameter and Tool Number are the most important variables when creating a tool. The lengths of each tool will come from the offsets tab at the machine.
    c. Set speeds and feeds according to video. Her numbers work well and won't destroy any tools.
    d. Geometry tab: Nothing to do here. Will automatically recognize the rough stock size from the setup.
    e. Heights tab: **Set all the "From" boxes to model top**. Because we are facing, the bottom of the cut is the top of the part.
    f. Passes tab: Set Pass Extension to .6" if using a 1" diameter tool. Step over can be 95% of tool diameter. Set Direction to Climb. I check the "Multiple Depths" and set my maximum pass depth to .025" for our 25 mm end mill. This will prevent crashing if kids didn't measure the height of the stock correctly. You could check "Stock to Leave" if you want a finishing pass but it adds time so I don't bother.
D. 3D Adaptive Clearing (15:20)
    a. Select the 3D drop down from the upper right hand corner and select **3D Adaptive Clearing.**
    b. Tool: Create/select a **⅜" end mill**. Needs to be at least 1" of cutting flutes. Spindle speed **4000-5000 rpm**, cutting feedrate **20 in/min**. Ramp and plunge to 15 in/min
    c. Geometry: **Select Rest Machining**. Nothing else needs to be done here.
    d. Heights: Set all the **"Froms" to Model Top**. **Bottom Height offset of -.77**.
    e. Passes: **Set Optimal Load to .015"**. Check the Machine Cavities box. Set Direction to climb. Set **Maximum Roughing Stepdown to .8".** Check the Flat Area Detection box. If you want to speed up machining time DO NOT CHECK STOCK TO LEAVE. This will eliminate the next 2 operations which are a 2D contour and a 2D pocket.

f. Linking Tab: Stay-Down level 70%. No Engagement Feed Rate 90 in/min. Ramping angle 1 degree. Ramp Taper Angle 1 degree. Helical Ramp Diameter .35. Min Ramp Diameter .3

E. 2D Contour (26:00)
   a. Follow all the video prompts for the Tool, Geometry, and Heights tabs
   b. **In the Passes tab set the Compensation Type to In computer.** If you set it to wear the machine will stop about ¾ of the way through the previous operation.
   c. This operation can be skipped by not leaving any stock in the 3D adaptive operation.

F. 2D Pocket (31:30)
   a. Follow all the video prompts.
   b. This operation can be skipped by not leaving any stock in the 3D adaptive clearing operation.

G. 2D Chamfer (35:30)
   a. **Do not model chamfers in the 3D model.** Much easier to program if left as a square edge.
   b. **Set compensation type to In Computer.** If you use wear like the video says it will cause issues in the previous operation.
   c. Be sure to include a **Finishing Overlap of .1"** for best result

H. Second 2D Chamfer (39:35)
   a. **Set Compensation Type to In computer**
   b. In the passes tab add a **Chamfer Clearance of .05"** so you don't crash into the side of the part.

I. Spot Drilling (41:22)
   a. Follow the video. Nothing special to be aware of here.

J. Drill Holes (44:20)
   a. I change the feedrate to 10 in/min and I use a Deep Drilling-full retract drilling operation. I set the Peck Depth to .05". If you don't have CNC drill bits which are shorter, you will break regular drill bits using the setting from the video.

I don't have my kids tap the threads because this is the first part we CAM out.