

HOW TO CREATE MACHINING OPERATIONS IN FUSION FOR THE 3 AXIS TORMACH PCNC440

V2

FEBRUARY 11TH, 2025

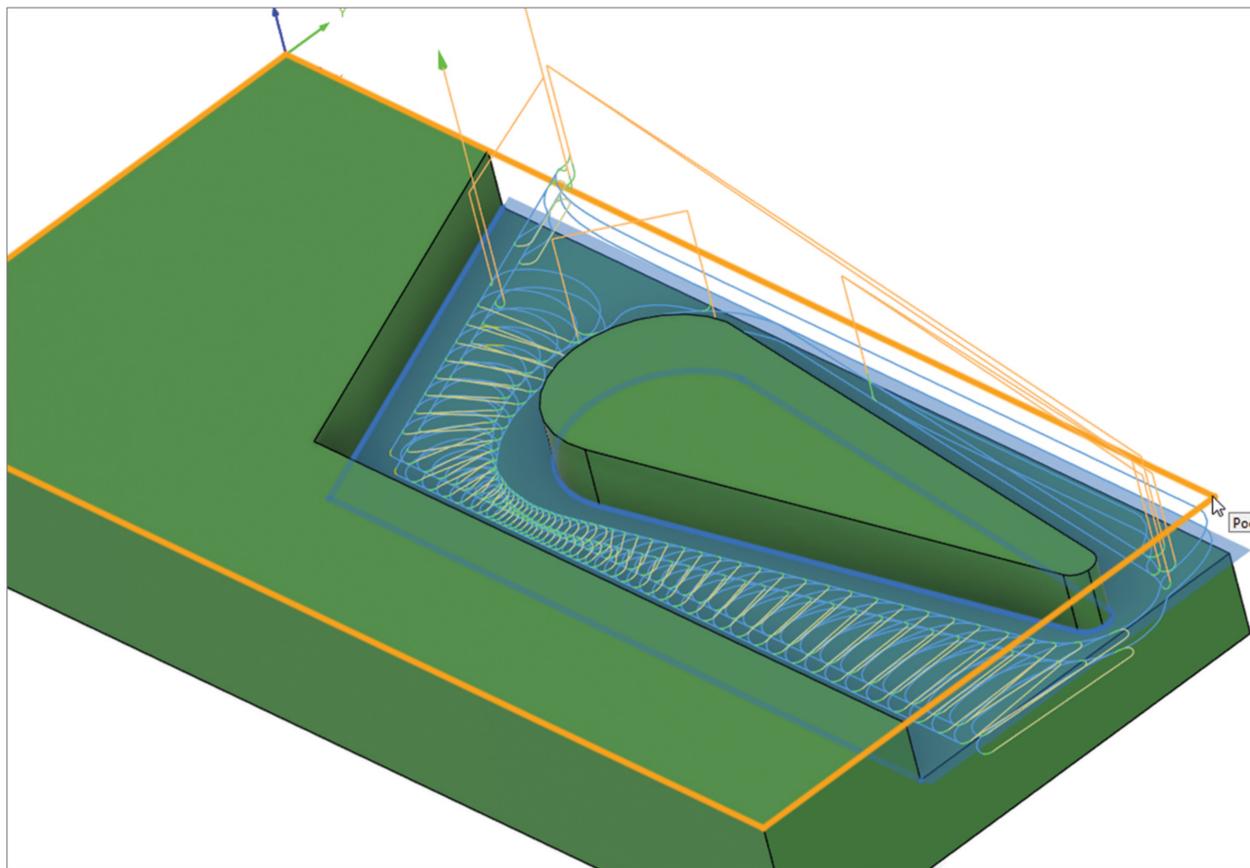


Table Of Contents

Page 1 - MANUFACTURE SETUP.

PROJECT 1

Pages 2 & 3 - THREE AXIS SWARF.

PROJECT 2

Pages 4 & 5 - 2D CONTOUR.

PROJECT 3

Pages 6 & 7 - OPERATION 1 - 2D ADAPTIVE CLEARING.

Pages 8 & 9 - OPERATION 2 - STEEP & SHALLOW.

PROJECT 4

Pages 10 & 11 - OPERATION 1 - FACE.

Pages 12 & 13 - OPERATION 2 - 2D POCKET ROUGHING.

Pages 14 & 15 - OPERATION 3 - 2D POCKET FINISHING.

Pages 16 & 17 - OPERATION 4 - 2D CONTOUR.

Page 18 - OPERATION 4 - HOLE RECOGNITION.

PROJECT 5

Pages 19 & 20 - FILLET EDGE - 2D CONTOUR.

PROJECT 6

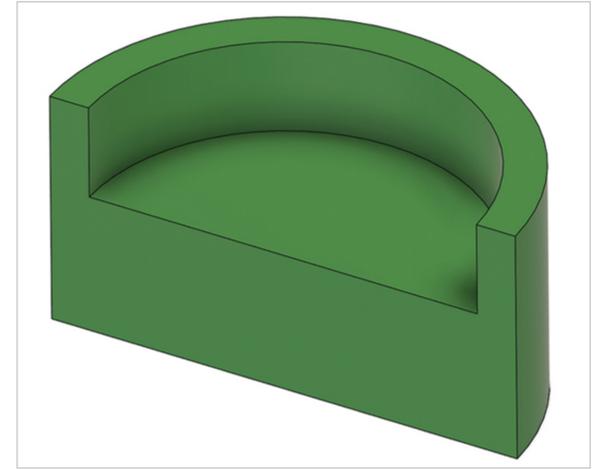
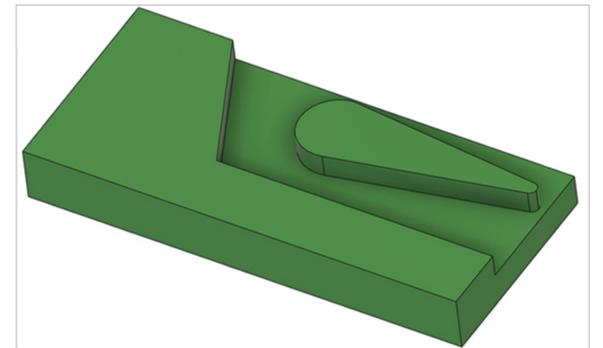
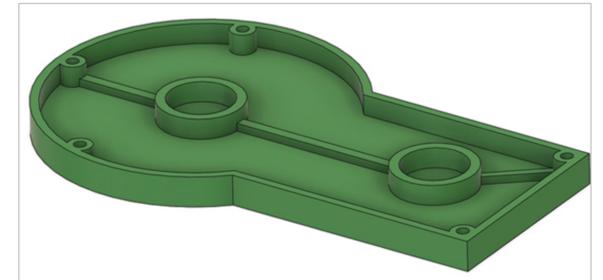
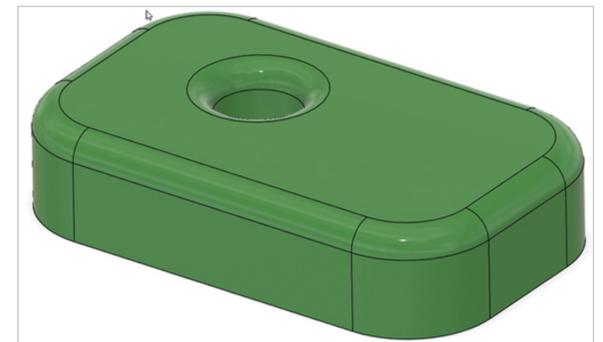
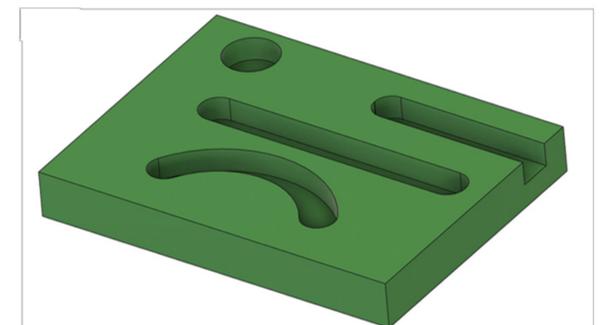
Pages 21 & 22 - SLOTS.

PROJECT 7

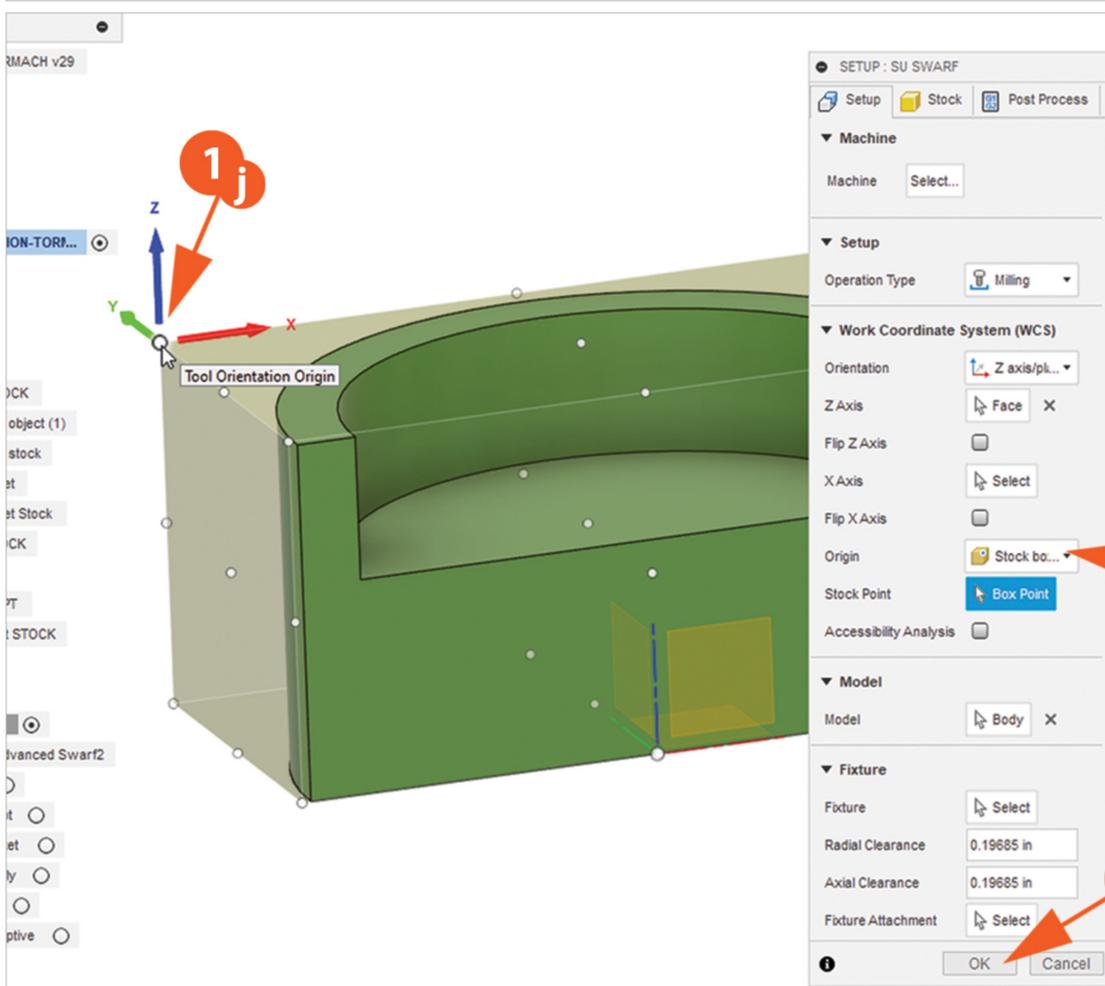
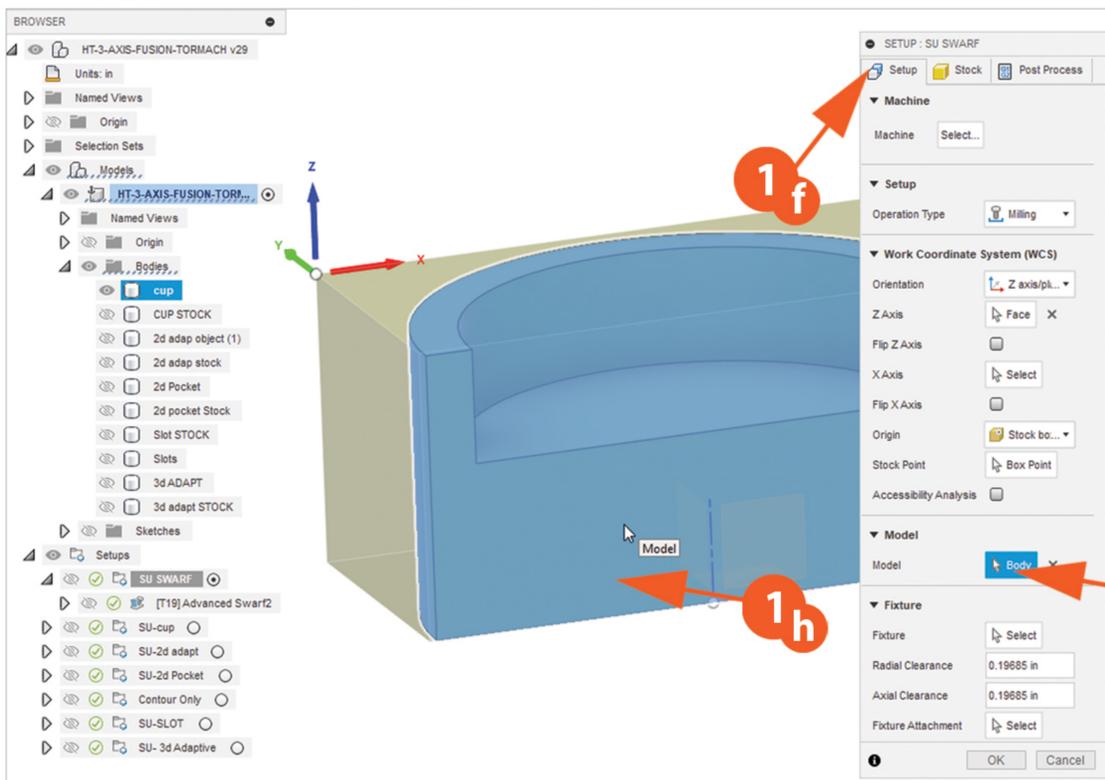
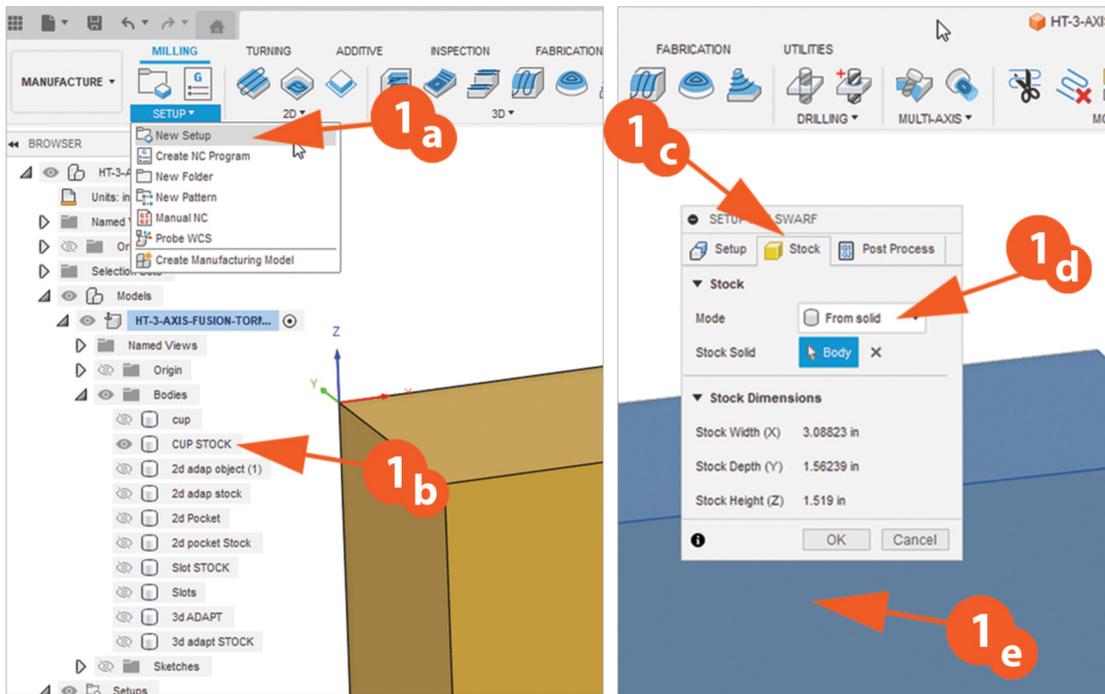
Pages 23 & 24 - OPERATION 1 - 3D ADAPTIVE.

Page 25 & 26 - OPERATION 2 - 3D PARALLEL.

Page 27 - POST PROCESS - CREATE THE G-CODE.


PROJECT 1 & PROJECT 2

PROJECT 3

PROJECT 4

PROJECT 5

PROJECT 6

PROJECT 7



1

MANUFACTURE SETUP

You need to create a setup for each object or each side of an object with multiple machined sides.

1a - Select SETUP/NEW SETUP.

1b - Make sure your STOCK is visible.

1c - Select the 2nd TAB.

1d - Select MODE/FROM SOLID.

1e - Select your STOCK.

1f - Go to the First TAB.

1g - Make sure your object is visible. Select MODEL/BODY.

1h - Select your MODEL.

1i - Select WORK COORDINATE SYSTEM/ORIGIN/STOCK BOX POINT.

1g - Select the Top- Left- Back point of your Stock.

1k - Click OK.

2

3 AXIS SWARF

Swarf is using the side of the tool bit to clear out a section of your stock.

If this is a new object, you will need to create a new setup. Please see page 1.

2a - Select MULTI-AXIS/SWARF.

2b - On the first tab of the pop-up menu select the following:

- Tool -#19 (6mm FEM Long)
- Coolant = Flood
- Spindle Speed = 10000
- Cutting Feedrate = 15in/min
- Plunge Feedrate = 8 in/min
- Ramp Feedrate = 8in/min

2c - Select the 2nd TAB.

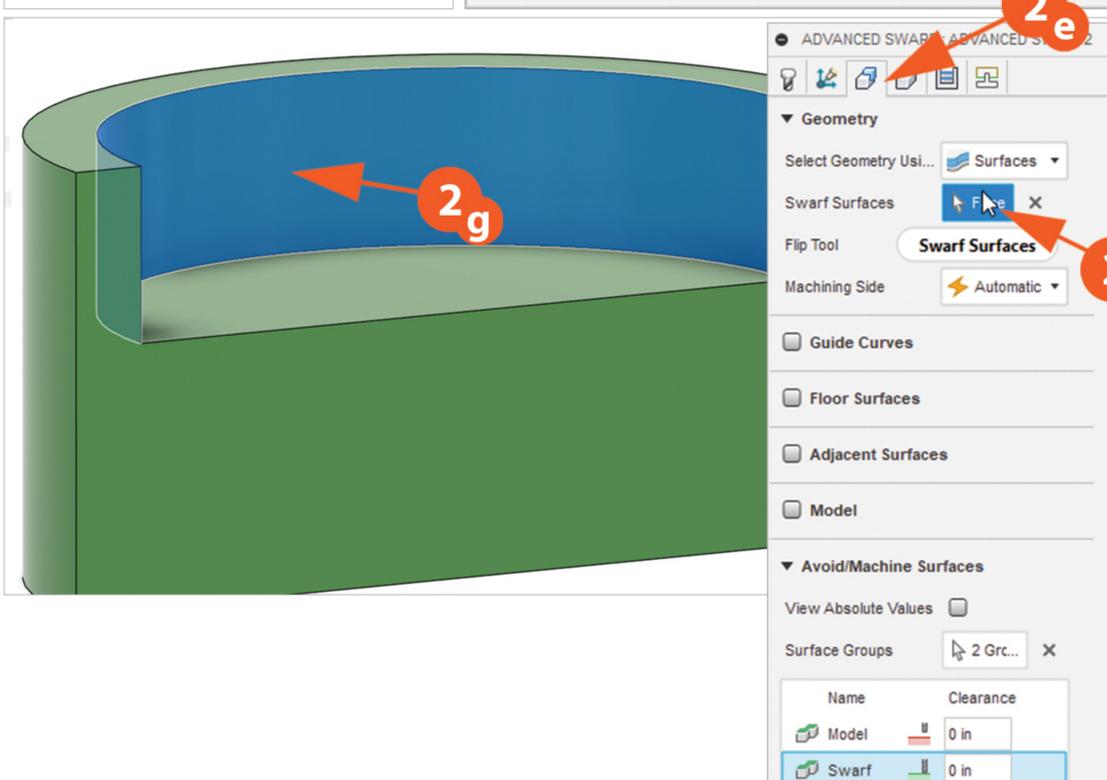
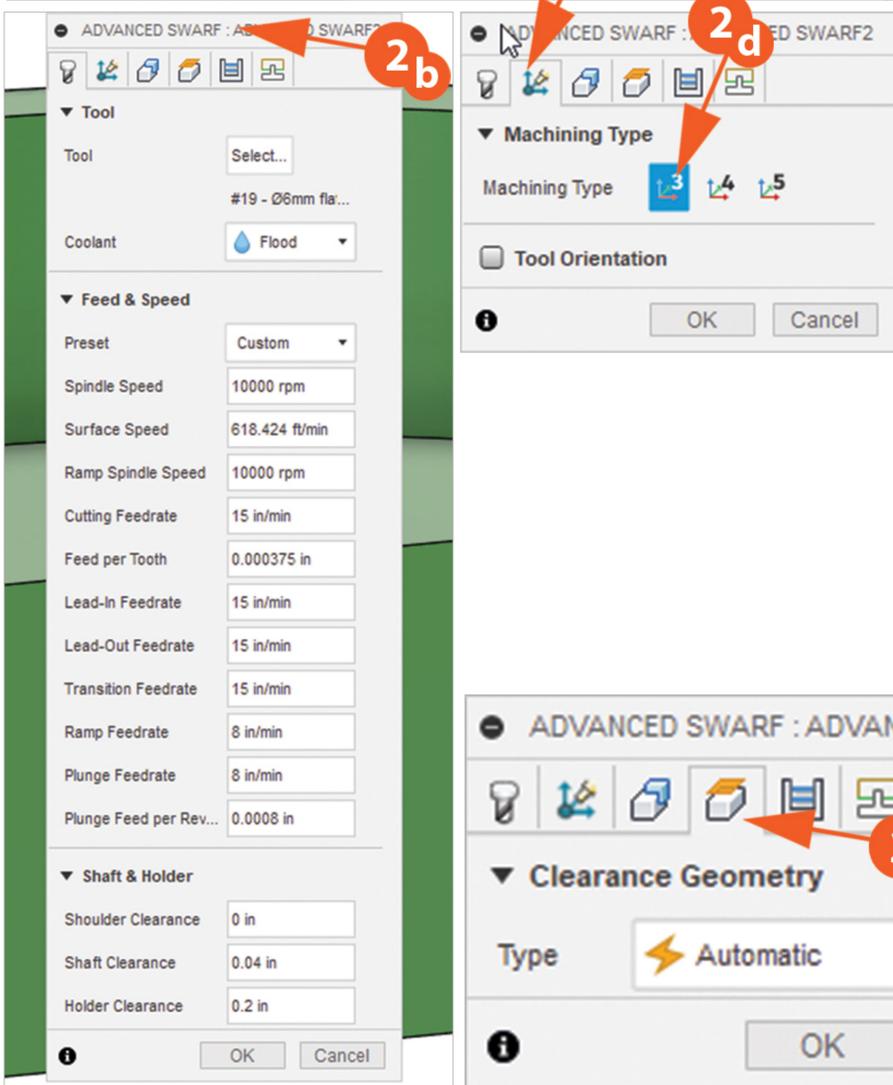
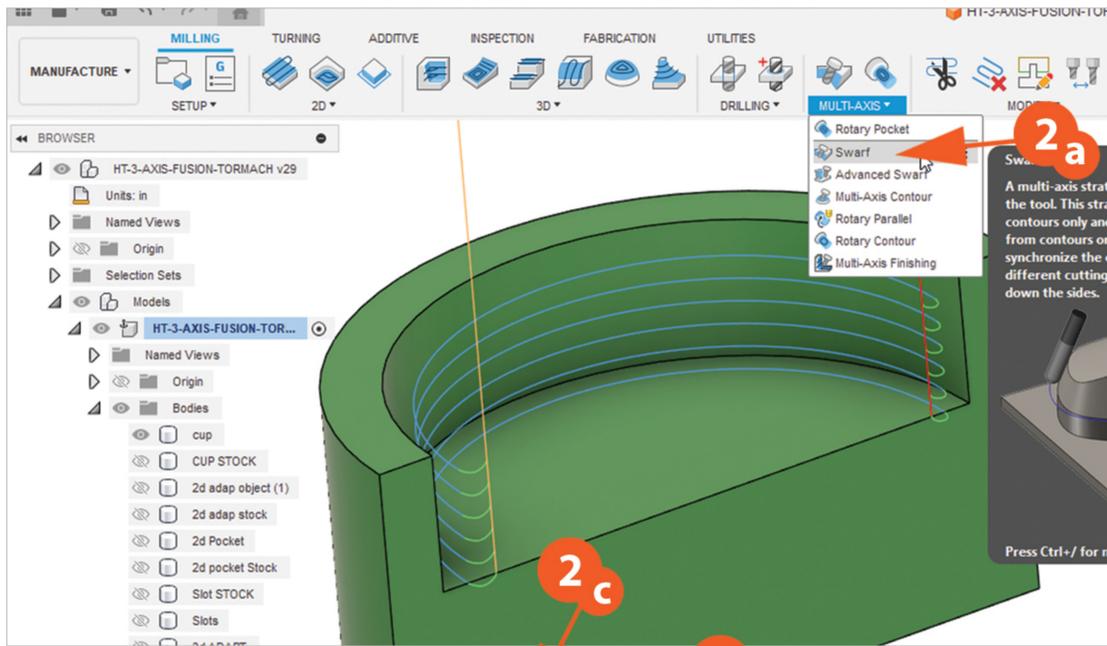
2d -Machine type should be 3.

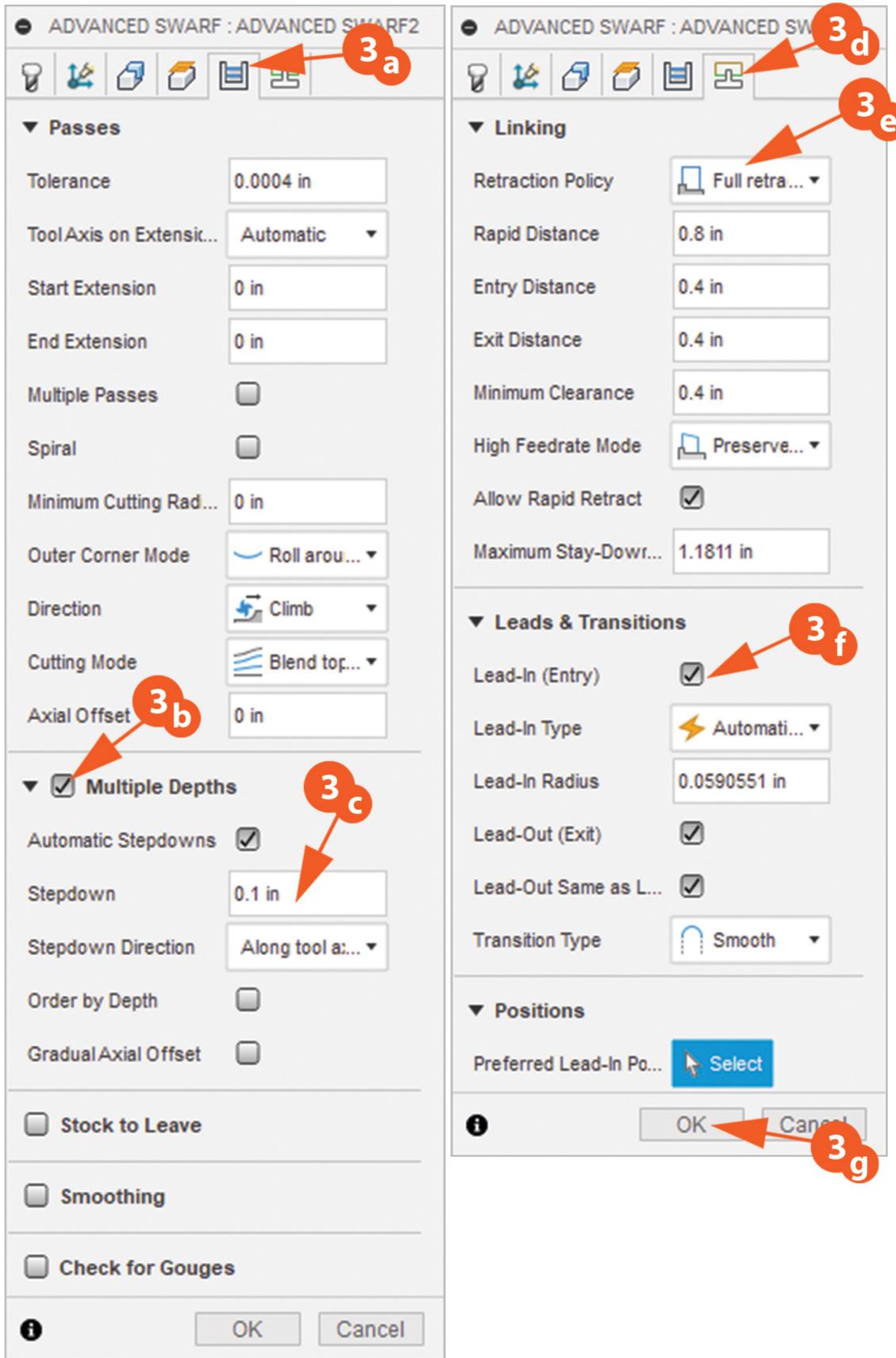
2e - Select the 3rd TAB.

2f - Select SWARF/SURFACES.

2g - Select the FACE you need to SWARF.

2h - Select the 4th TAB, it should be on AUTOMATIC.





3

3 AXIS SWARF Continued

3a - Select the 5th TAB.

3b - CHECK MULTIPLE DEPTHS.

3c - STEPDOWN .01"

3d - Select the 6th TAB.

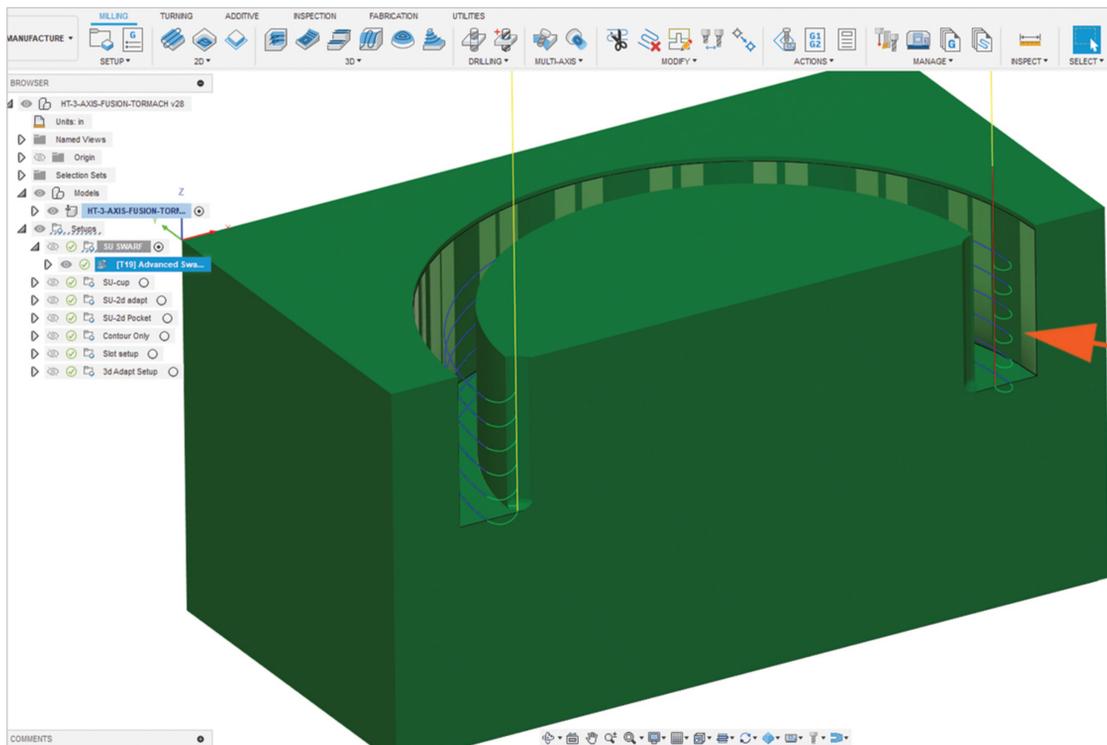
3e - Select FULL RETRACTION.

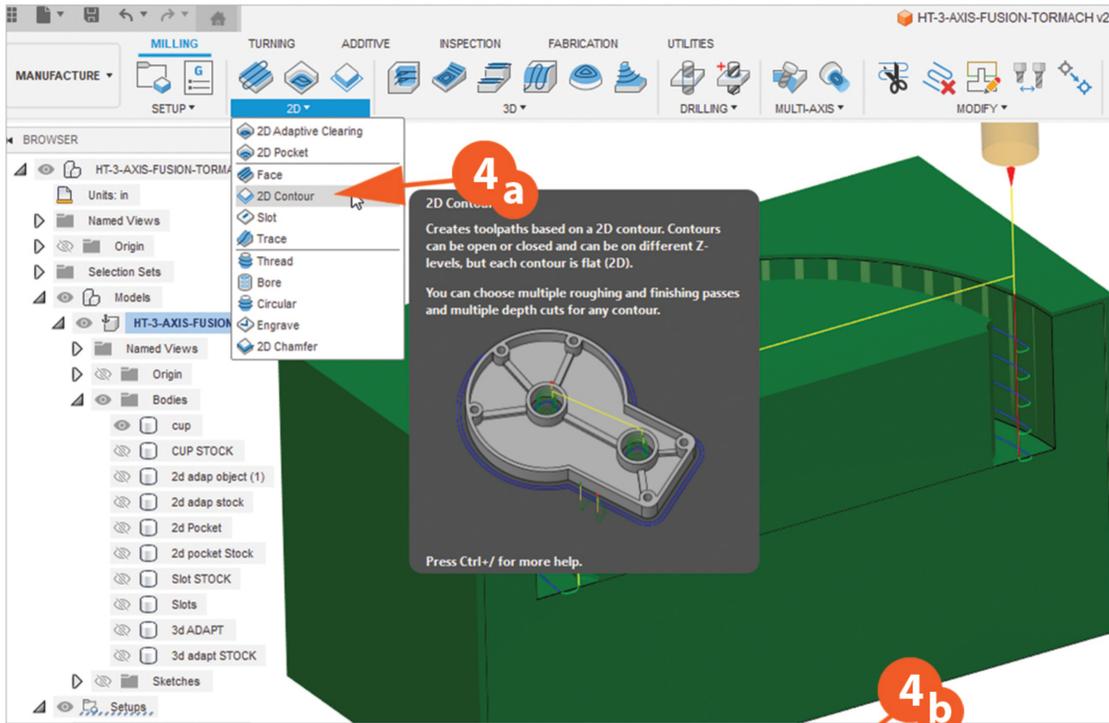
3f - Select LEAD-IN (Entry).

3g - Click OK.

3h - The Tool Path should look like this.

See the Post Processing Chapter (on page 27) to create your G-Code for the Tormach PCNC440.





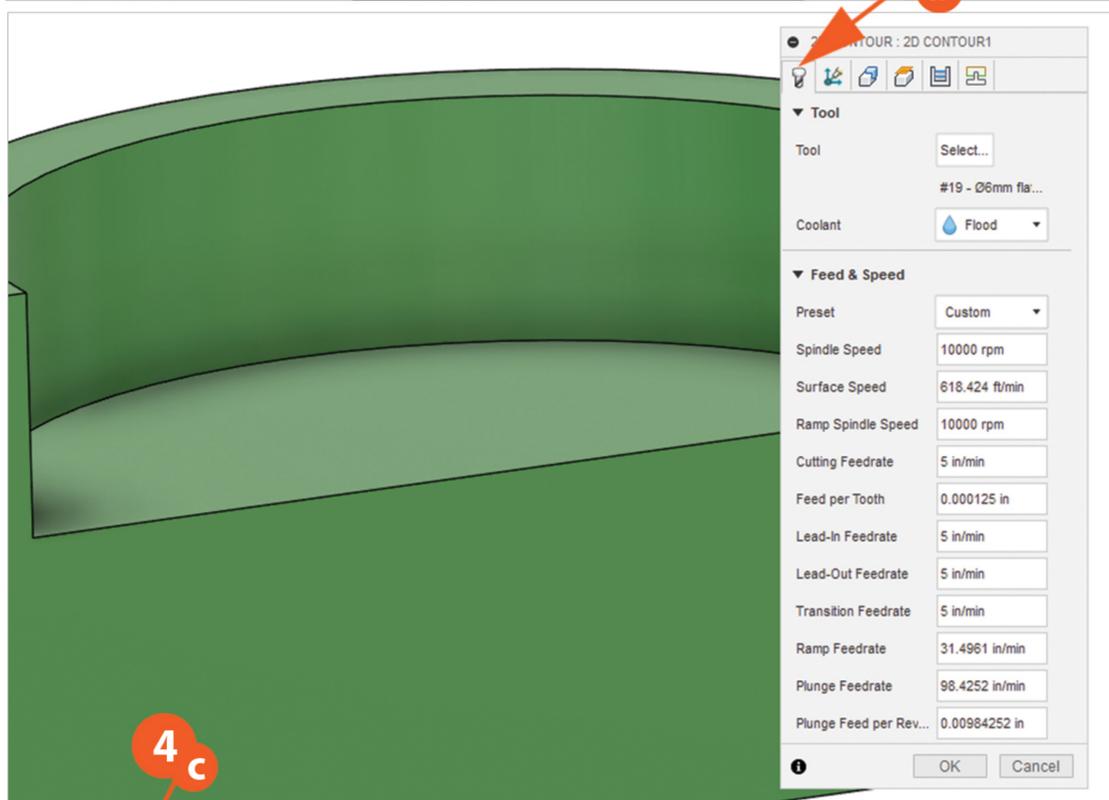
4

2D CONTOUR

2d Contour is an alternate way to create a SWARF cut, and also to cut out a profile.

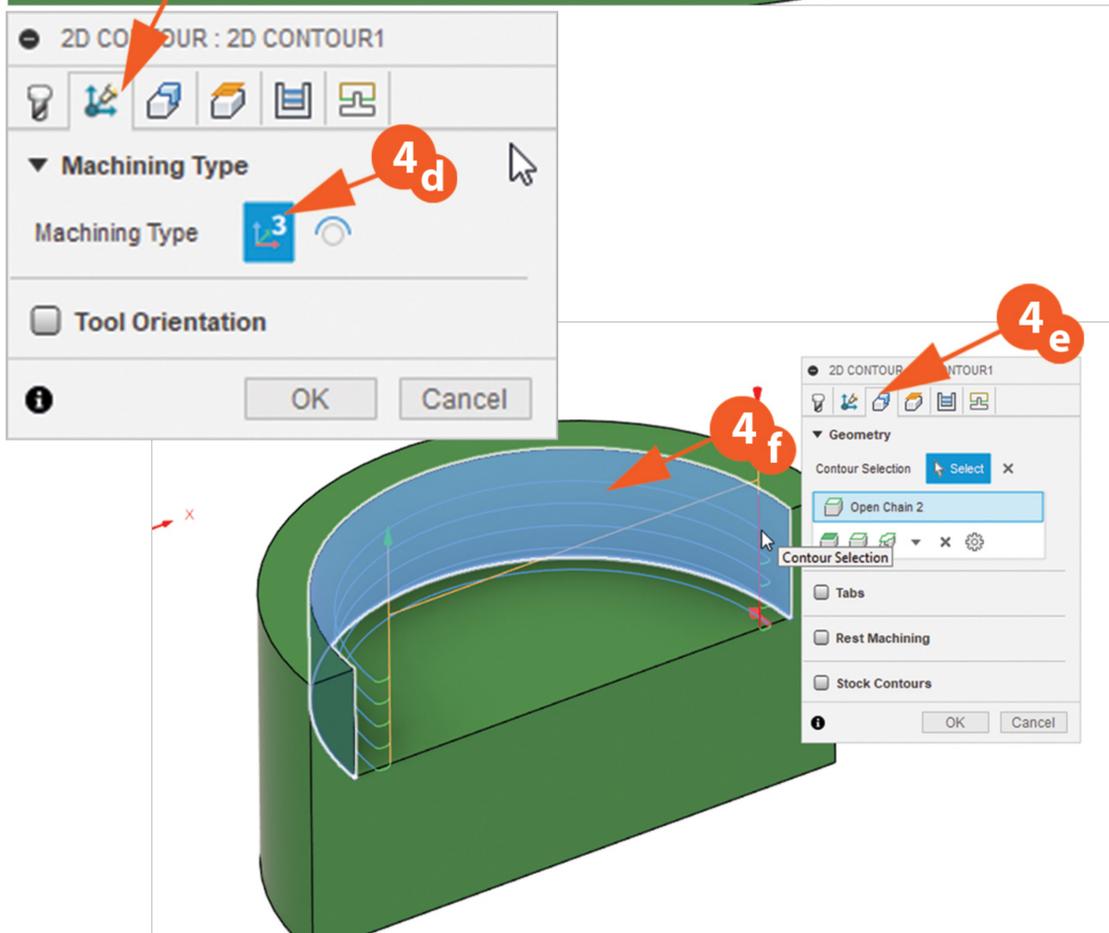
If this is a new object, you will need to create a new setup. Please see page 1.

4a - Select 2D/2D CONTOUR.



4b - Select TAB 1 & input the following:

- Tool = #19 (6mm FEM Long)
- Coolant = Flood
- Spindle Speed = 10000rpm
- Cutting Feedrate = 5 in/min
- Plunge Feedrate = 2 in/min

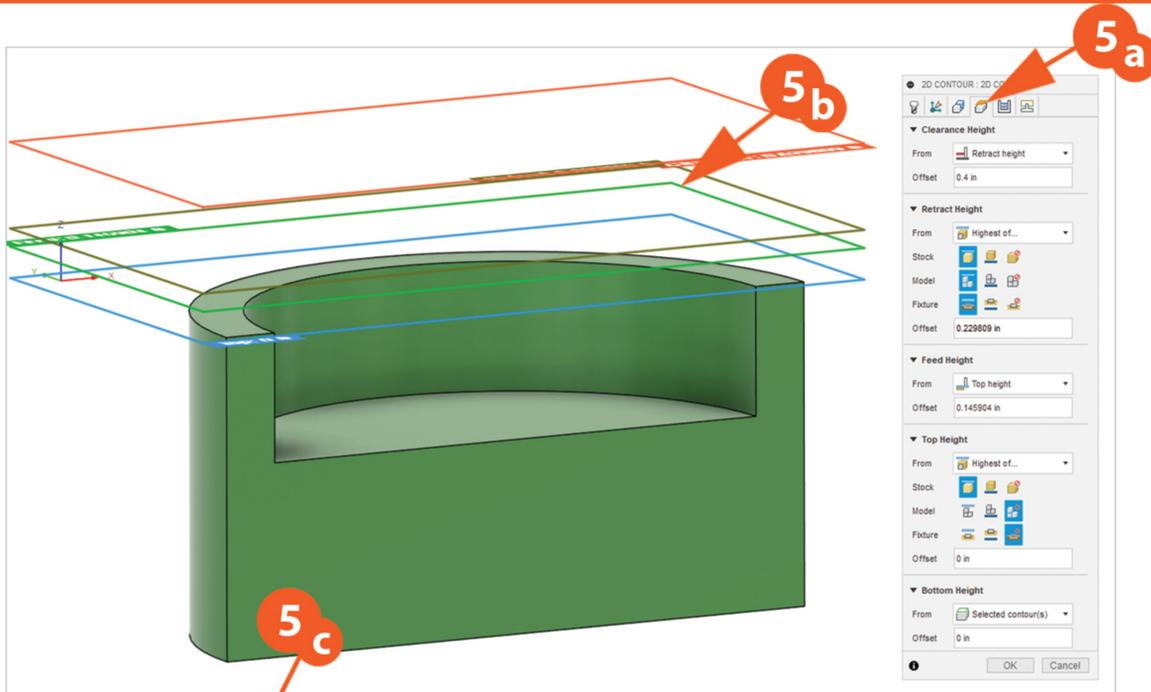


4c - Click TAB #2.

4d - It should be on 3.

4e - Click TAB #3.

4f - Select the FACE to be cut.



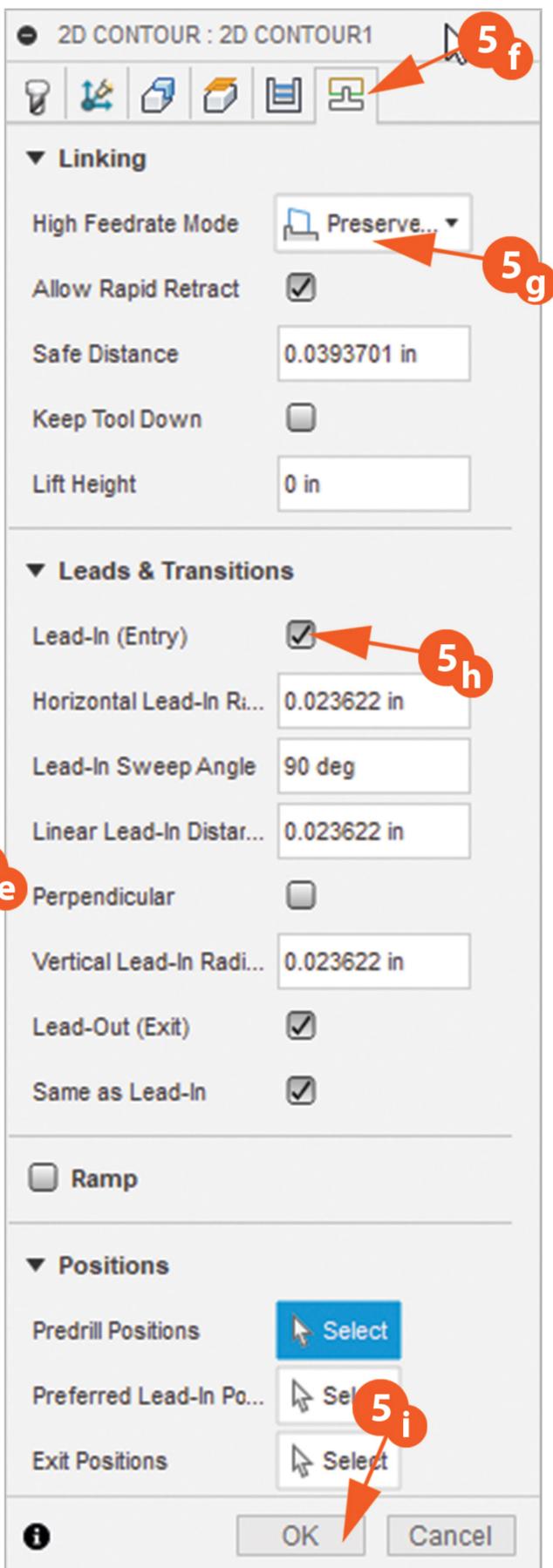
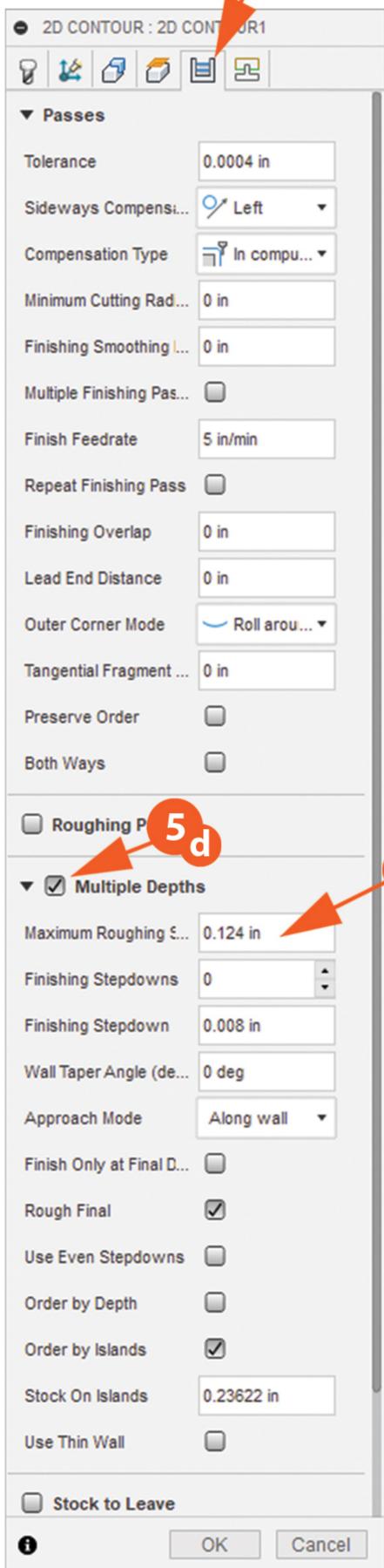
5

2D CONTOUR CONTINUED

5a - Select TAB 4.

5b - The CLEARANCE & RETRACT heights should be close to shown here.

5c - Select TAB 5.



5d - Check MULTIPLE DEPTHS.

5e - MAXIMUM ROUGHING DEPTH Set .124"

5f - Select TAB 6.

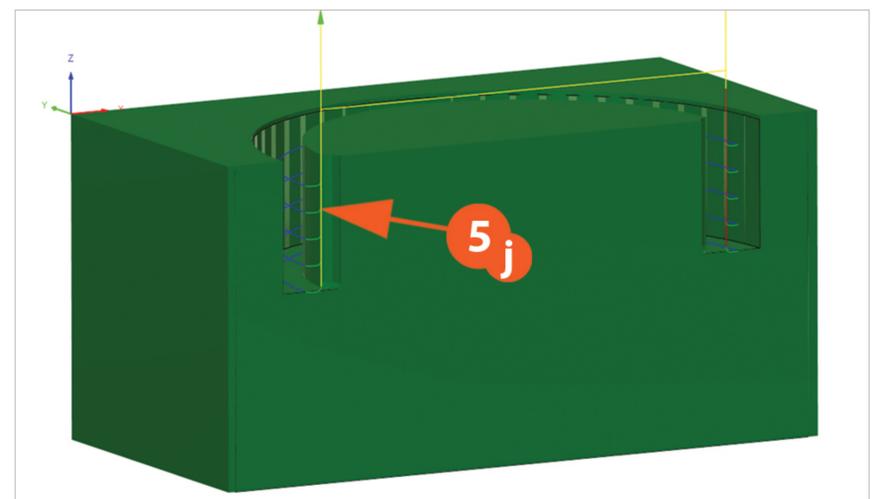
5g - PRESERVE RAPID RETRACT.

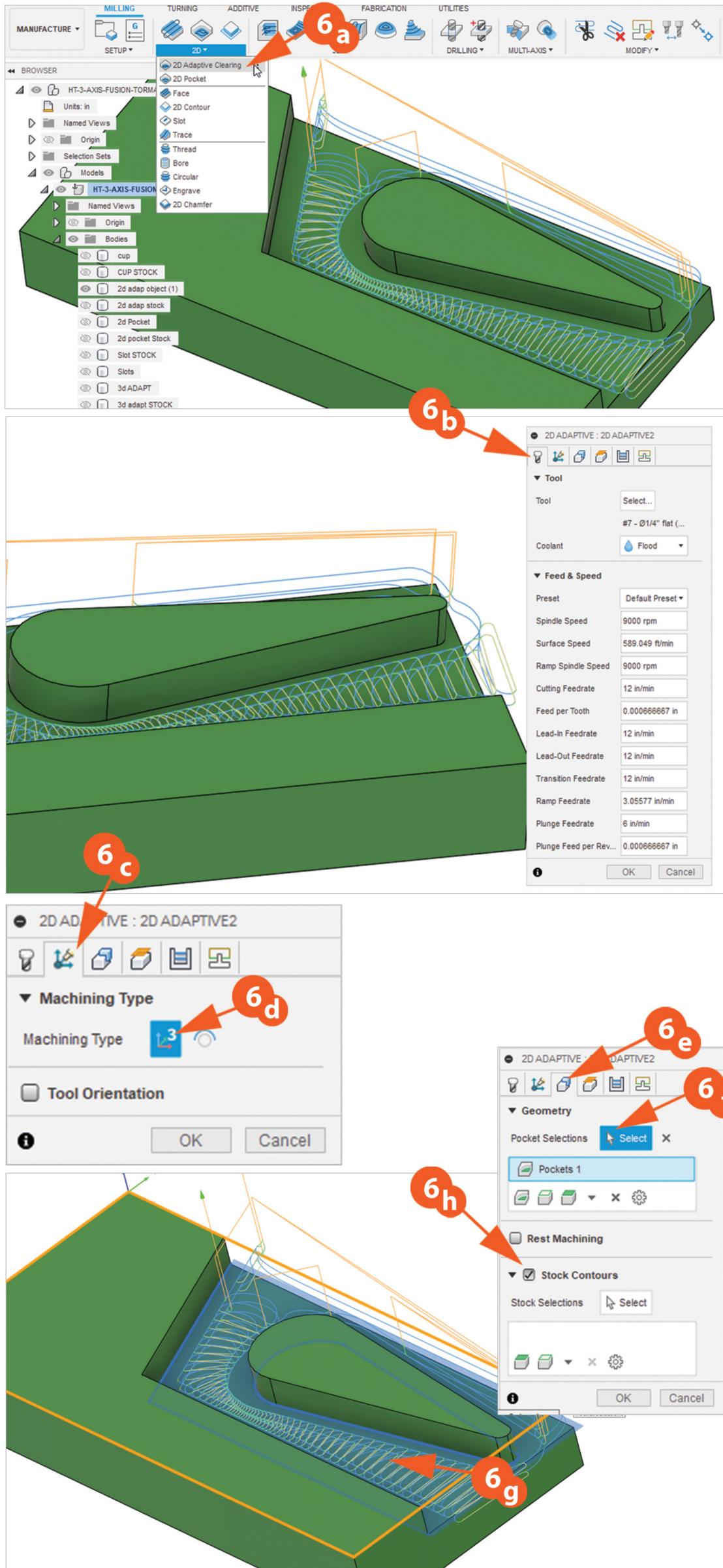
5h - Check LEAD IN ENTRY.

5i - Click OK.

5j - The machine path should look like this.

See the Post Processing Chapter (on page 27) to create your G-Code for the Tormach PCNC440.





6

We will be using **2D Adaptive** to rough machine the area, and then **Steep & Shallow** to finish the area.

2D ADAPTIVE - OPERATION #1

2D Adaptive is used to rough out an area quickly.

If this is a new object, you will need to create a new setup. Please see page 1.

6a - Select 2D/ADAPTIVE CLEARING.

6b - In TAB 1, input the following settings:

- Tool = #7 (.25" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 12 in/mi
- Plunge Feedrate = 6 in/min

6c - Select TAB 2.

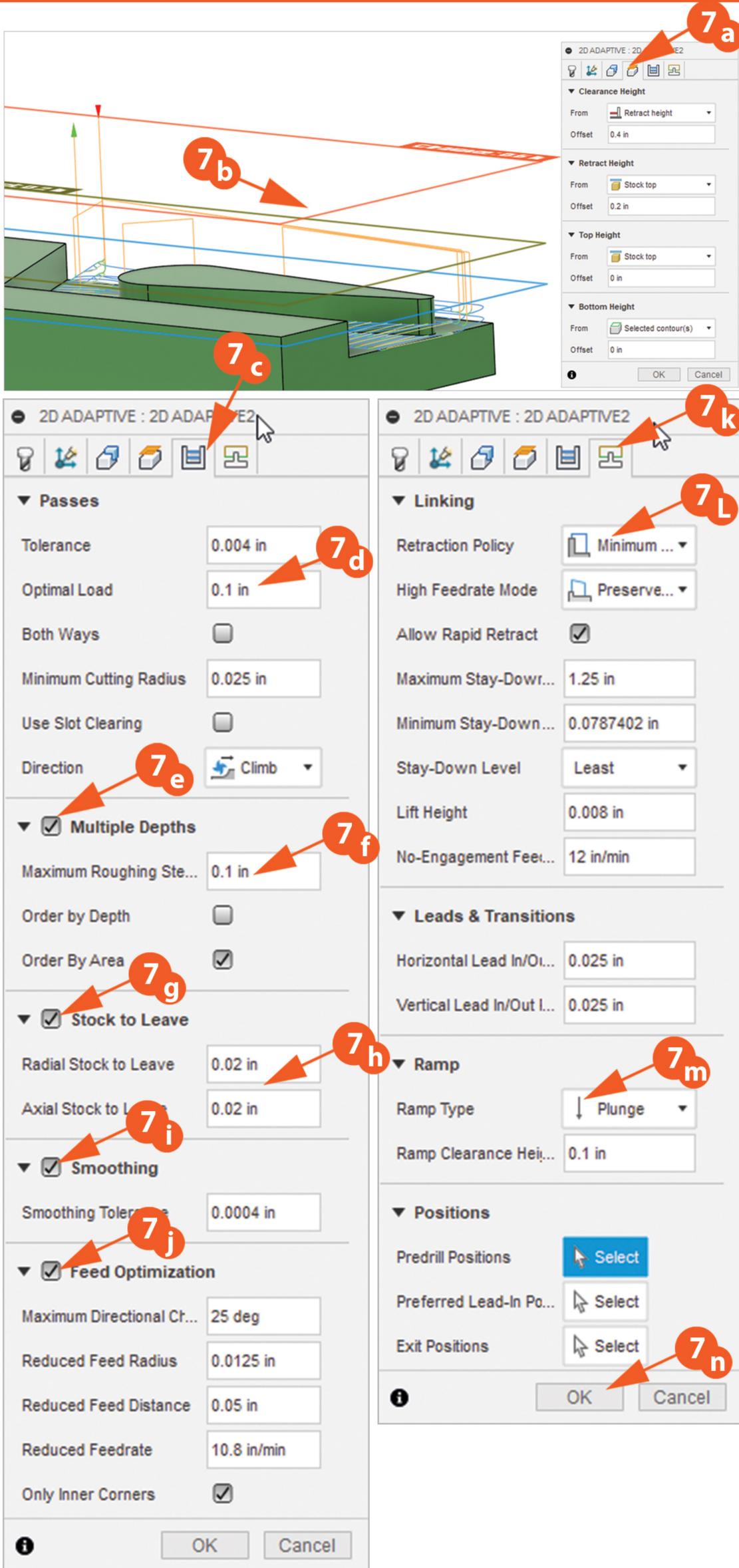
6d - It should be on 3.

6e - Select TAB 3.

6f - Select GEOMETRY/POCKET SELECTIONS.

6g - Select the area you want machined.

6h - Check STOCK CONTOURS.
Do not select anything here, it will default to the profile outline.



7

2D ADAPTIVE CONTINUED

7a - Select TAB 4.

7b - The HEIGHTS should look something like this. You can use Selected Contours for the Bottom Height.

7c - Select TAB 5.

7d - PASSES/OPTIMAL LOAD = .1"

7e - Check MULTIPLE DEPTHS.

7f - MAXIMUM ROUGHING = .1"

7g - Check STOCK TO LEAVE.

7h - RADIAL & AXIAL = .02"

7i - Check SMOOTHING.

7j - Check FEED OPTIMIZATION.

7k - Select TAB 6.

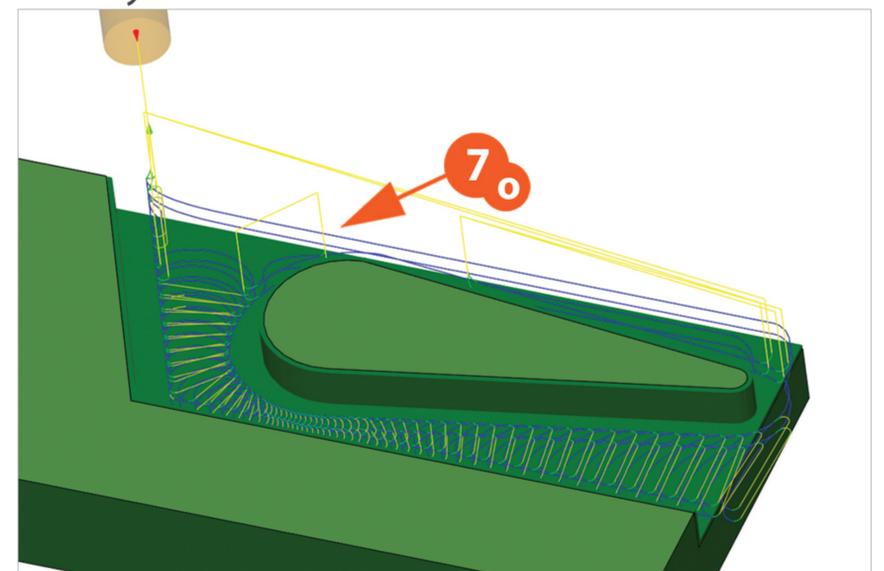
7L - LINKING/RETRACTION POLICY set to Minimum.

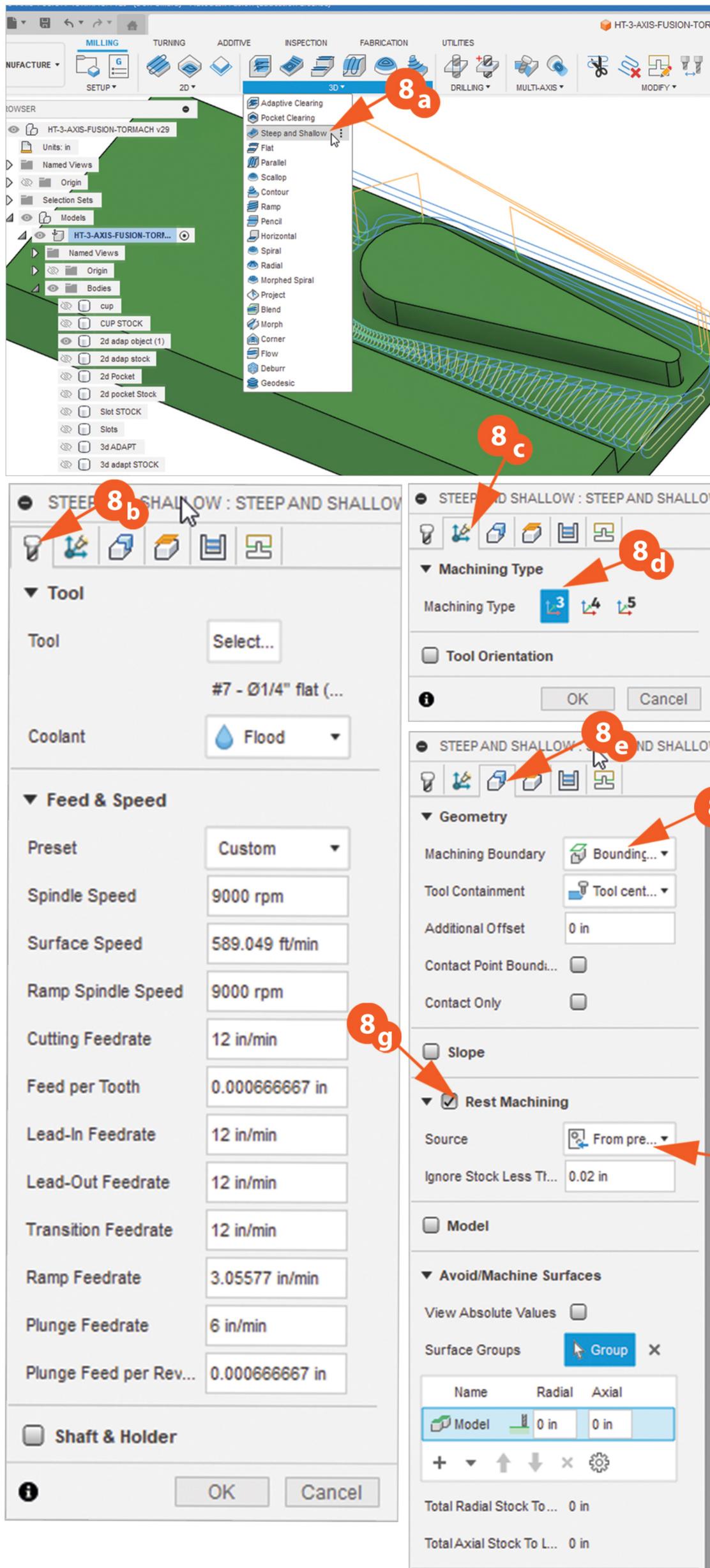
7m - RAMP/RAMP TYPE is Plunge.

7n - Click OK.

7o - Your tool path should look like this.

See the Post Processing Chapter (on page 27) to create your G-Code for the Tormach PCNC440.





8

We will use **Steep & Shallow** to finish our object. We will continue to use the same Setup that was used for the 2D Adaptive operation.

STEEP & SHALLOW - OPERATION #2

8a - Select 3D/STEEP & SHALLOW.

8b - On TAB 1, input the following:

- Tool = #7 (.25" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 12 in/mi
- Plunge Feedrate = 6 in/min
- Ramp Feedrate = 6 in/min

8c - Select TAB 2.

8d - It should be on 3.

8e - Select TAB 3.

8f - MACHINING/GEOMETRY should be on BOUNDING BOX.

8g - Check REST MACHINING.

8h - Select FROM PREVIOUS OPERATION.

The screenshot displays the Fusion 360 'STEEP AND SHALLOW' toolpath settings. The interface is divided into several sections:

- Clearance Height:** From: Retract height, Offset: 0.4 in.
- Retract Height:** From: Stock top, Offset: 0.2 in.
- Top Height:** From: Stock top, Offset: 0 in.
- Bottom Height:** From: Model bottom, Offset: 0.222174 in.
- Passes:** Tolerance: 0.0004 in, Recognize Areas as: Steep an..., Threshold Angle: 30 deg, Overlap Distance: 0.03 in, Order by Depth: , Priority: Top first.
- Steep Passes:** Spiral: , Direction: Climb, Stepdown: 0.01 in, Cusp Height: 0.00707126 in.
- Shallow Passes:** Type: Scallop, Optimize Open Pockt...: , Spiral: , Direction: Climb, Stepper: 0.01 in, Cusp Height: 0.00707126 in, Smooth Offsets: , Remove Cusps at Ju...: .
- Stock to Leave:**
- Fillets:**
- Smoothing:** , Smoothing Mode: Fit arcs.
- Feed Optimization:**
- Linking:** Retraction Policy: Minimum..., High Feedrate Mode: Preserve..., Safe Distance: 0.08 in, Maximum Stay-Down...: 0.5 in.
- Leads & Transitions:** Lead-In Radius: 0.025 in, Vertical Lead-In Radi...: 0.025 in, Lead-Out Radius: 0.025 in, Vertical Lead-Out Ra...: 0.025 in, Transition Type: Smooth.

Callouts 9a through 9j point to specific UI elements: 9a (Tab 4), 9b (HEIGHTS), 9c (Tab 5), 9d (Stepdown), 9e (Stepover), 9f (Smoothing), 9g (Tab 6), 9h (Retraction Policy), 9i (OK button), and 9j (Toolpath visualization).

9

STEEP & SHALLOW CONTINUED

9a - Select TAB 4.

9b - The HEIGHTS should be close to what is shown here.

9c - Select TAB 5.

9d - STEEP PASSES/STEPDOWN = .01"

9e - SHALLOW PASSES/STEPOVER = .01"

9f - Check SMOOTHING.

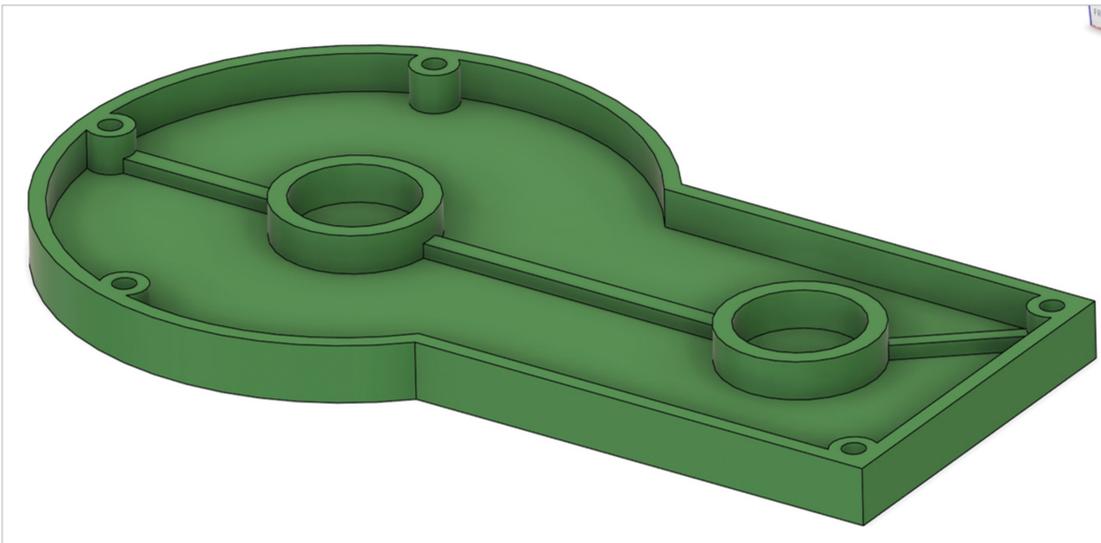
9g - Select TAB 6.

9h - RETRACTION POLICY set to Minimum.

9i - Click OK.

9j - Your tool path should look like this.

See the Post Processing Chapter (on page 27) to create your G-Code for the Tormach PCNC440.



10

We will use a number of operations to create our object: Face, 2D Pocket Roughing, 2D Pocket Finishing, 2D Contour and Hole Recognition.

You will need to create a new setup. Please see page 1.

FACE - OPERATION #1

Face is using a large tool bit or Hog Shear to machine off a layer of the stock to assure an even and flat surface.

10a - Select 2D/FACE.

10b - In TAB 1, input the following settings:

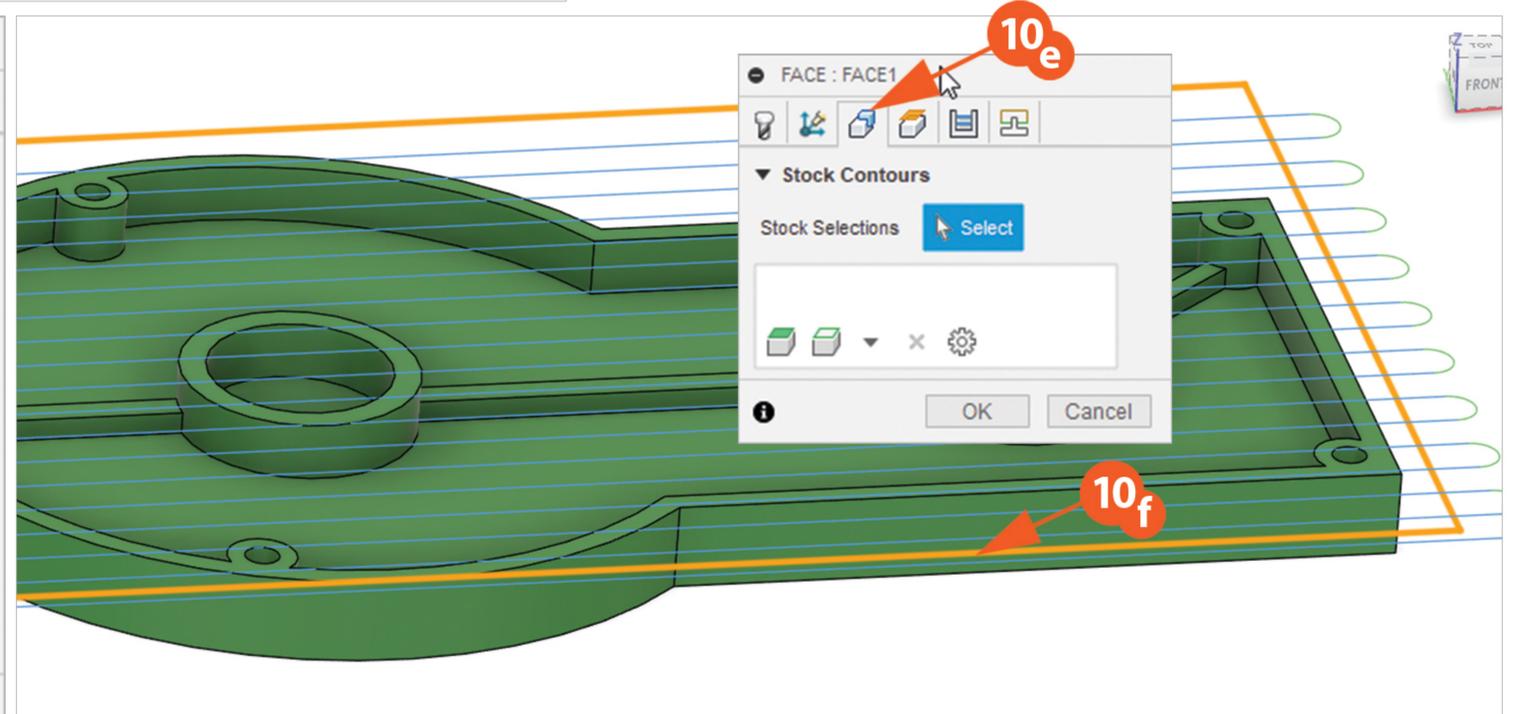
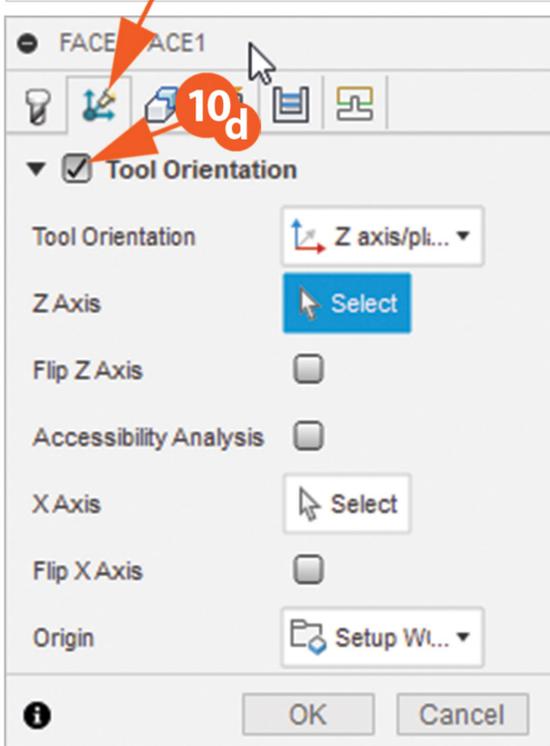
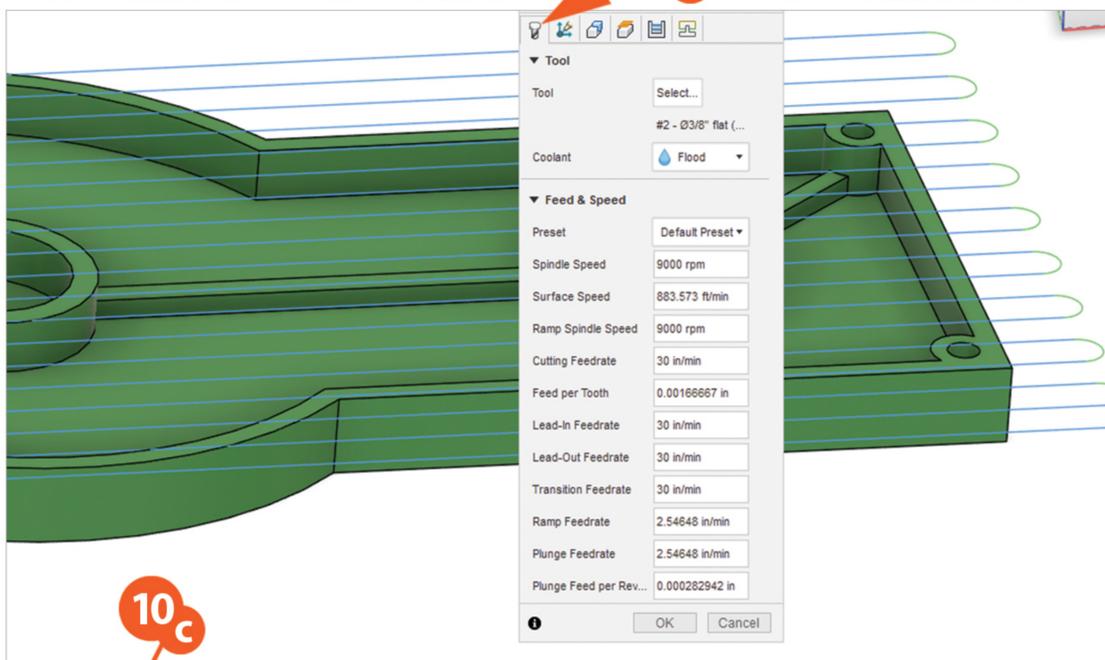
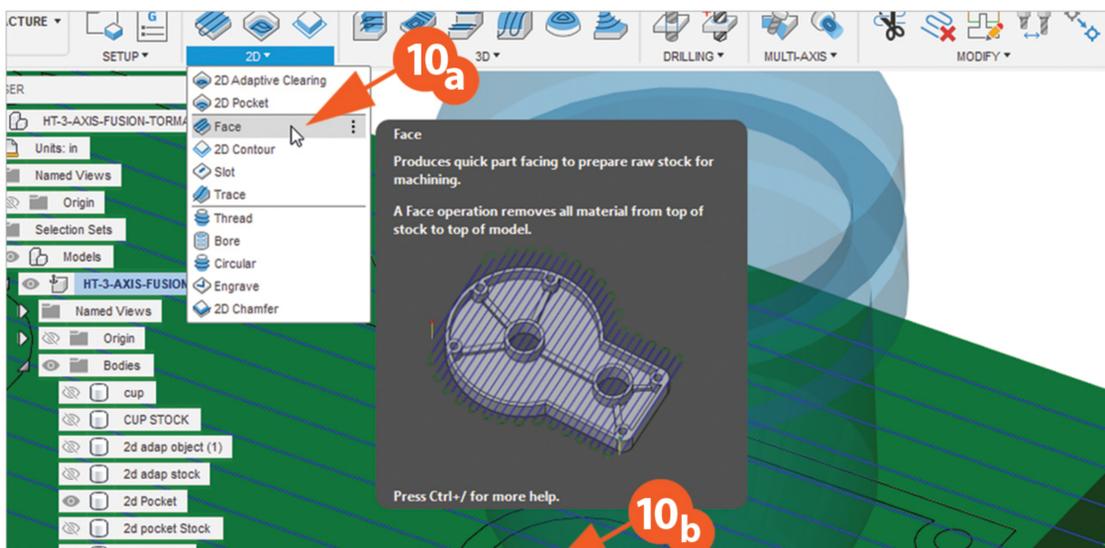
- Tool = #2 (3/8" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 30 in/min
- Plunge Feedrate = 15 in/min
- Ramp Feedrate = 15 in/min

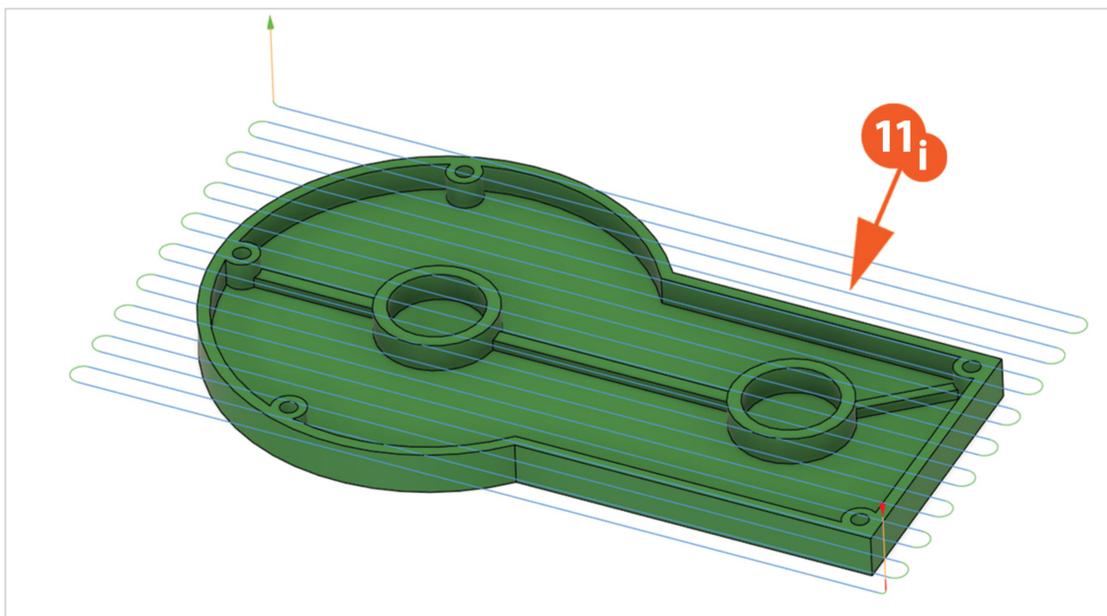
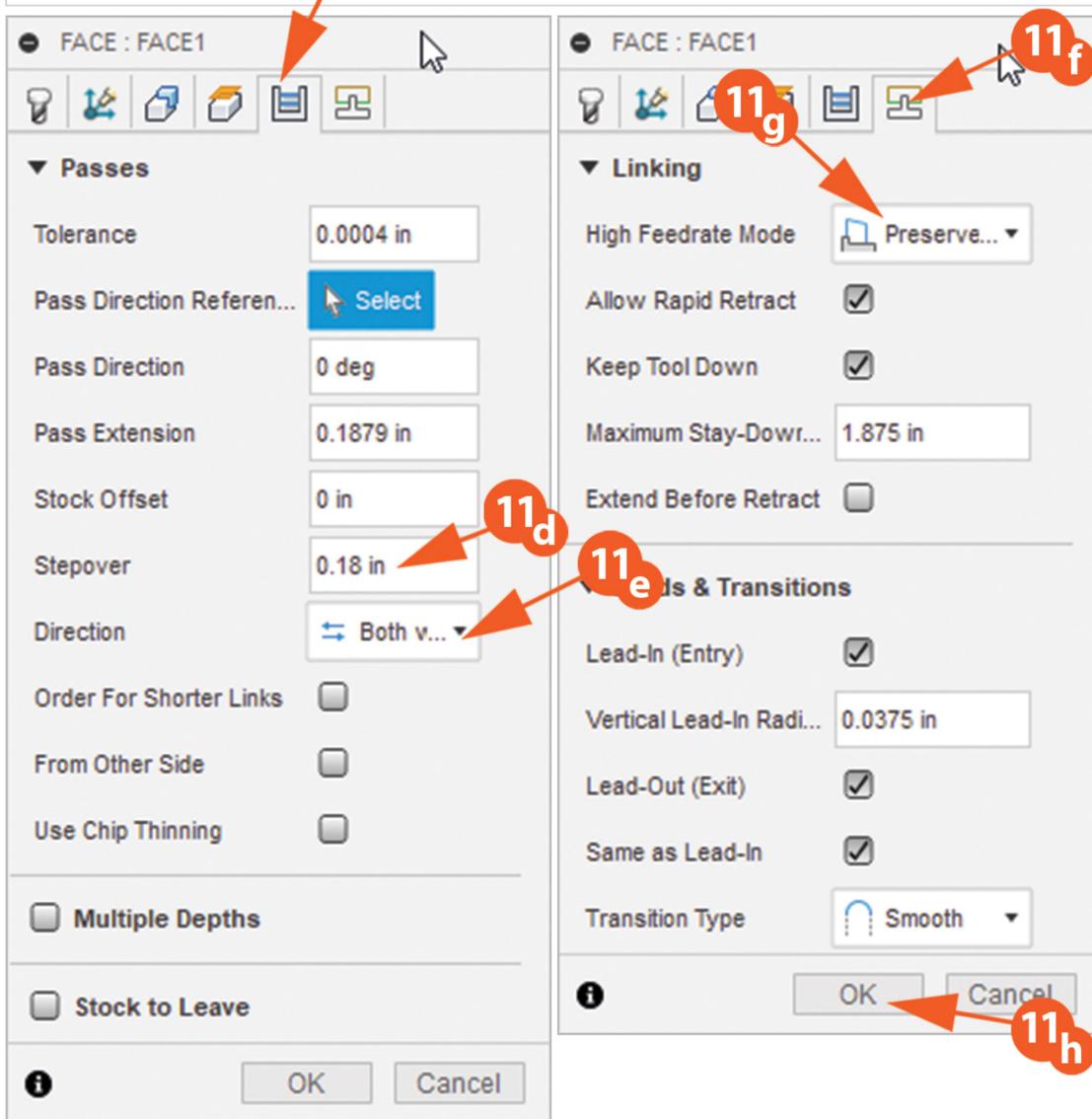
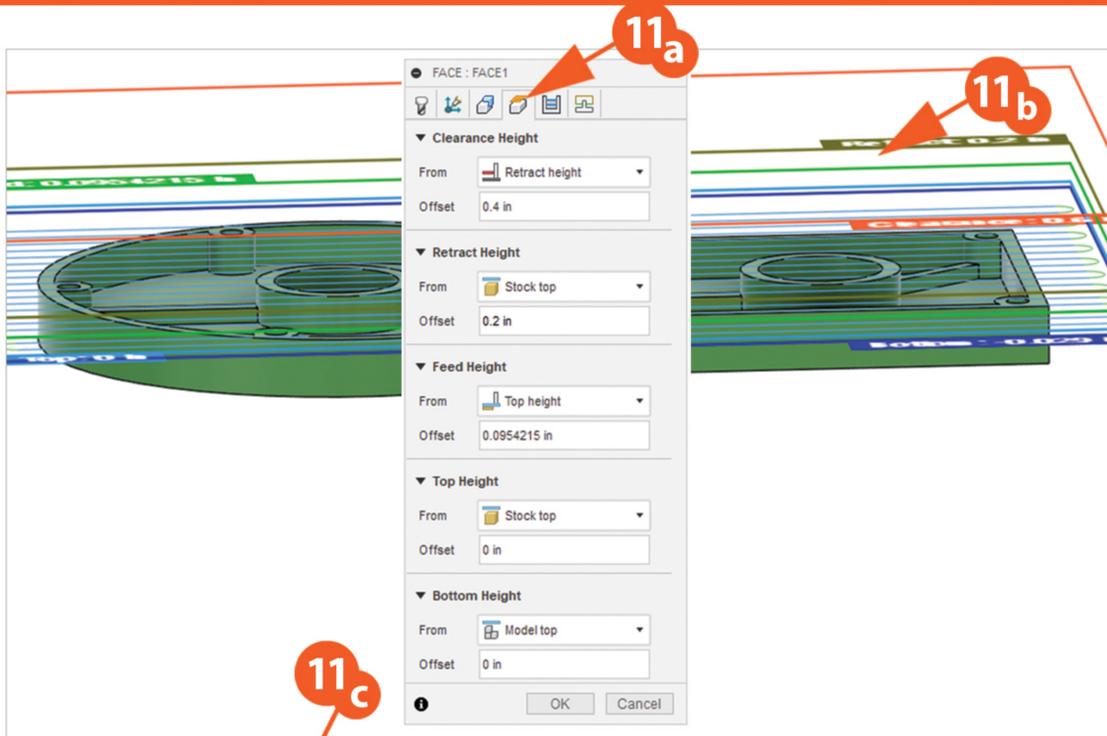
10c - Select TAB 2.

10d - Check TOOL ORIENTATION.

10e - Select TAB 3.

10f - Do not select anything, it will default to the Stock outline.





11

FACE CONTINUED

11a - Select TAB 4.

11b - Set the HEIGHTS as shown here.

11c - Select TAB 5.

11d - Set the STEPOVER to .18"

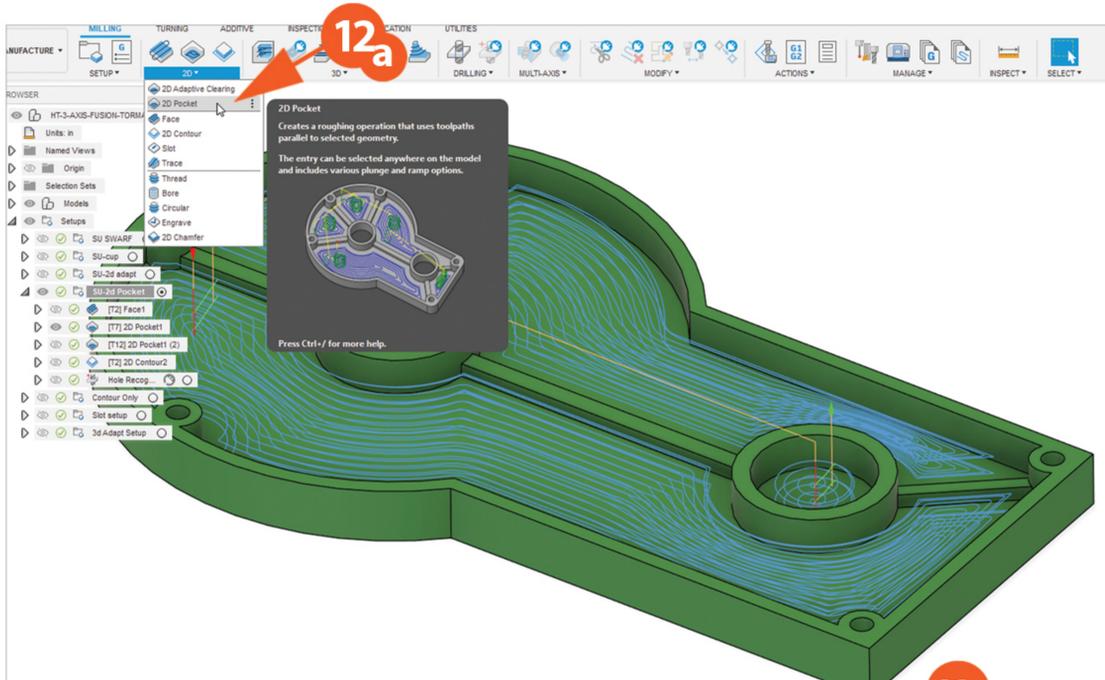
11e - Select BOTH DIRECTIONS.

11f - Select TAB 6.

11g - Select PRESERVE RAPID.

11h - Click OK.

11i - Your machining path should look like this.



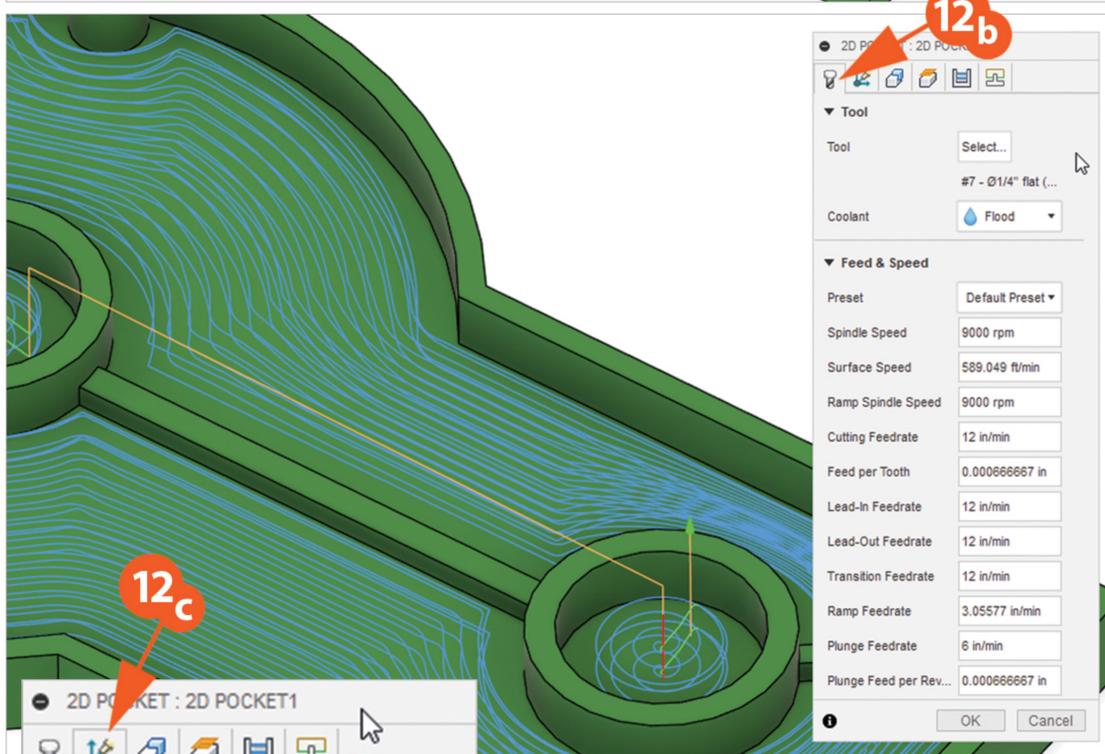
12

2D POCKET ROUGHING - OPERATION #2
 2D Pocket is used to quickly rough out a pocket area.

12a - Select 2D/2D POCKET.

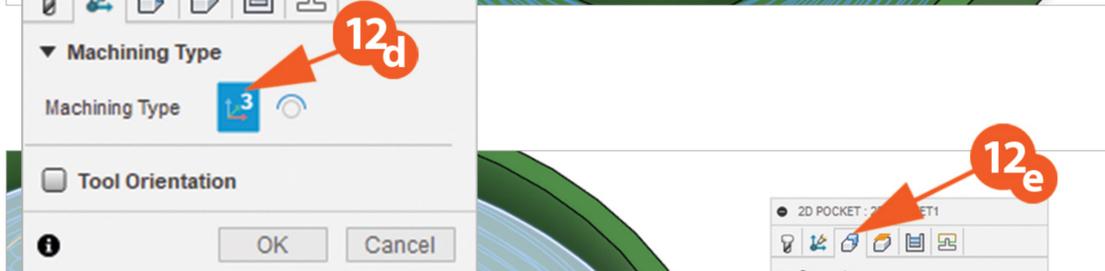
12b - In TAB 1, input the following settings:

- Tool = #7 (.25" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 12 in/mi
- Plunge Feedrate = 6 in/min
- Ramp Feedrate = 6 in/min



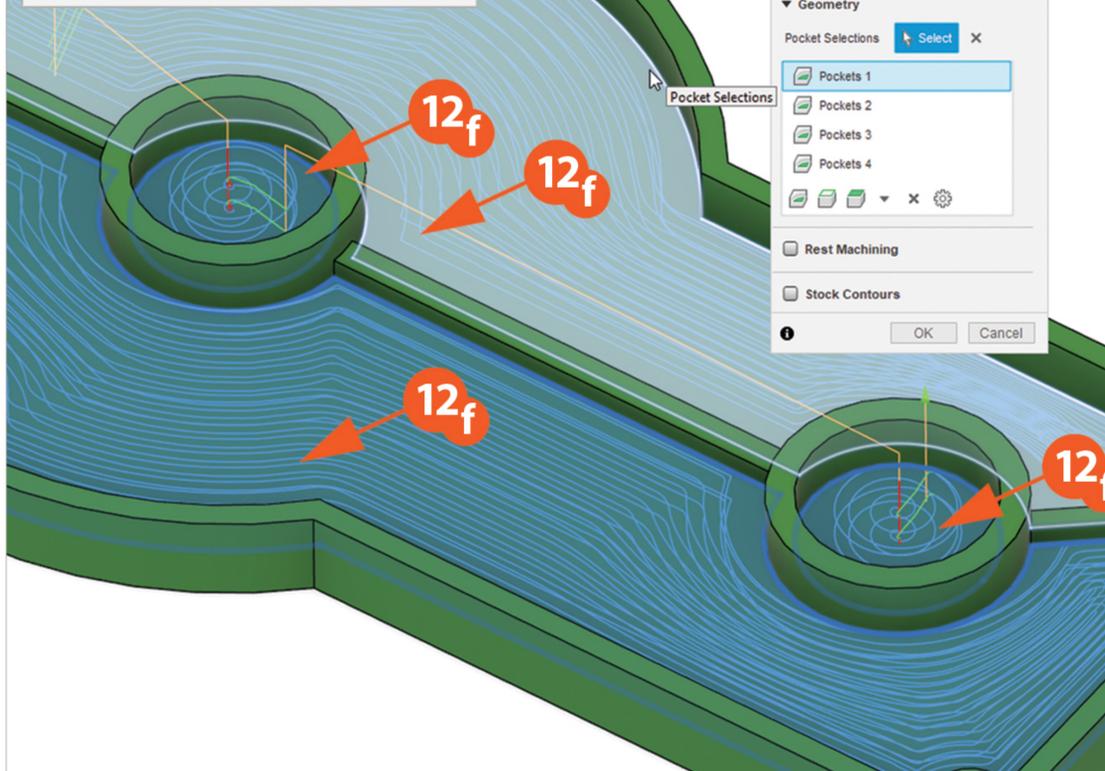
12c - Select TAB 2.

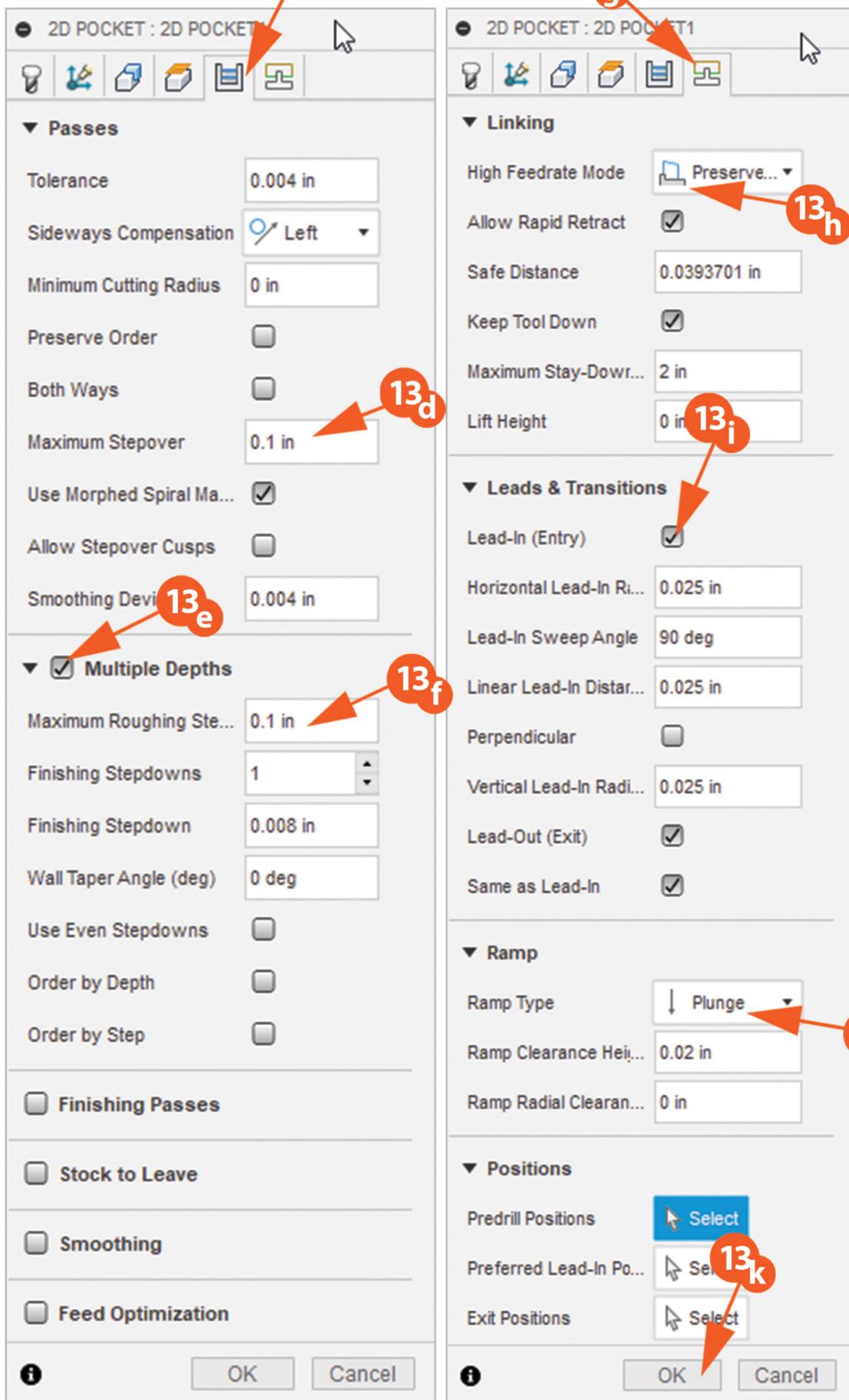
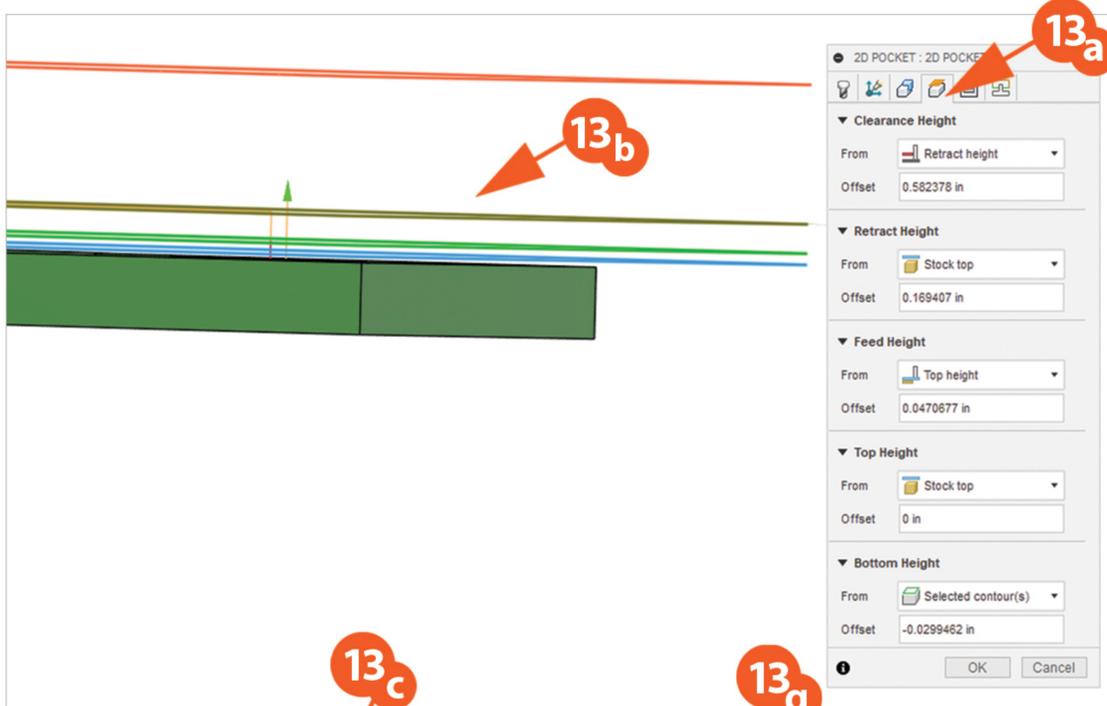
12d - It should be at 3.



12e - Select TAB 3.

12f - Select all of the Bottom Faces of the Pockets.





13

2D POCKET ROUGHING - CONTINUED

13a - Select TAB 4.

13b - Set the HEIGHTS as shown.

13c - Select TAB 5.

13d - Set MAXIMUM STEPOVER to .1"

13e - Check MULTIPLE DEPTHS.

13f - MAXIMUM ROUGHING STEPDOWN to .1"

13g - Select TAB 6.

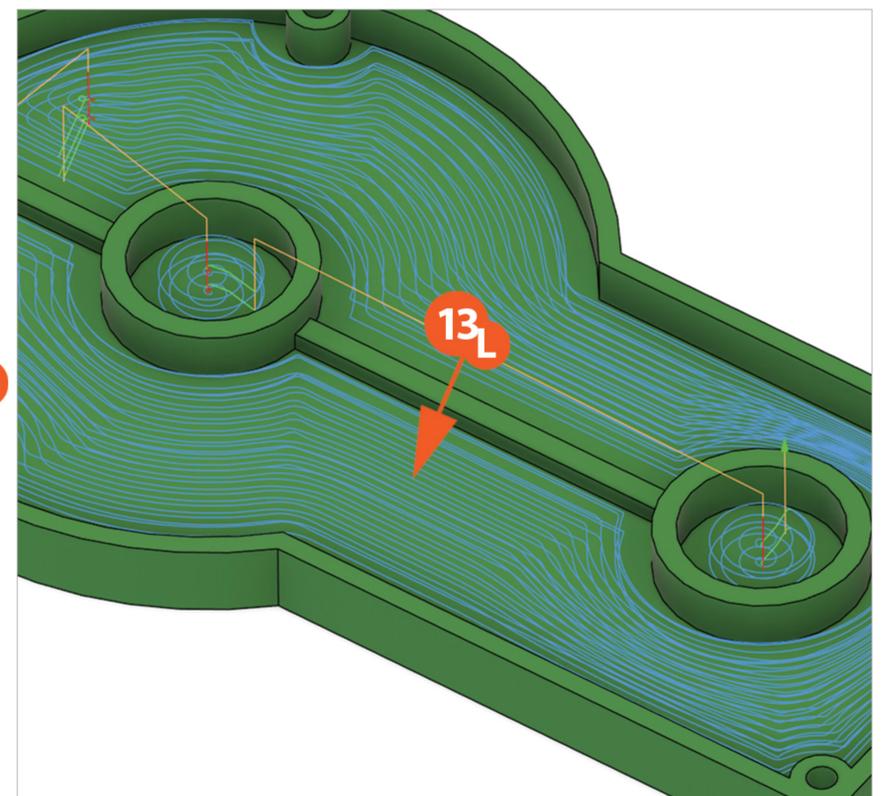
13h - PRESERVE RAPID.

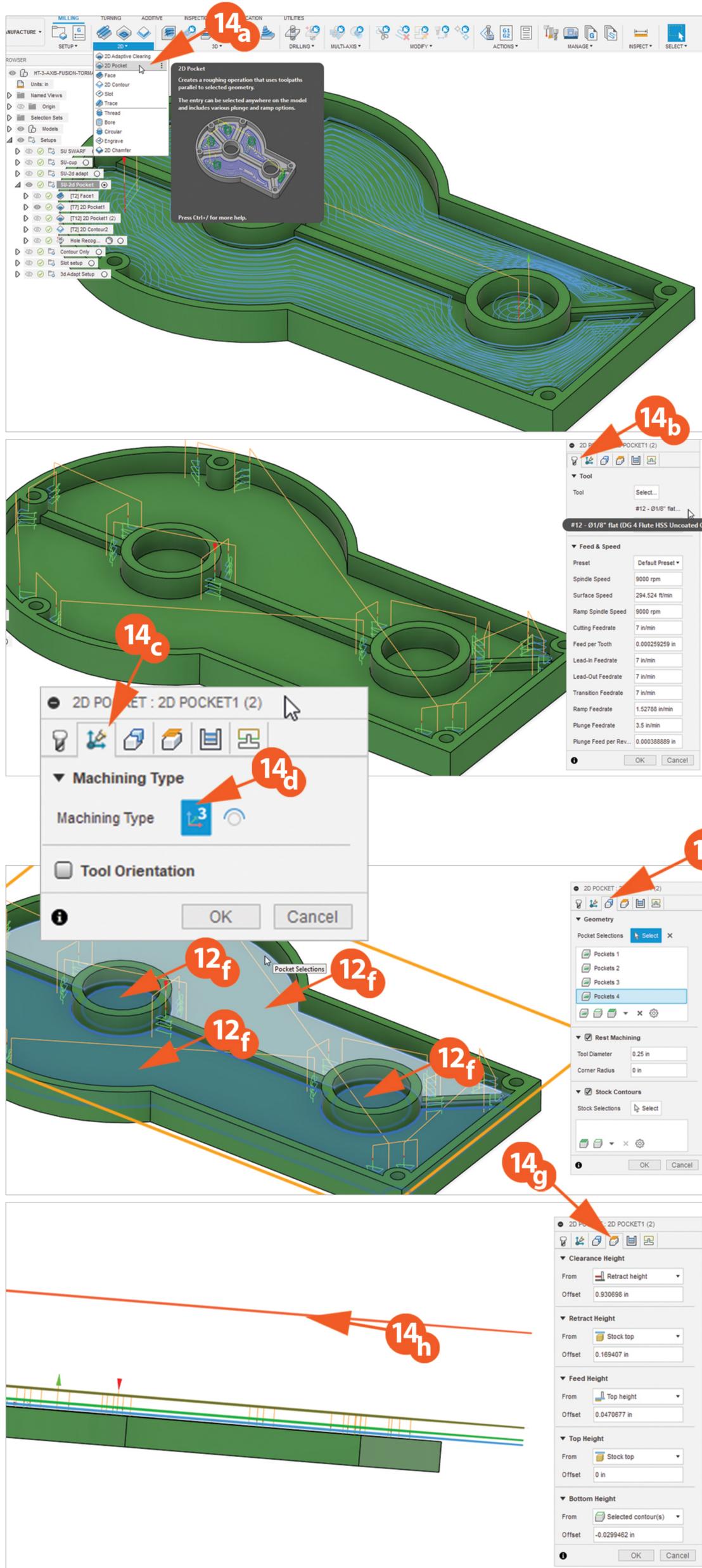
13i - Check LEAD-IN.

13j - RAMP is PLUNGE.

13k - Click OK.

13L - The tool path's should look like this.





14

2D POCKET FINISHING - OPERATION #3
This 2D Pocket is using a smaller tool bit to get into the corners with a smaller radius.

14a - Select 2D/2D POCKET.

14b - In TAB 1, input the following settings:

- Tool = #12 (.125" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 7 in/mi
- Plunge Feedrate = 3.5 in/min
- Ramp Feedrate = 3.5 in/min

14c - Select TAB 2.

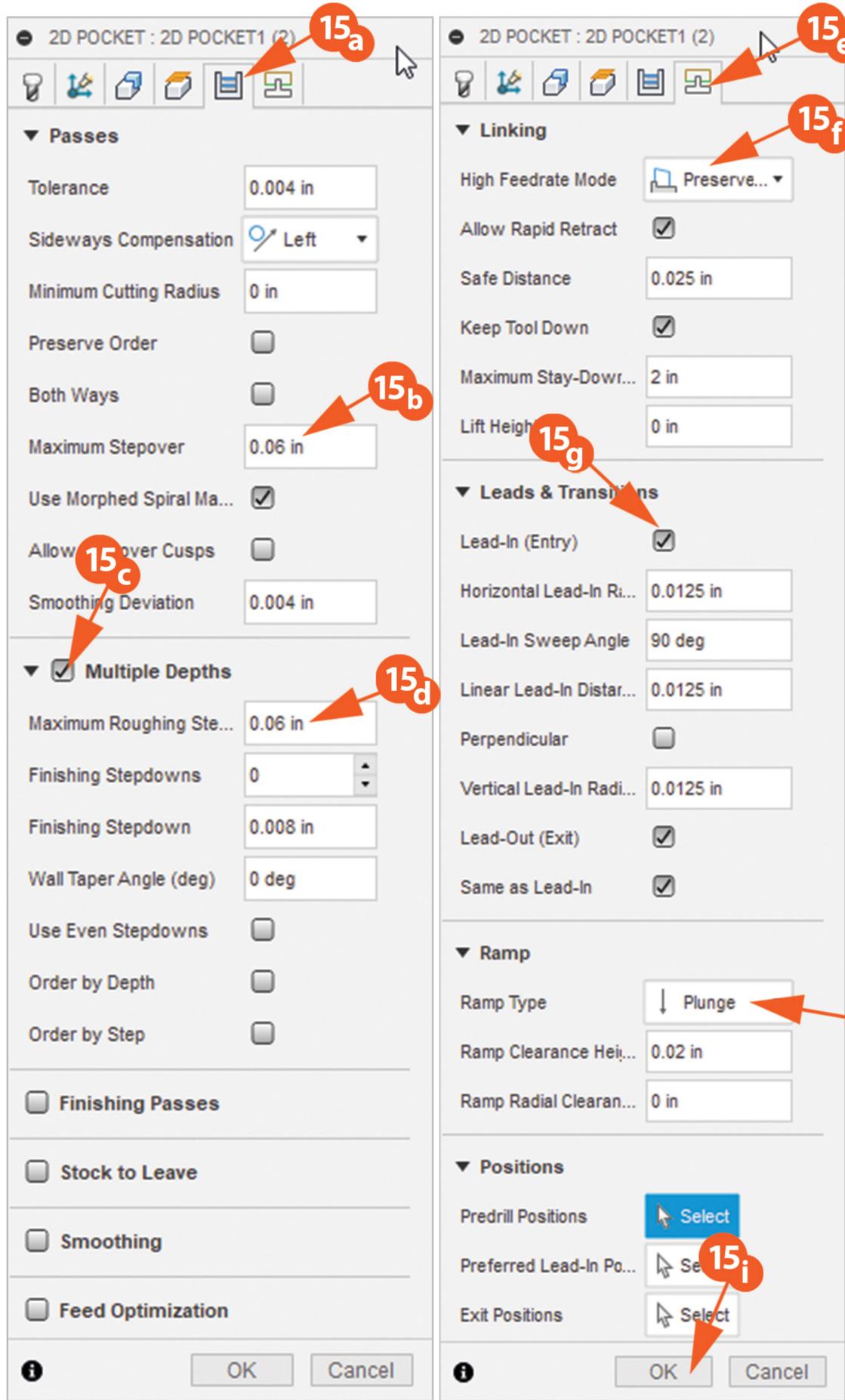
14d - It should be at 3.

14e - Select TAB 3.

14f - Select all of the Bottom Faces of the Pockets.

14g - Select TAB 4.

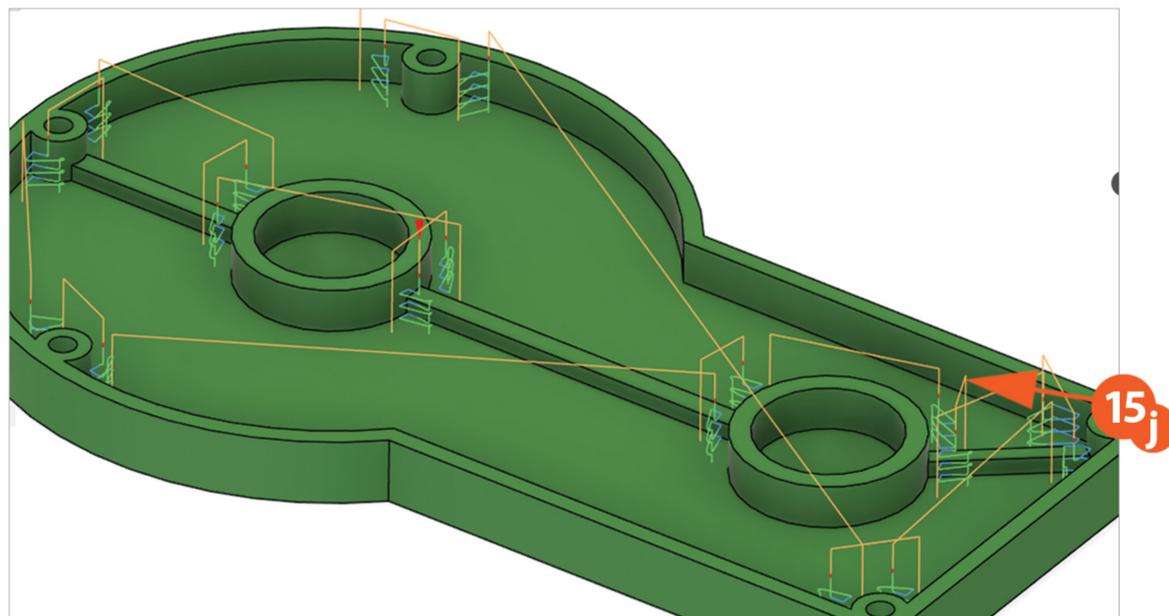
14h - Your HEIGHTS should be as shown here.

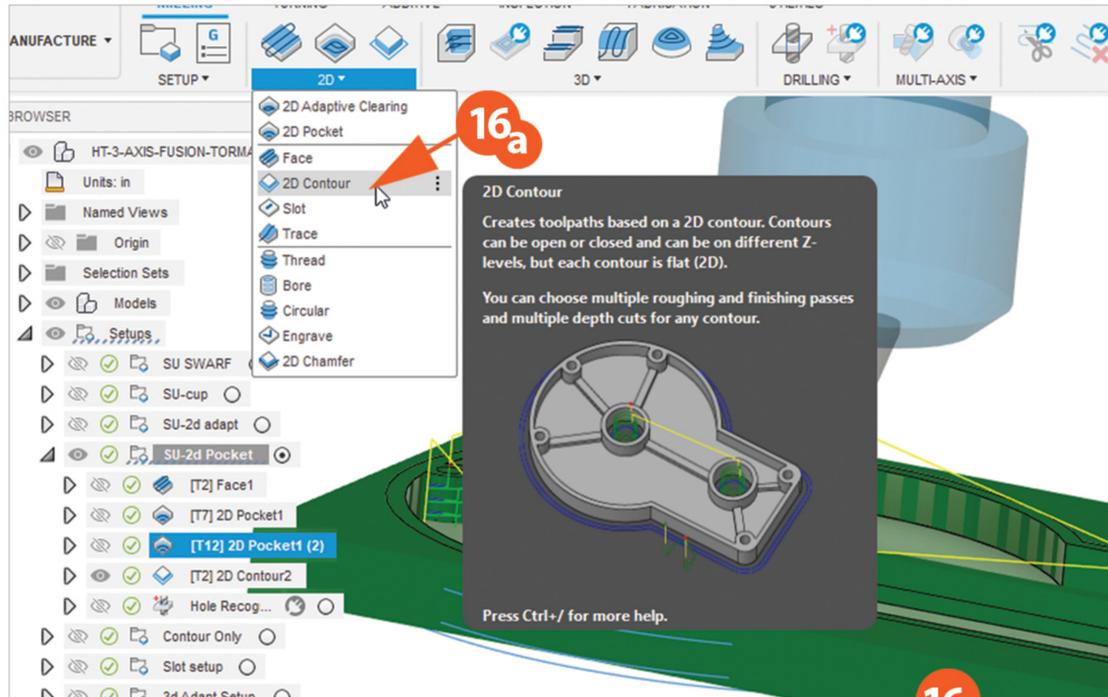


15

2D POCKET FINISHING CONTINUED

- 15a - Select TAB 5.
- 15b - MAXIMUM STEPOVER set to .06"
- 15c - Check MULTIPLE DEPTHS.
- 15d - MAXIMUM ROUGHING STEPDOWN is .06"
- 15e - Select TAB 6.
- 15f - Select PRESERVE RAPID.
- 15g - Check LEAD-IN.
- 15h - RAMP TYPE/PLUNGE.
- 15i - Click OK.
- 15j - Your Tool Paths should look like this.



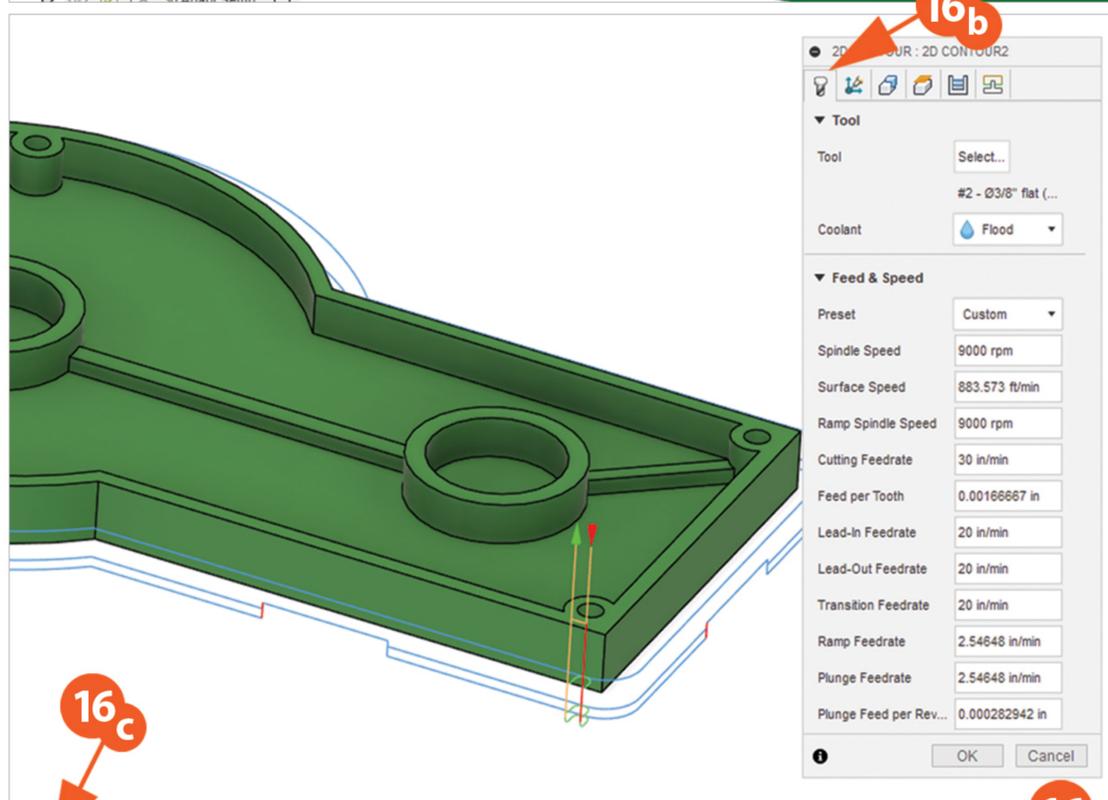


16

2D CONTOUR - OPERATION #4

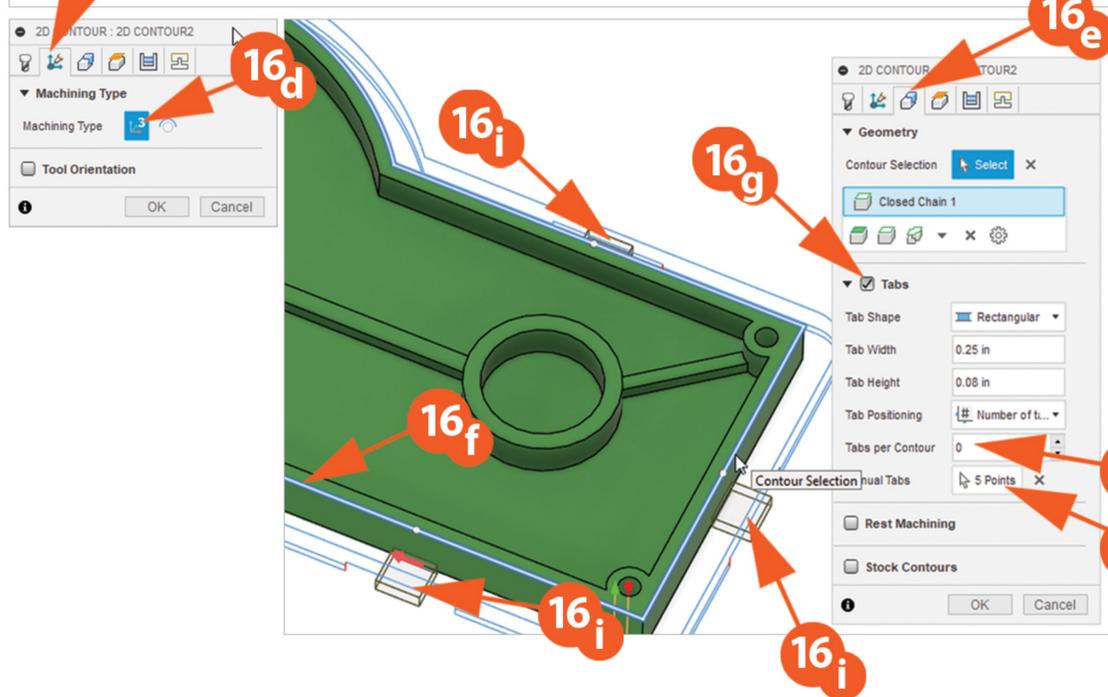
2D Contour is used to cut out the profile shape of the object.

16a - Select 2D/2D Contour.



16b - In TAB 1, input the following settings:

- Tool = #2 (3/8" FEM)
- Coolant = Flood
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 30 in/min
- Plunge Feedrate = 15 in/min
- Ramp Feedrate = 15 in/min



16c - Select TAB 2.

16d - It should be at 3.

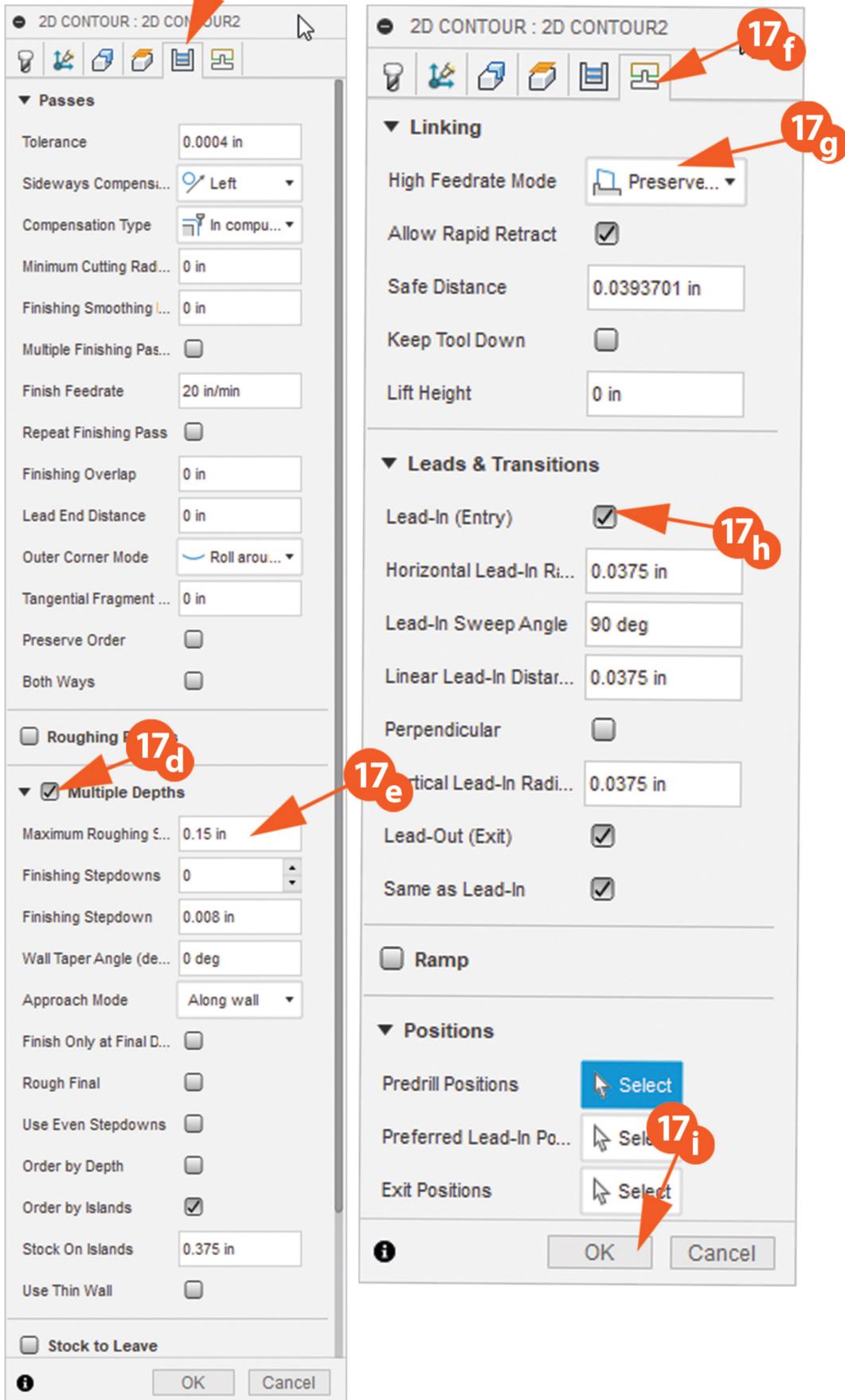
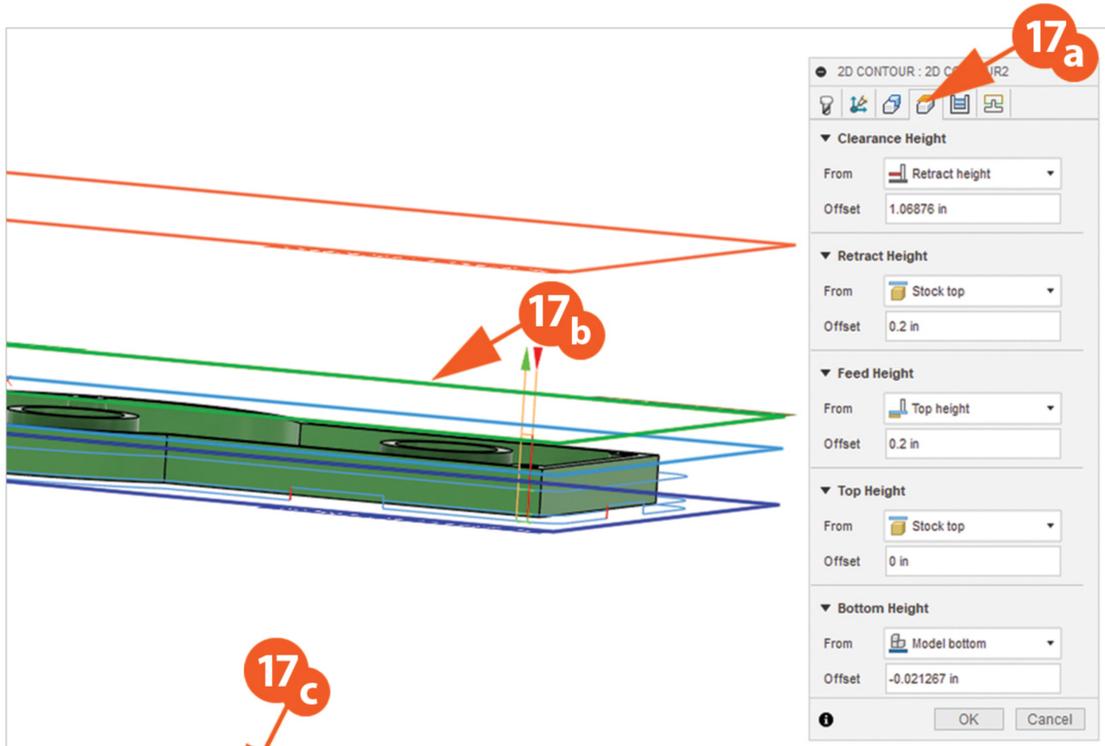
16e - Select TAB 3.

16f - Select the outer contour edge of the object.

16g - Check TABS.

16h - In TABS/TABS PER CONTOUR, enter 0.

16i - Use MANUAL TABS, select once on each side at straight sections of the shape.



17

2D CONTOUR CONTINUED

17a - Select TAB 4.

17b - Set the HEIGHTS as shown.

17c - Select TAB 5.

17d - Check MULTIPLE DEPTHS.

17e - MAXIMUM ROUGHING STEPDOWN to .15"

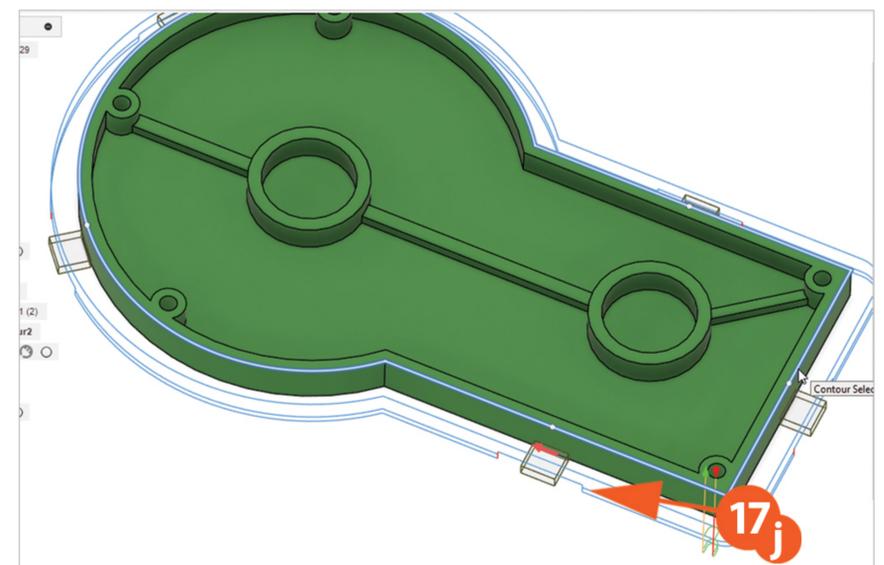
17f - Select TAB 6.

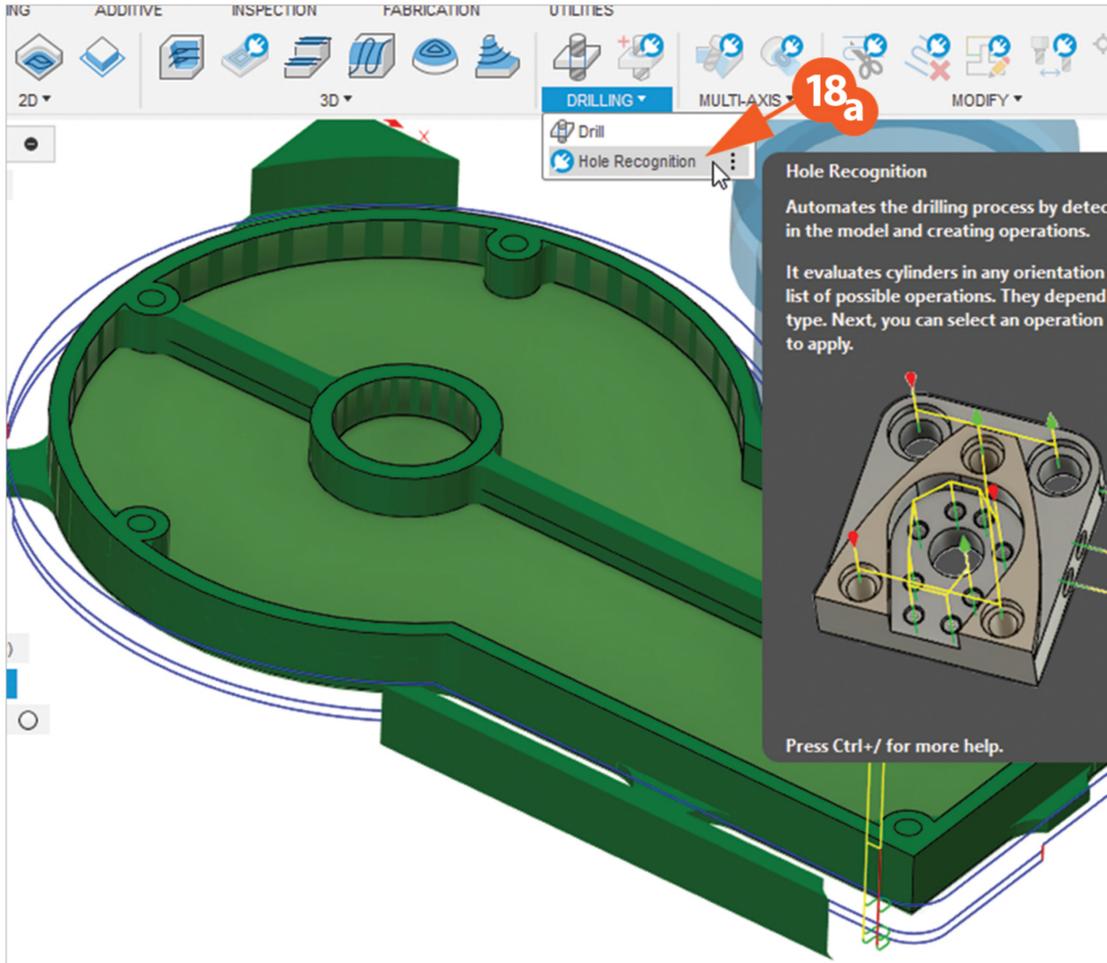
17g - PRESERVE RAPID.

17h - Check LEAD-IN.

17i - Click OK.

17j - The tool path's should look like this.





18

HOLE RECOGNITION - OPERATION #5

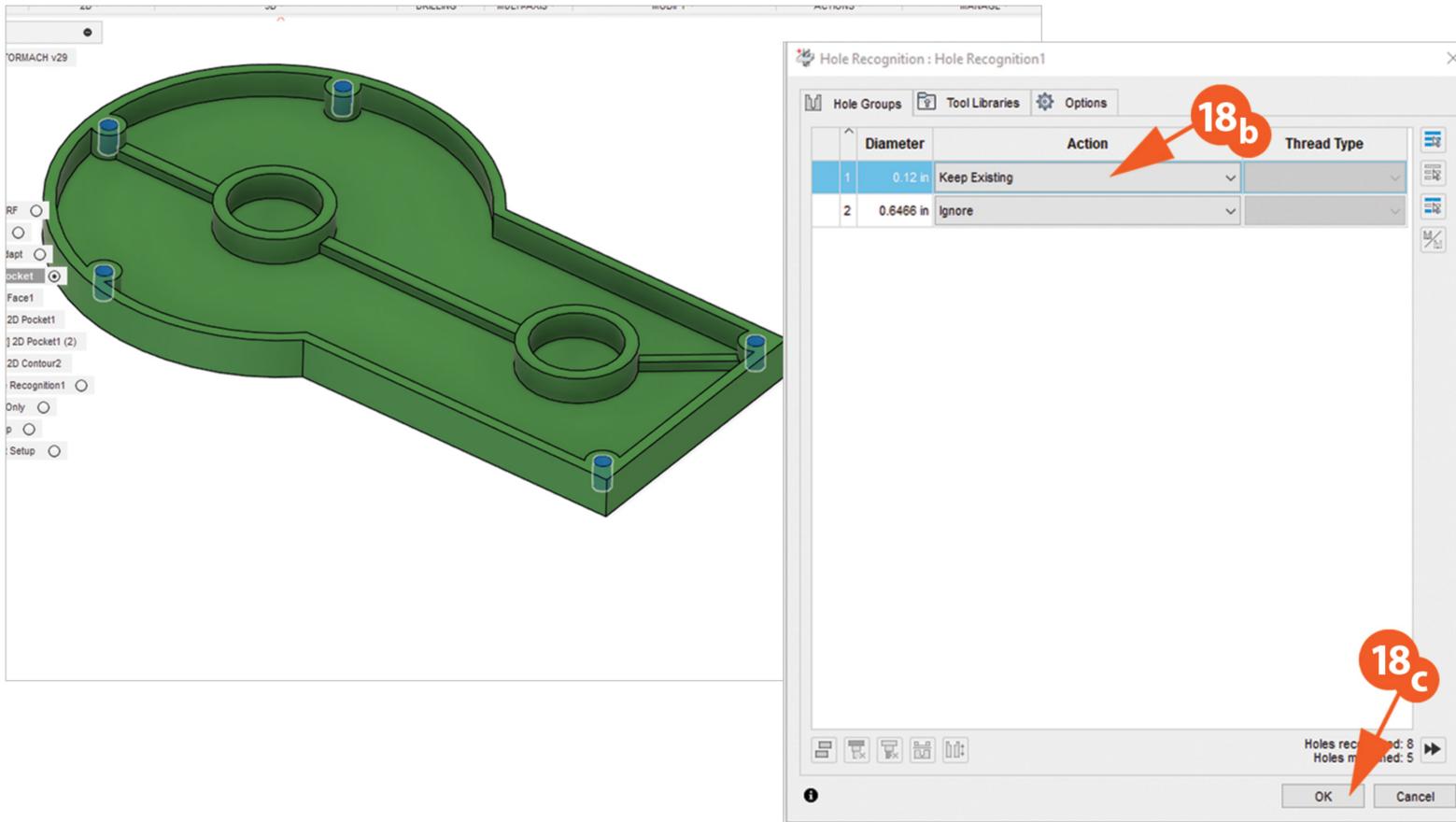
Hole recognition is used for same size multiple holes using a tool bit the correct diameter of the holes.

18a - Select UTILITIES/DRILLING/HOLE RECOGNITION.

18b - From the pop-up menu you will see any holes the program has found. In this example it found the 5 holes at .12" diameter. You will need to use a .12" drill bit for this operation.

It also found the two larger holes/pockets, but it will IGNORE this.

18c - Click OK.



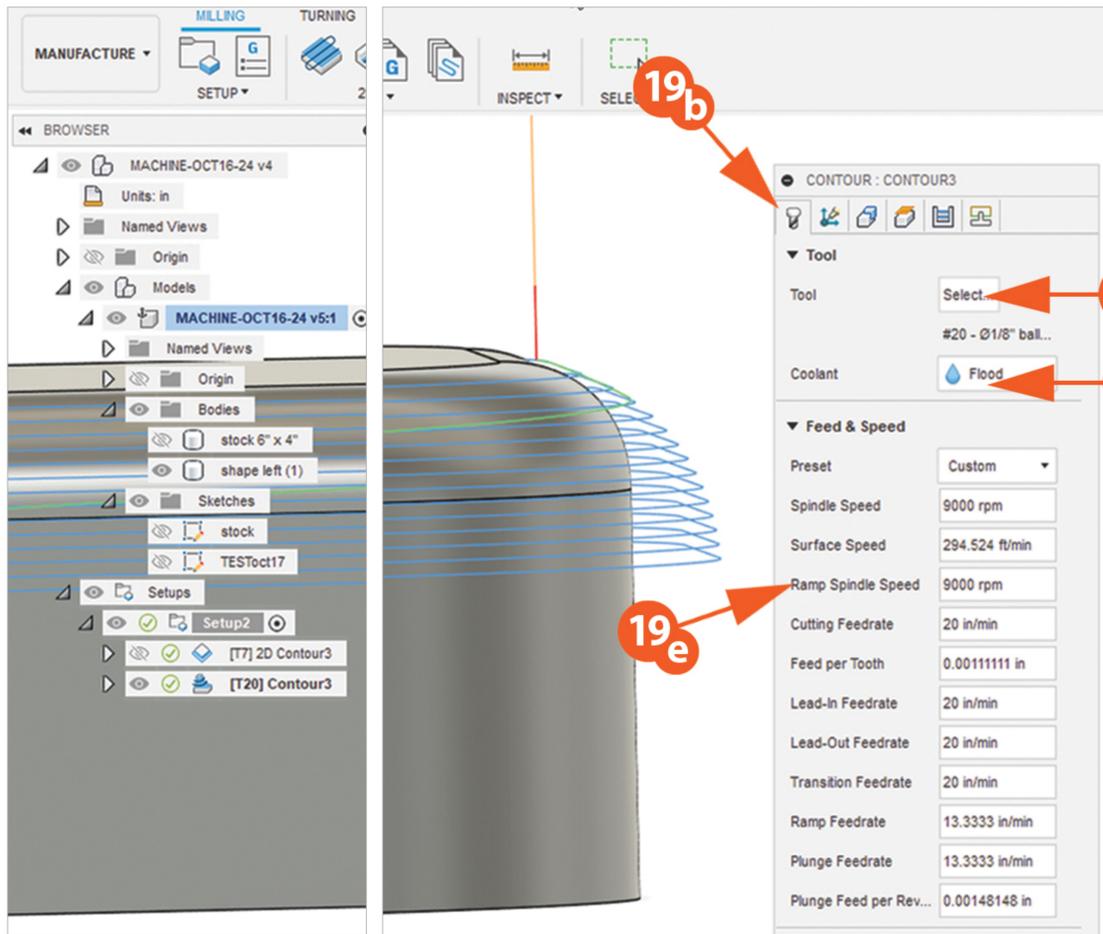
To create the G-Code for all 5 operations, go to the Post Processing Chapter on page 27.



19

For a curved or fillet edge we will use the 3D Contour operation.

If this is a new object, you will need to create a new setup. Please see page 1.



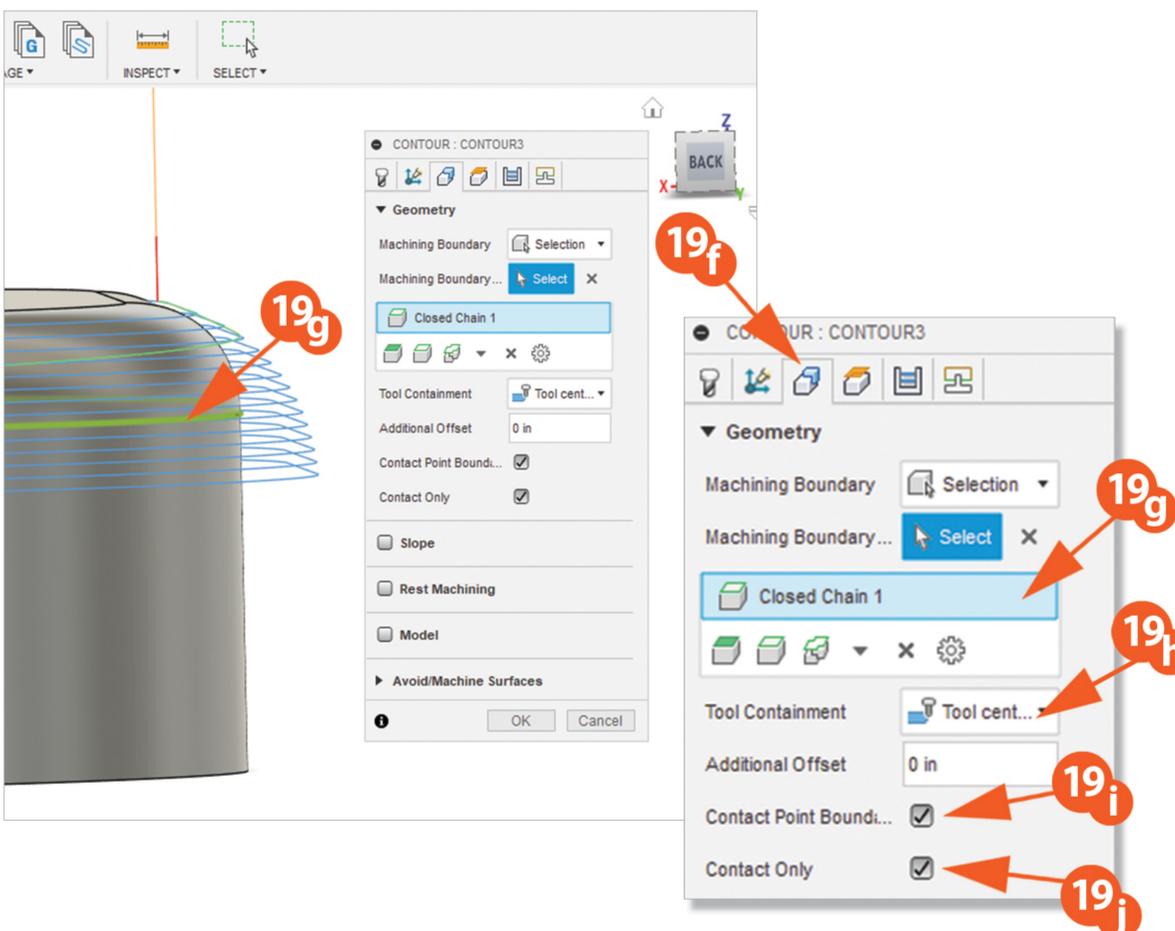
19a - Select **3D OPERATION/CONTOUR**.

19b - Select TAB 1.

19c - Select tool #20 (1/8" REM).

19d - COOLANT = FLOOD.

19e - Spindle speed from 9,000 rpm.



19f - Select TAB 3, Geometry.

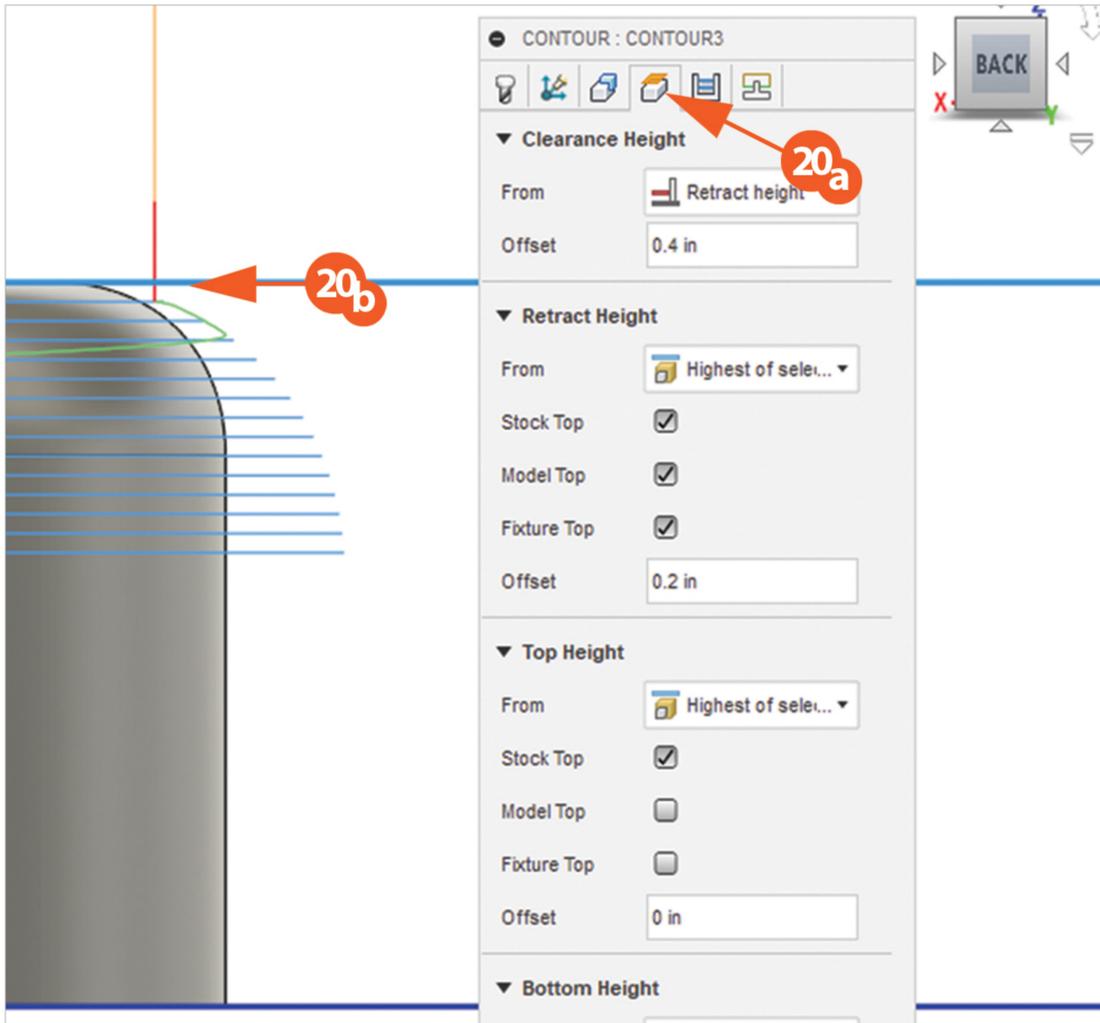
19g - Select the Bottom line of the fillet for your chain.

19h - Tool Containment = Tool Center Boundary.

19i - Check the Contact Point Boundary.

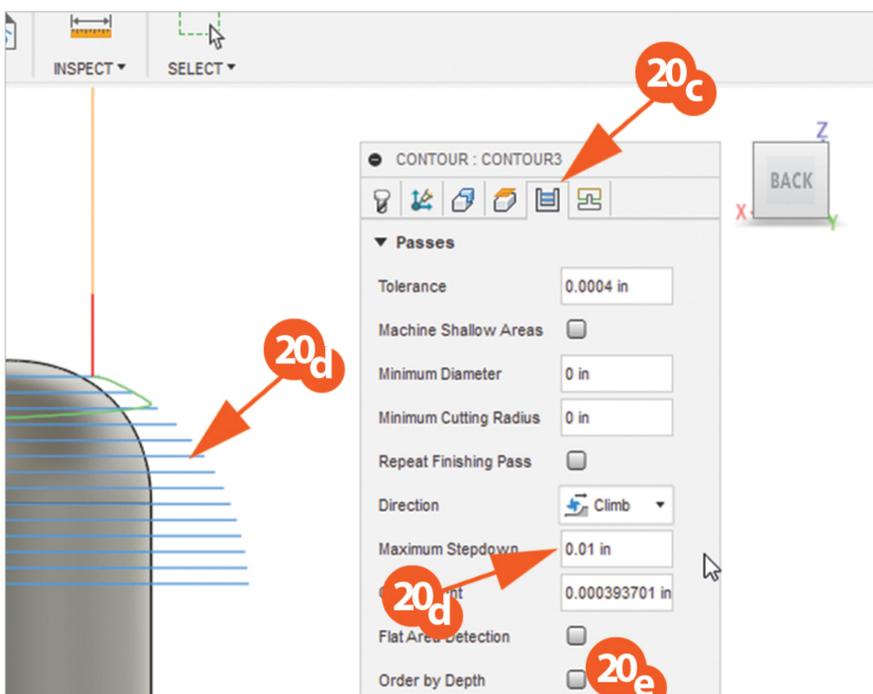
19j - Check the Contact Only.

20



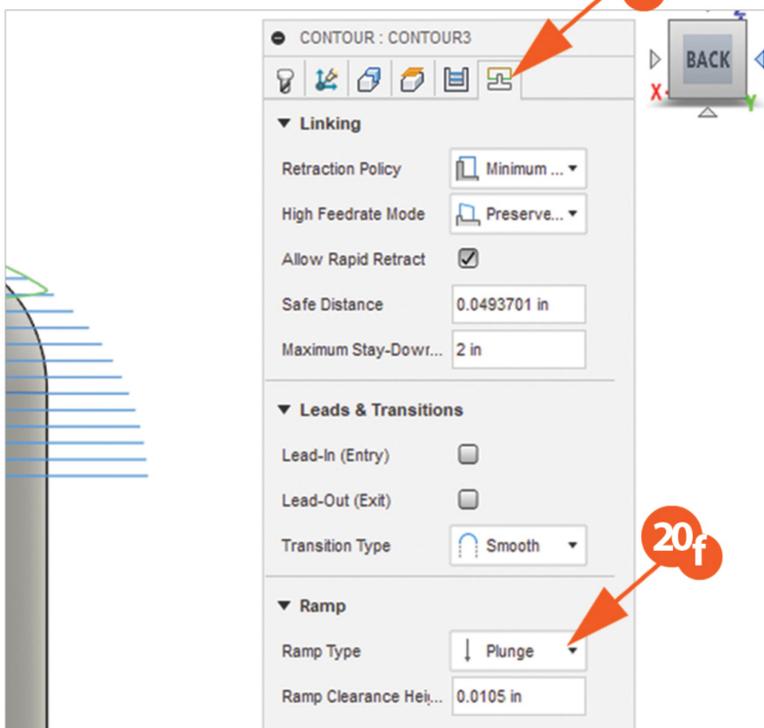
20a - Select TAB 4, Clearance.

20b - Top height should be top of your object and or the fillet.



20c - Select TAB 5, Passes.

20d - Stepdown should be very small to create a smooth fillet that also goes to the top properly. Play with different settings here and preview the results. We are using a .01" Stepdown.



20e - Select TAB 6, Ramp.

20f - Ramp Type = Plunge.

20g - Click OK.

To create the G-Code for all 5 operations, go to the Post Processing Chapter on page 27.

21

SLOT

The SLOT operation is used when you have a slot to be machined at the **exact size of a specific tool bit**.

You will need to create a new setup. Please see page 1.

21a - Select 2D/SLOT.

21b - On TAB 1 select the following:

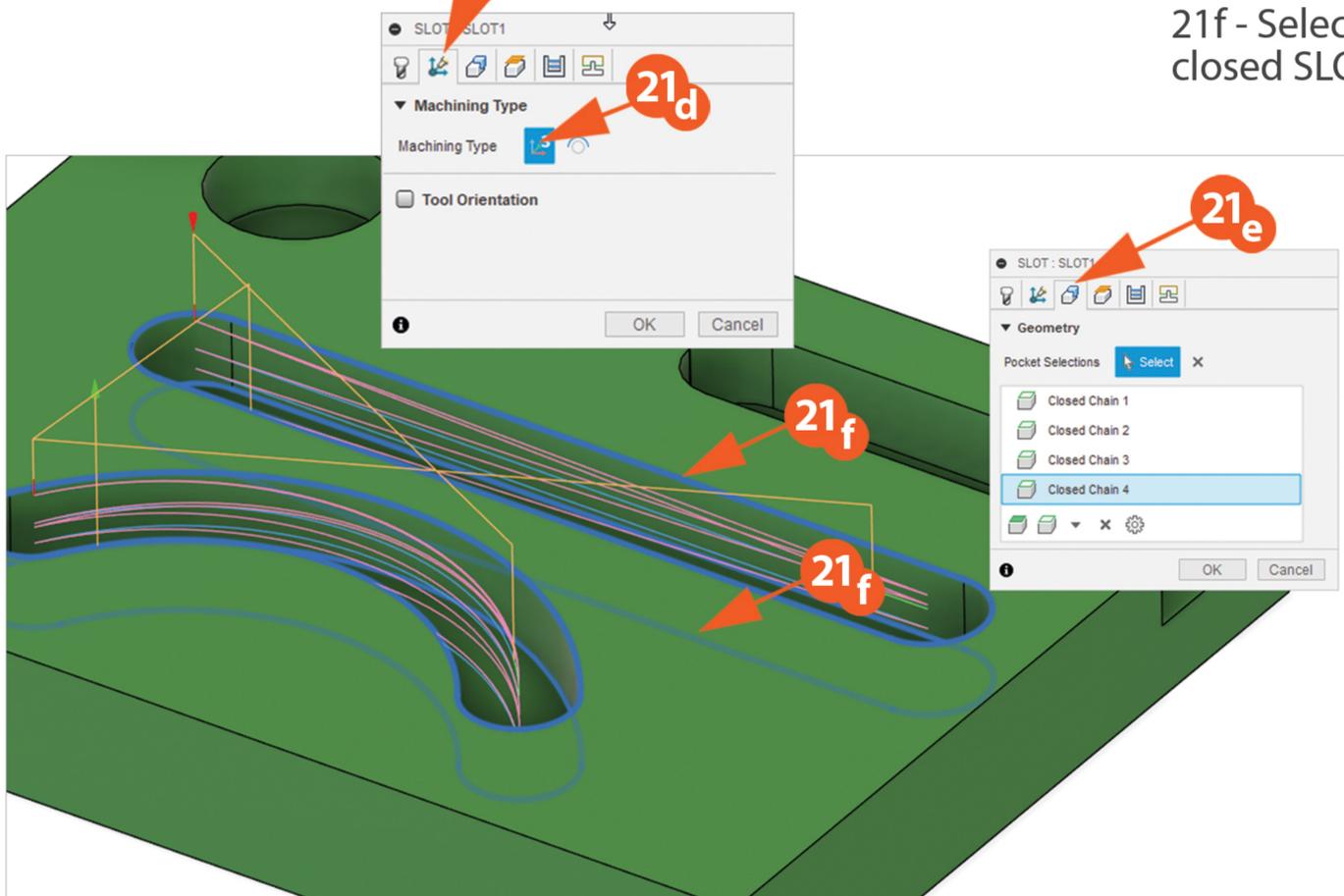
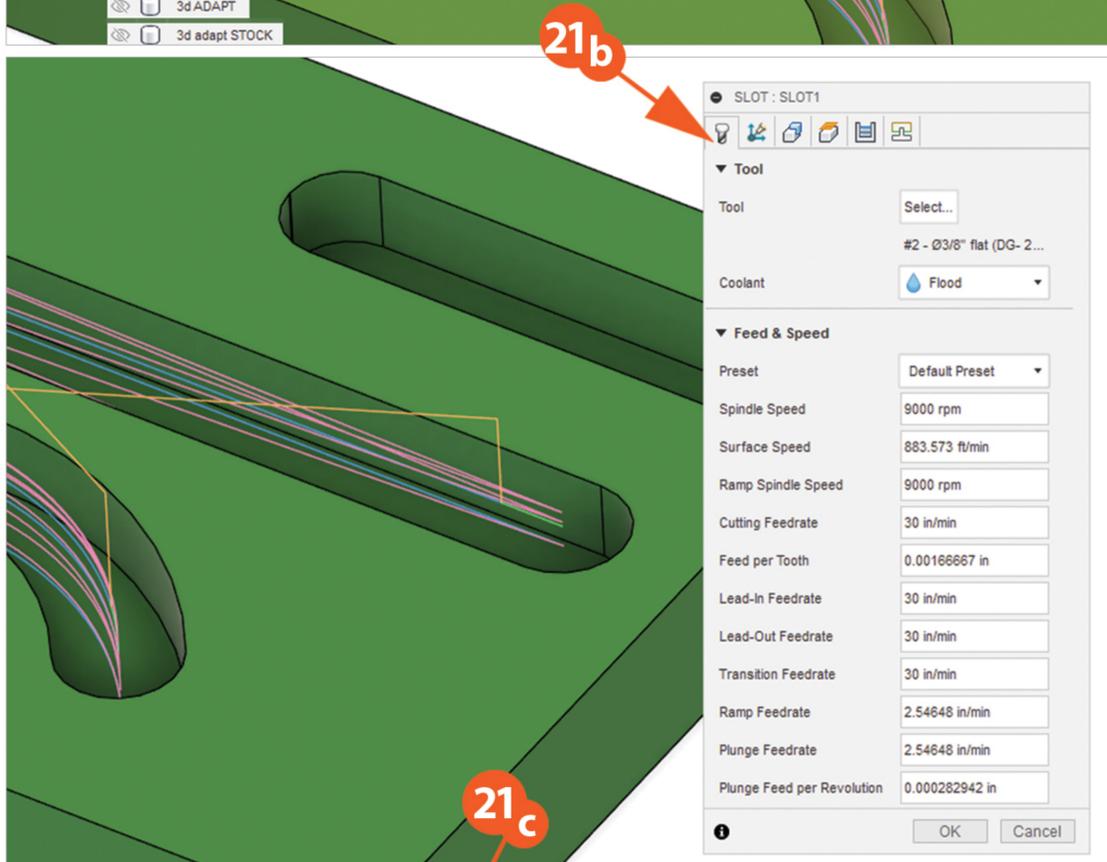
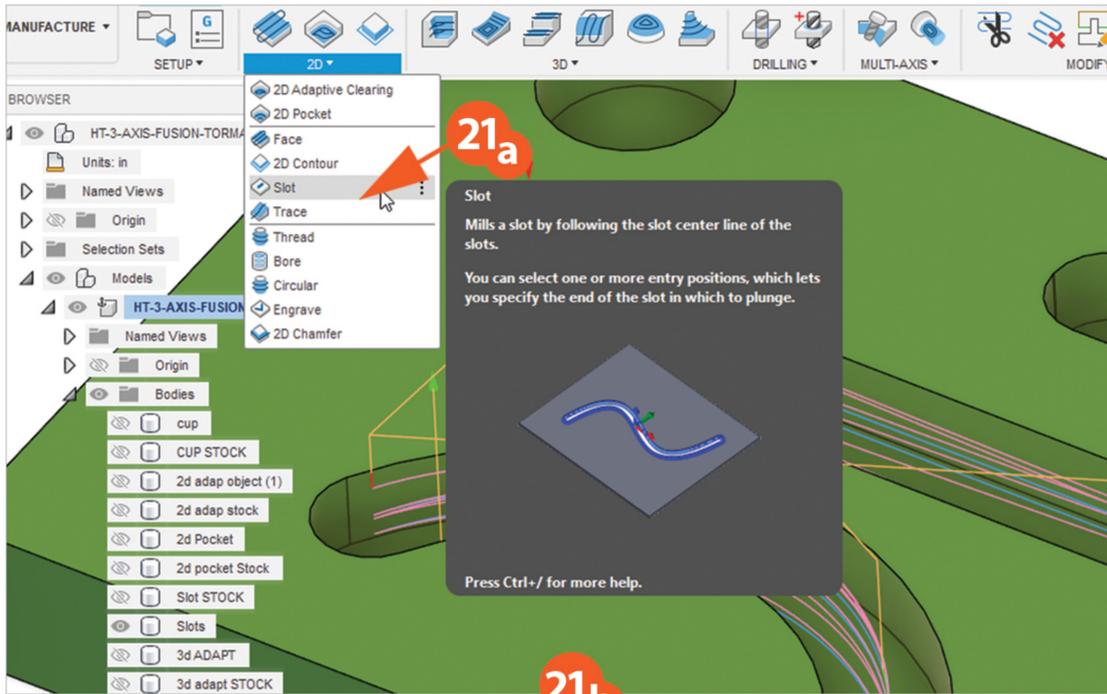
- Tool = #2 (3/8" FEM)
- Coolant = FLOOD
- Spindle Speed = 9000 rpm
- Cutting Feedrate = 30 in/min
- Ramp Feedrate = 15 in/min
- Plunge Feedrate = 15 in/min

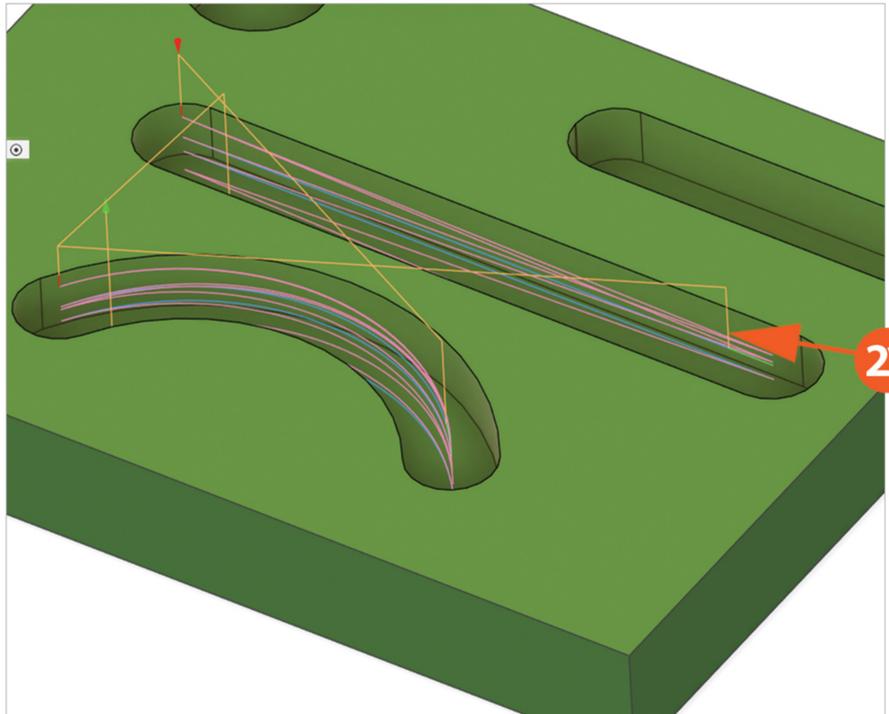
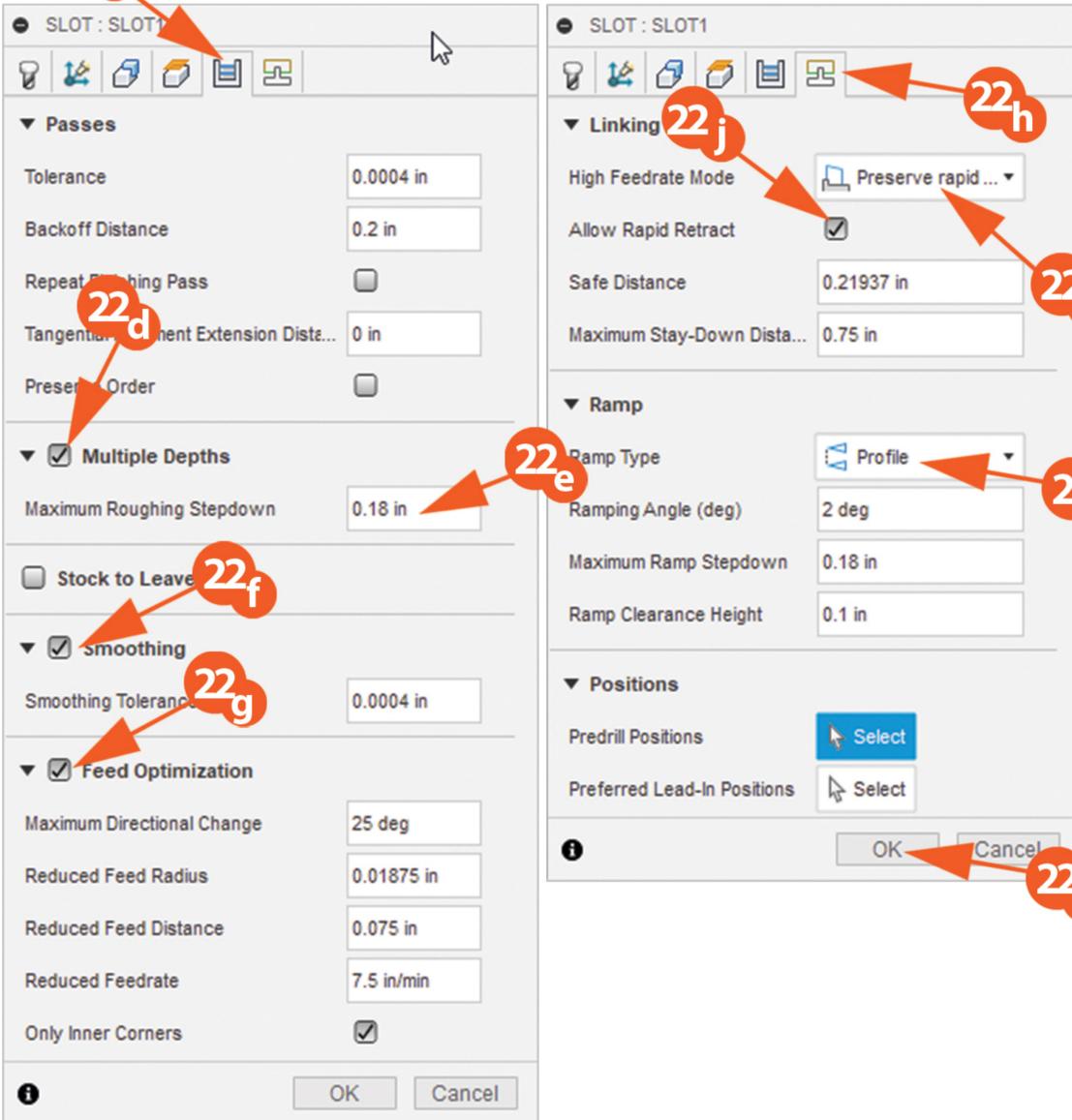
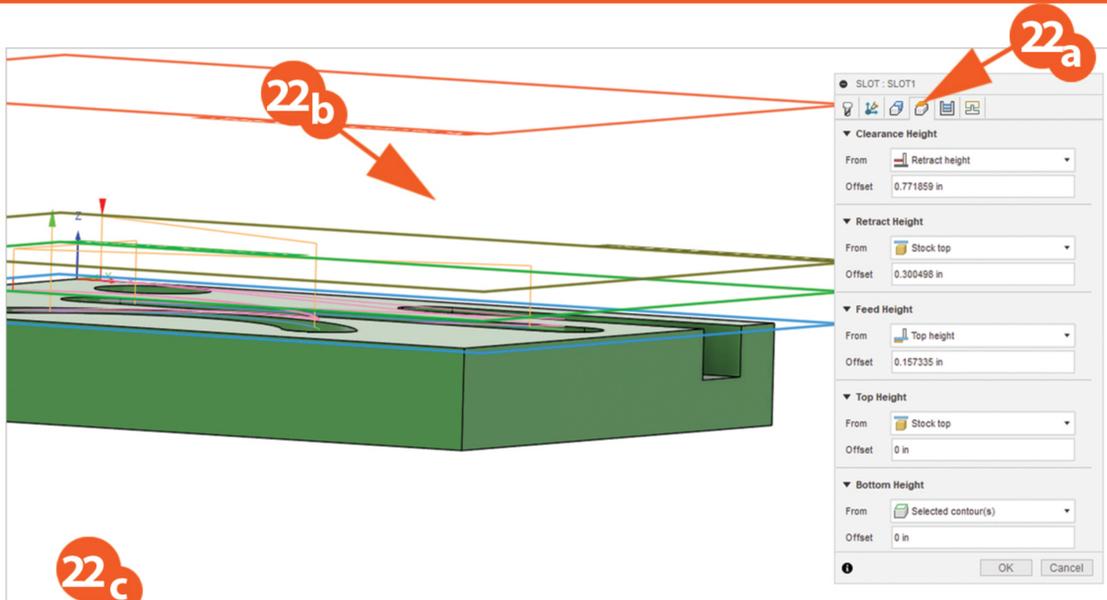
21c - Select TAB 2.

21d - Machining Type = 3.

21e - Select TAB 3.

21f - Select the top and bottom outlines of each closed SLOT.



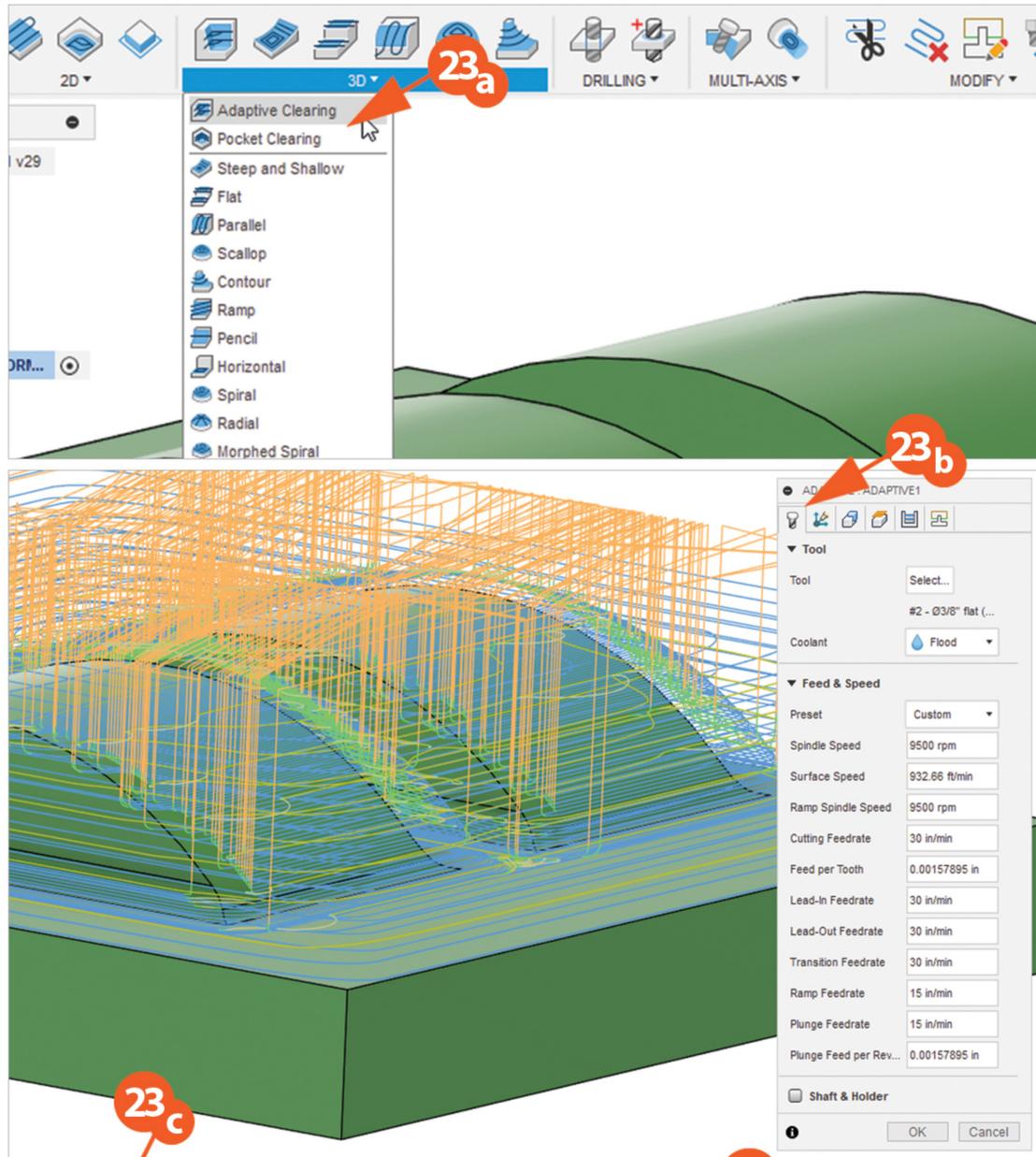


22

SLOT CONTINUED

- 22a - Select TAB 4.
- 22b - The HEIGHTS should be close to shown here.
- 22c - Select TAB 5.
- 22d - Check MULTIPLE DEPTHS.
- 22e - MAXIMUM STEPDOWN = .18"
- 22f - Check SMOOTHING.
- 22g - Check FEED OPTIMIZATION.
- 22h - Select TAB 6.
- 22i - HIGH FEEDRATE MODE/PRESERVE RAPID.
- 22j - Check ALLOW RAPID RETRACT.
- 22k - RAMP TYPE = PROFILE.
- 22L - Check OK.
- 22m - The machine paths should look like this.

To create the G-Code, go to the Post Processing Chapter on page 27.



23

We will use two operations to create our object. First is 3D Adaptive to rough out the shape. Second will be Parallel to finish the object.

You will need to create a new setup. Please see page 1.

3D ADAPTIVE - OPERATION #1

3D Adaptive will be used to quickly clear out the basic shape.

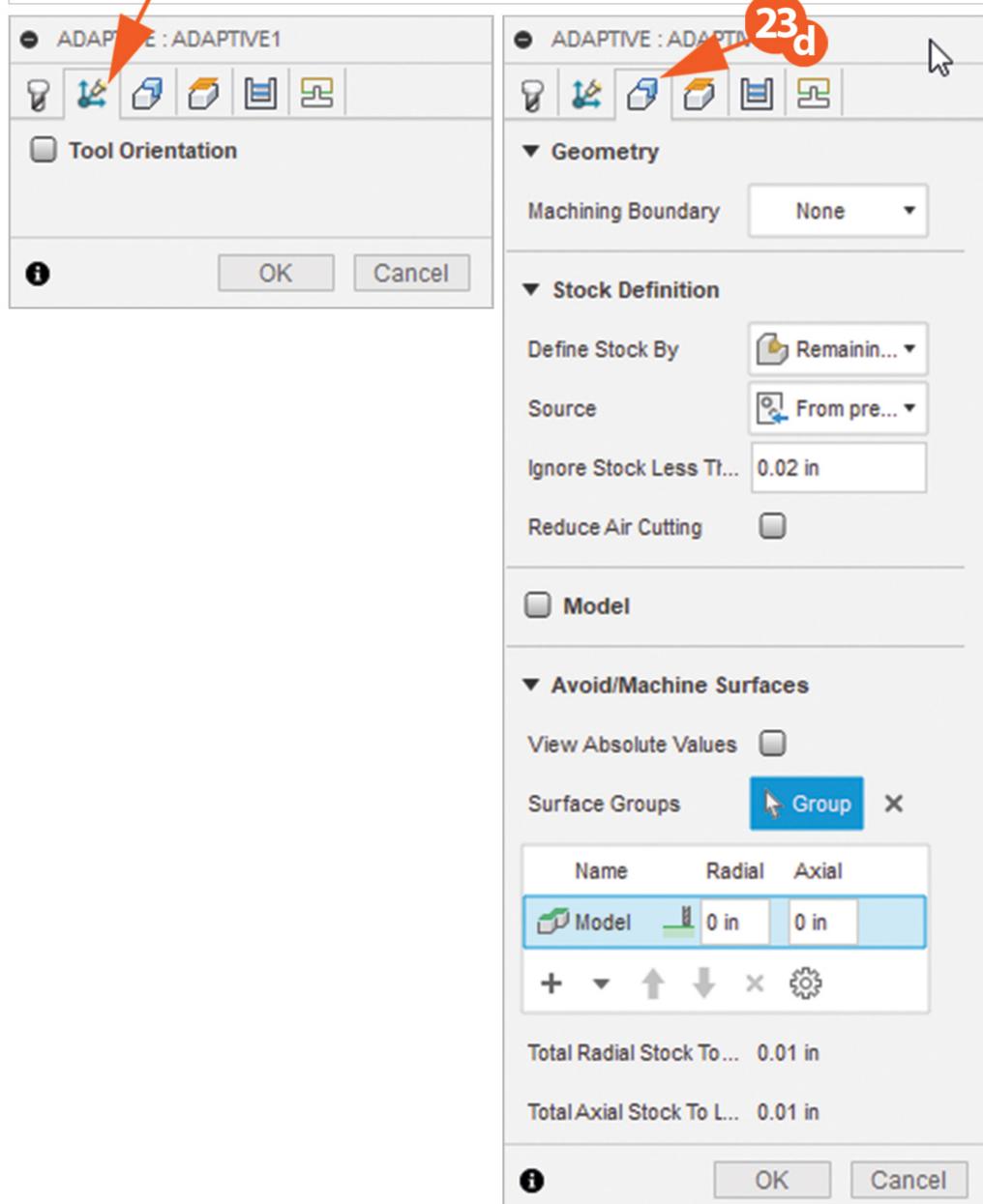
23a - Select 3D/ADAPTIVE CLEARING.

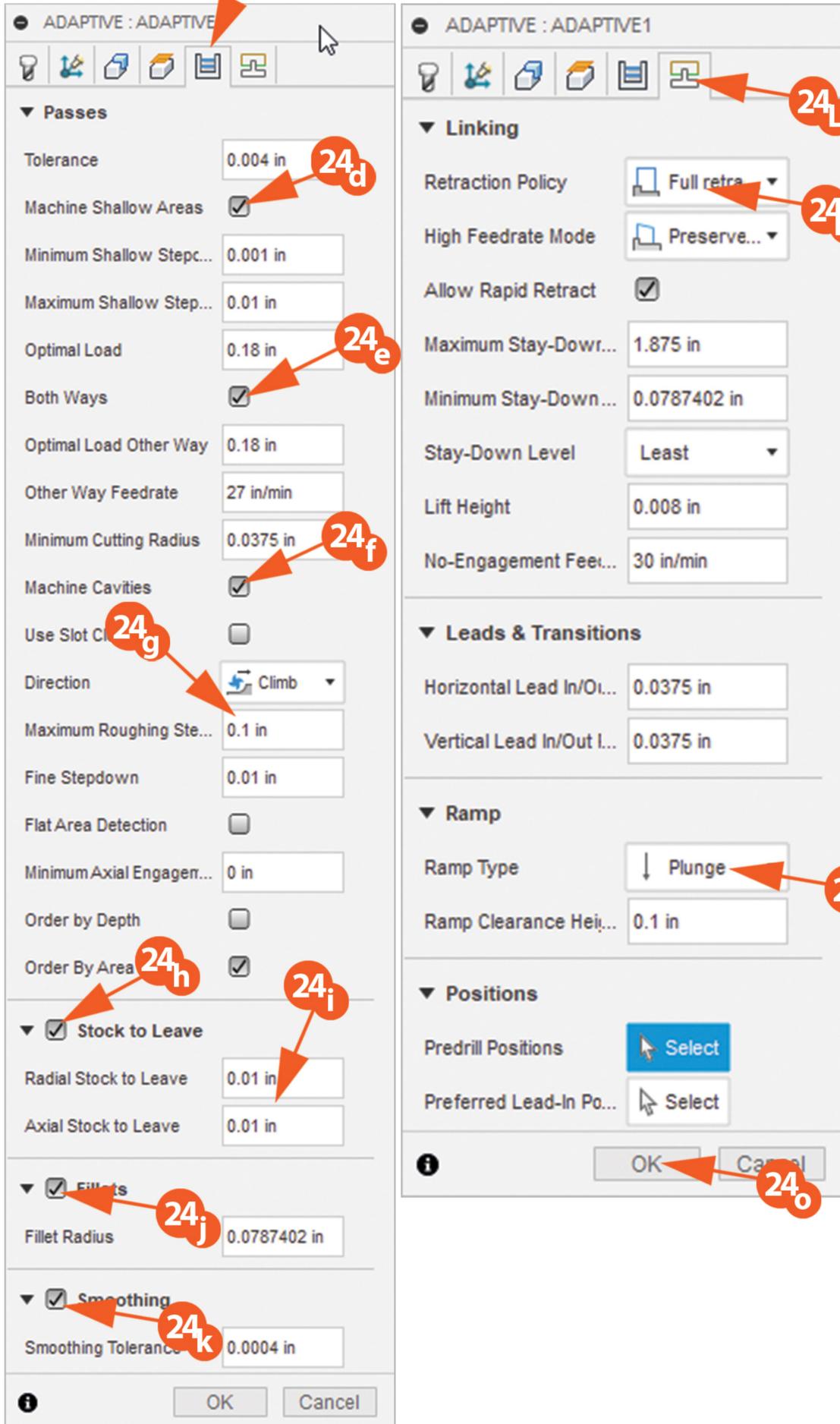
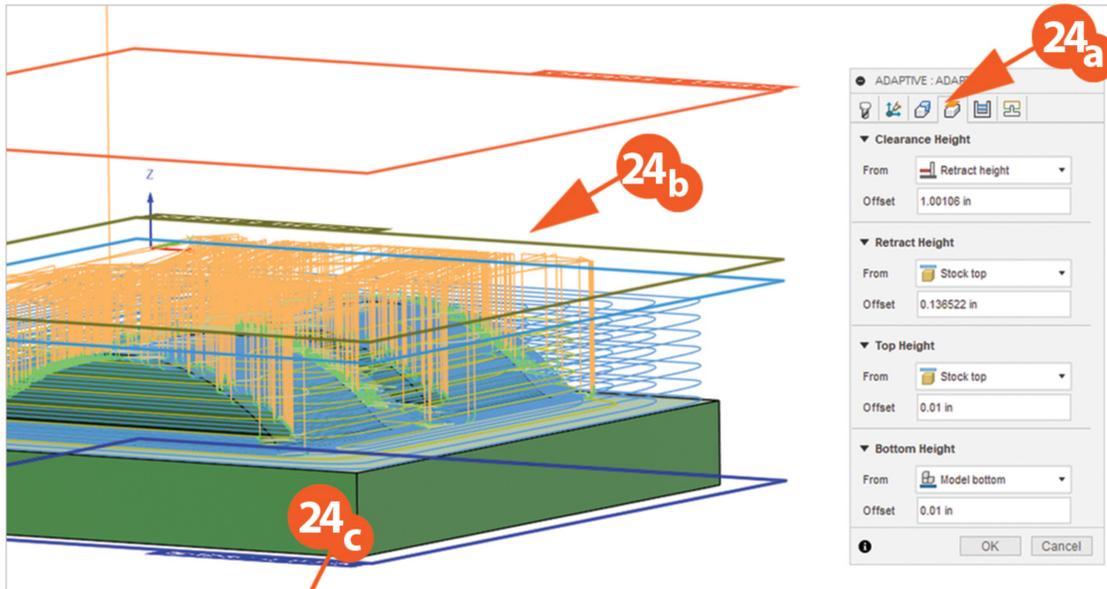
23b - On TAB 1 select the following:

- Tool = #2 (3/8" FEM)
- Coolant = FLOOD
- Spindle Speed = 9500 rpm
- Cutting Feedrate = 30 in/min
- Ramp Feedrate = 15 in/min
- Plunge Feedrate = 15 in/min

23c - Select TAB 2. Nothing to change here.

23d - Select TAB 3. Nothing to change here.





24

3D ADAPTIVE CONTINUED

24a - Select TAB 4.

24b - The HEIGHTS should look like this.

24c - Select TAB 5.

24d - Check MACHINE SHALLOW AREAS.

24e - Check BOTH WAYS.

24f - Check MACHINE CAVITIES.

24g - MAXIMUM ROUGHING STEPDOWN = .1"

24h - Check STOCK TO LEAVE.

24i - RADIAL/AXIAL = .01"

24j - Check FILLETS.

24k - Check SMOOTHING.

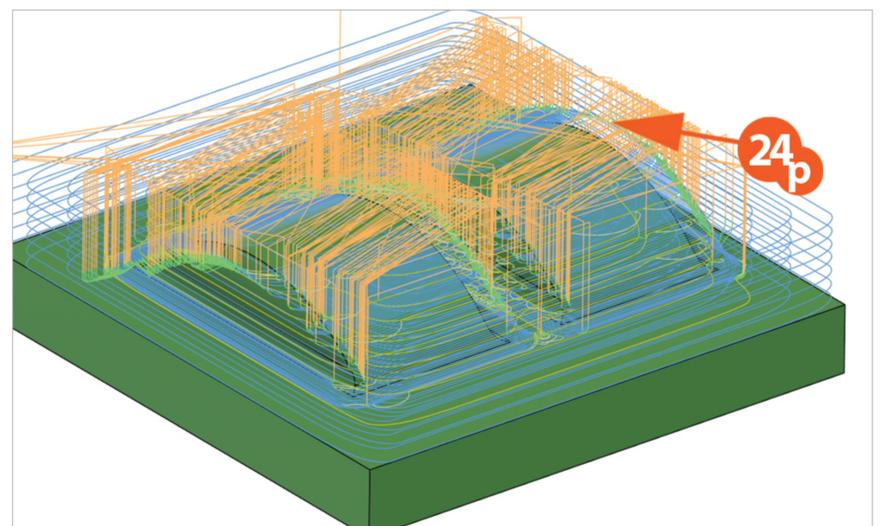
24L - Select TAB 6.

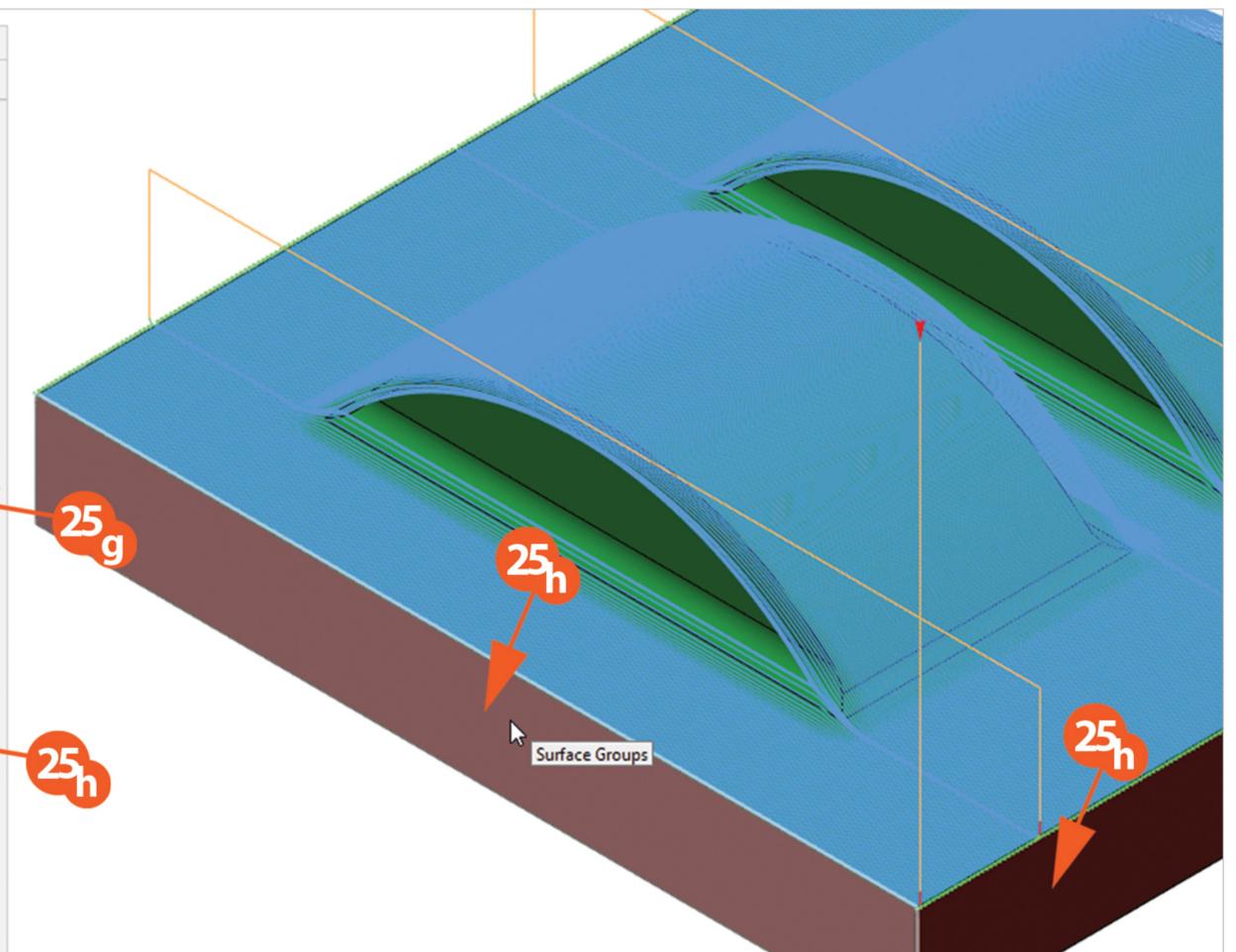
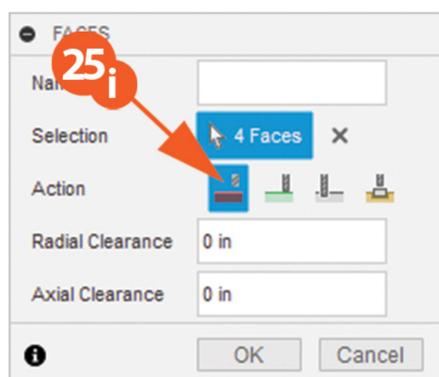
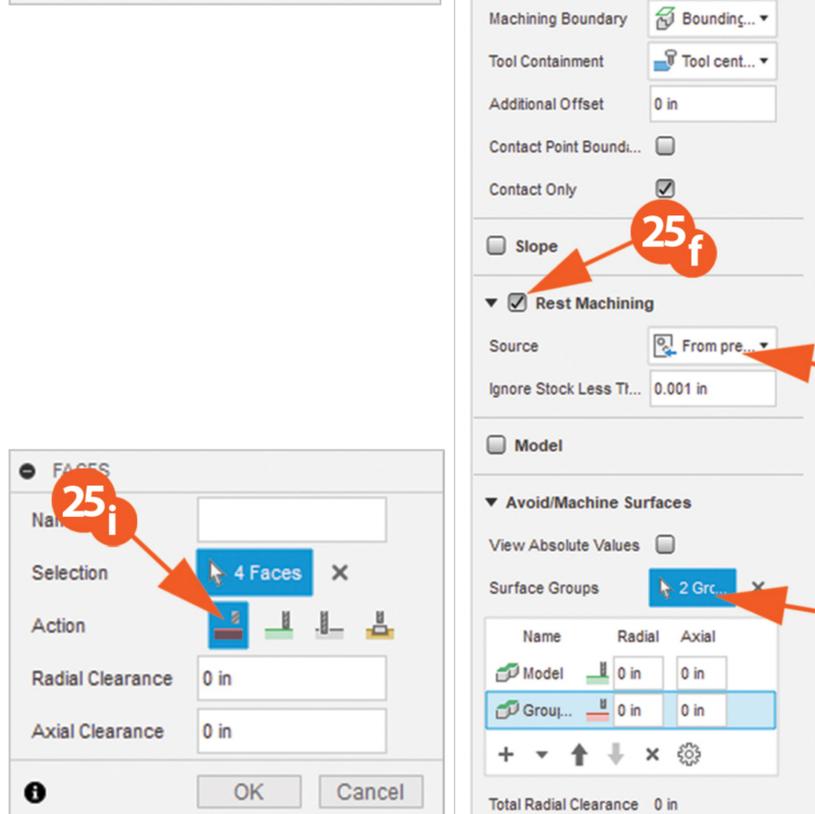
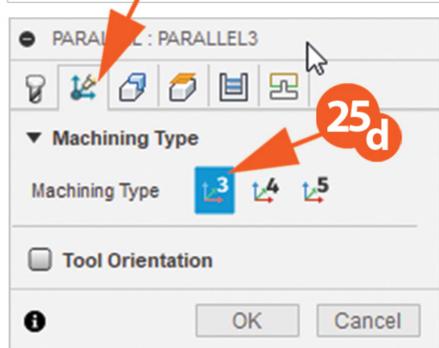
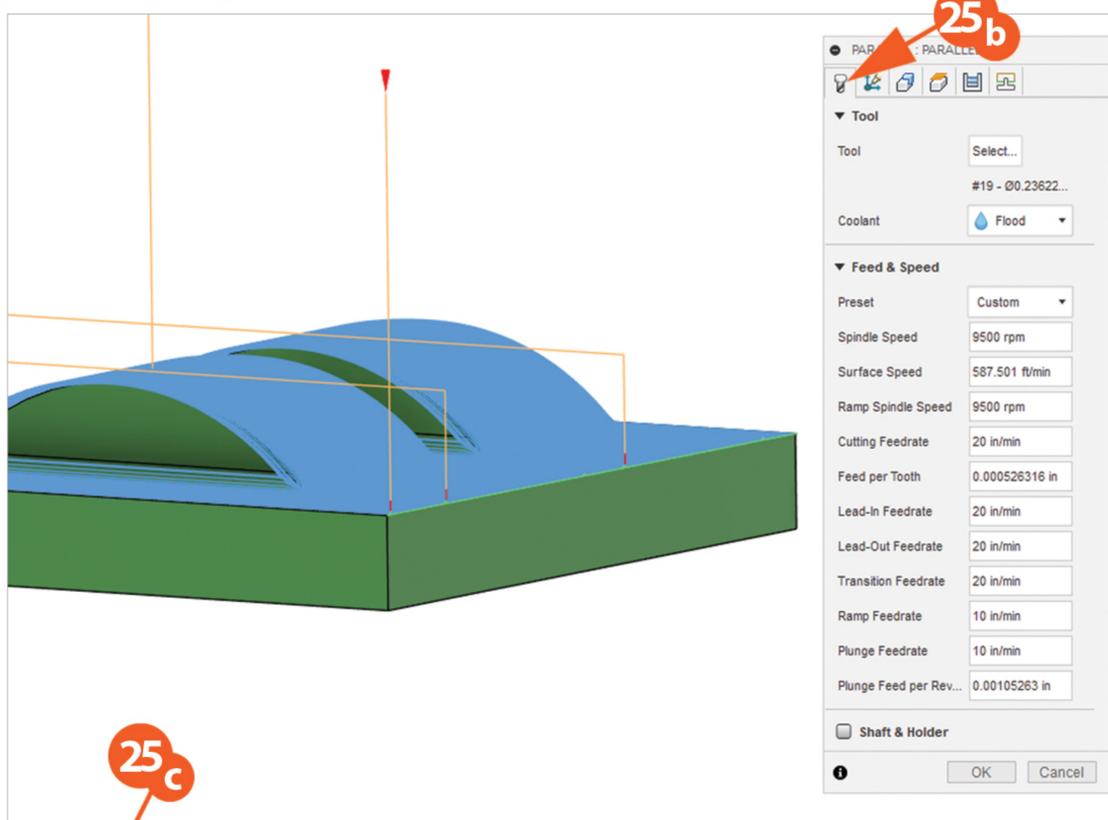
24m - RETRACTION POLICY = FULL RETRACT.

24n - RAMP = PLUNGE.

24o - Click OK.

24p - The paths should look like this.





25

3D PARALLEL - OPERATION #2

3D Parallel will be used to finish the object.

25a - Select 3D/PARALLEL.

25b - On TAB 1 select the following:

- Tool = #19 (6mm FEM Long)
- Coolant = FLOOD
- Spindle Speed = 9500 rpm
- Cutting Feedrate = 20 in/min
- Ramp Feedrate = 10 in/min
- Plunge Feedrate = 10 in/min

25c - Select TAB 2.

25d - MACHINE TYPE = 3.

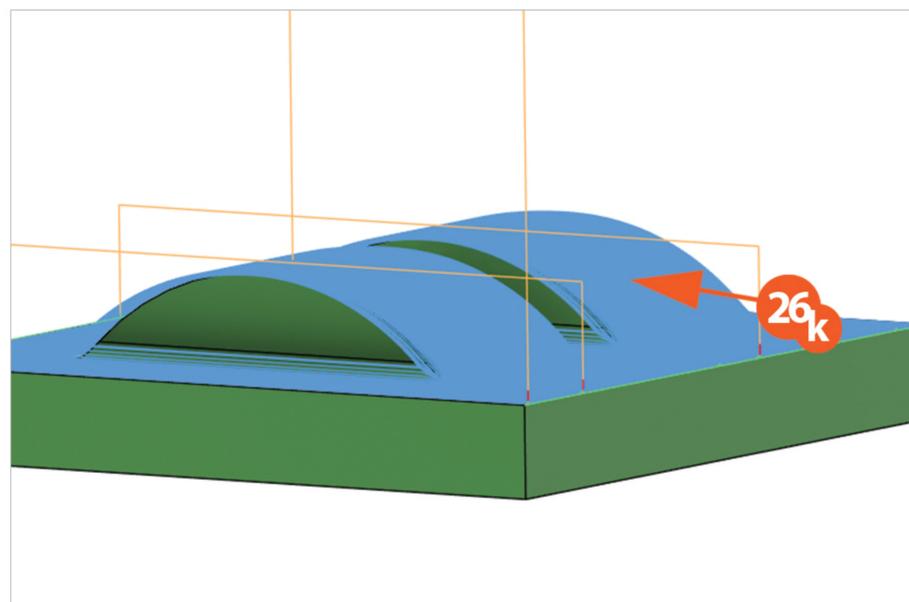
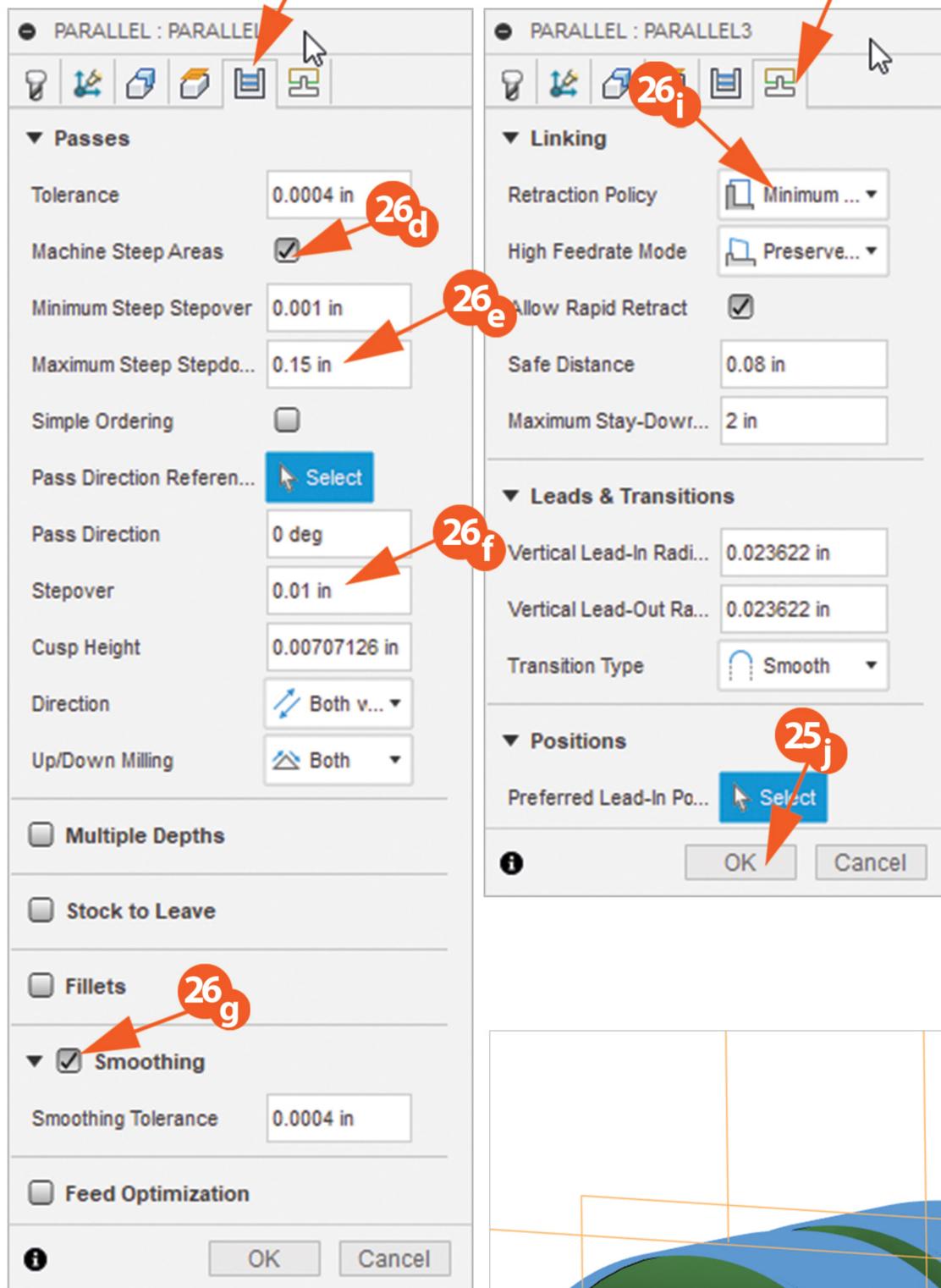
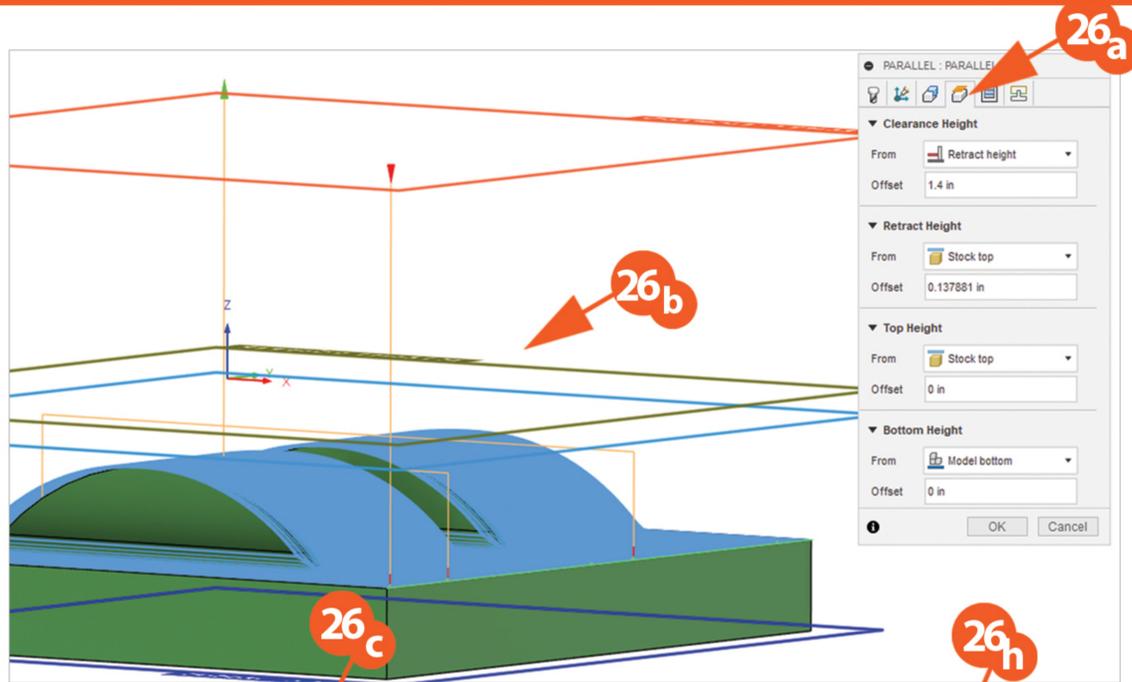
25e - Select TAB 3.

25f - Check REST MACHINING.

25g - SOURCE/PREVIOUS OPERATION.

25h - AVOID/MACHINE SURFACES, select the 4 sides of the object.

25i - Make sure to select AVOID SURFACE.



26

3D PARALLEL CONTINUED

26a - Select TAB 4.

26b - The HEIGHTS should look like this.

26c - Select TAB 5.

26d - Check MACHINE STEEP AREAS.

26e - MAXIMUM STEPDOWN = .15"

26f - STEPOVER = .01"

26g - Check SMOOTHING.

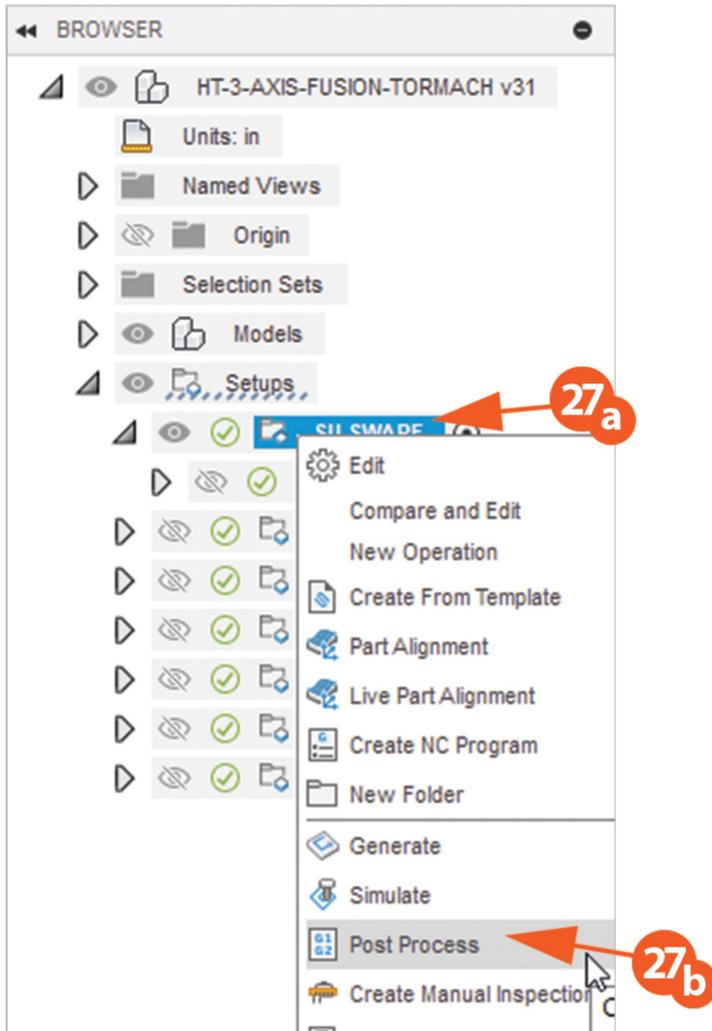
26h - Select TAB 6.

26i - RETRACTION POLICY/ MINIMUM RETRACT.

26j - Click OK.

26k - Your machine path should look like this.

To create the G-Code for the 2 operations, go to the Post Processing Chapter on page 27.



27

POST PROCESS

For each Setup and it's operations you have two choices. You can create a G-Code for all the operations in one file, or create a G-Code for each operation.

We are going to create the most common way, creating a G-Code for all of the operations.

27a - Select the SETUP and RIGHT CLICK on it.

27b - From the pop-up select POST PROCESS.

27c - POST/TORMACH PATH PILOT.

27d - Name your file.

27e - Select the OUTPUT FOLDER where you want your file to be saved to.

27f - ROTARY TABLE/NO ROTARY.

27g - Check USE COOLANT.

27h - Click on POST to save the file.

