Tormach Training Workshops

Workshop 2: Helping Hands Base (.5-20 thread)

1. **Prerequisite**
    a. Session 2.1
        i. Complete the Session 2.1 Prerequisite Quiz

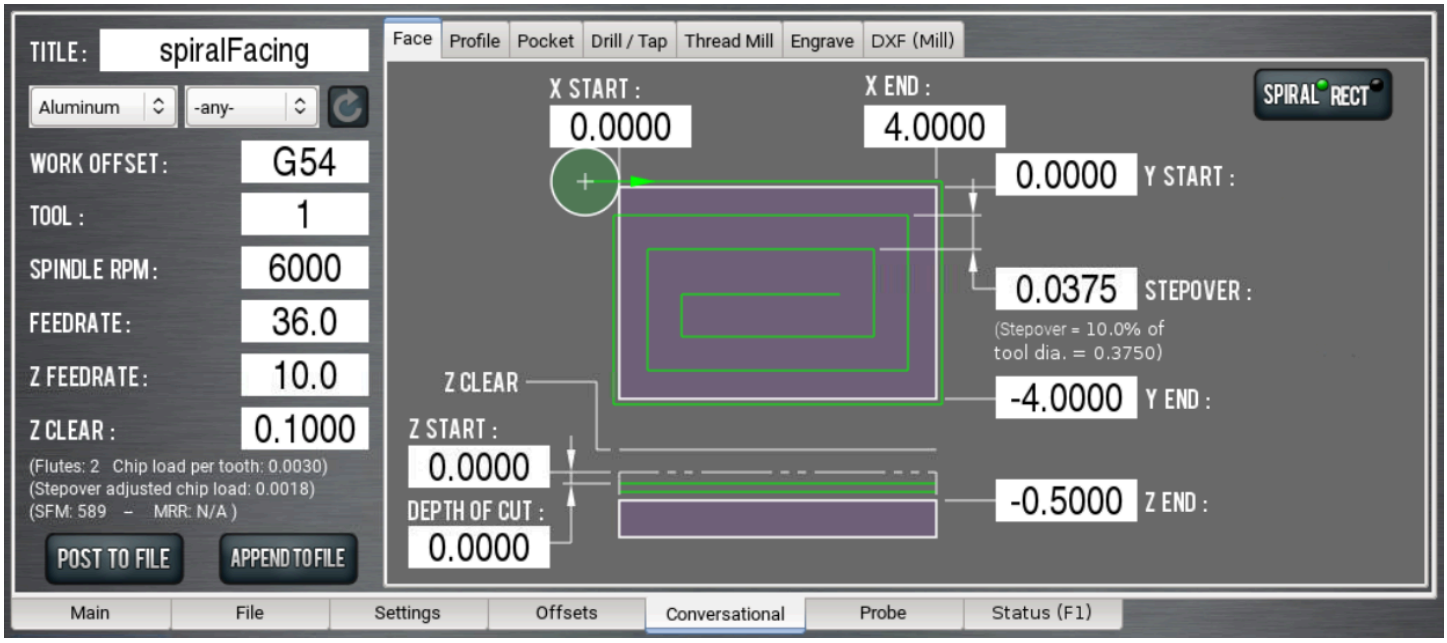
    b. Session 2.2
        i. Complete the Session 2.2 Prerequisite Quiz

2. **Purpose**
    a. Learn about speeds & feeds
    b. Learn about threads
    c. Basic introduction to CAM software using Fusion 360
    d. Transferring G-code programs to PathPilot
    e. Safety discussion
    f. Machine a base and assemble a helping hands tool

3. **Session 2.1** (Classroom)(Time: 45 minutes)
    a. Questions or issues from Workshop 1
    b. Review Session 2.1 Prerequisite Quiz
    c. Speeds & Feeds Worksheet (NYC CNC)
    d. "Tormach Tooling Tips" document

4. **OPTIONAL - Speeds and Feeds Demonstration** (Machine)(Time: 2 hours)
    a. Using PathPilot conversational, program a profile cut to demonstrate speeds and feeds. (the X and Y end values in the following picture will need to be adjusted depending on material stock size used in the exercise)
        i. Start Up Procedure
        ii. Tool Setting Procedure
        iii. Work Offset Procedure
            1. Use this as an opportunity to practice edge finding.
            2. Safety Note - you may need to lean inside the machine to see the edge finder better, especially in X. Be aware of the rotating spindle.

        iv. Pathpilot conversational programming for 6061 aluminum
            1. ⅜ 2FL HSS endmill = S6000 F36. (589 SFM, .003 IPT)
                a. .500 axial (1.3 times cutter diameter)
                b. .0375 radial (10% of cutter diameter)



v. Machining
   1. Position the Fogbuster nozzle so that it will not hit the part
   2. Adjust Spindle RPM and Feed Rate override sliders while cutting to show changes in chip thickness and sound
   3. Practice 3 methods of stopping the machine during this exercise (feed hold, stop, and emergency stop)
   4. Tool Chatter Notes - If tool chatter occurs while taking light cuts, it can usually be fixed by increasing the chip load. Try decreasing spindle rpm OR increasing feedrate. Tool chatter caused by taking too heavy of a cut will be made worse by increasing the chip load and can cause the tool to break or spindle to stall. You should stop the machine and adjust your program to take lighter cuts.

vi. Cleanup
   1. Notice the chips built up in the Y axis way cover (between table and Z axis column). Users may occasionally need to vacuum that area during the machining process and not just at the end when the part is finished.

vii. Shutdown Procedure

5. **Session 2.2** (Classroom)(Time: 2 Hours)
   a. Review Session 2.2 Prerequisite Quiz
   b. Thread milling
      i. In our experience, the machine does not have enough horsepower to run a ⅜-16 tap in aluminum. A thread mill should be used for threads larger than ¼”.
      ii. Internal/External threads

iii. Single profile/full profile thread mills
iv. Calculating Pitch Diameter Offset
    1. Choose tap drill size for ½-20 2B threads Tap and Drill Chart
    2. Use the Thread-Mill-Calculator-Rev6 (NYC CNC) excel document to calculate the pitch diameter offset

c. Tapping
    i. Floating Tap Holder
        1. Rigid tapping is not available with stepper motors so a tension/compression style tapping head must be used. This will account for the difference between the programmed RPM and actual motor RPM. Tapping should be done in low range because the error in calibrated spindle rpm is smaller in low range than in high range.
    ii. Taps need cutting fluid (do not run dry)
    iii. Tapping by hand is another viable option depending on size and number of holes.

d. Basic introduction to Fusion 360 CAM software
    i. Helping hands base .f3d file is available in this folder
    ii. Show the location of the pitch diameter offset that was previously calculated
        1. Right click and Edit the thread milling toolpath, Passes tab
    iii. Setups (first side, second side, stock size, X0Y0Z0)
    iv. Operations (toolpaths)
        1. Facing, drilling, profile, chamfer, pocket
            a. Notice that the facing toolpath leads in from outside the part but does not leave the face on the left side. Machine travel limitations with fly cutter.
        2. Recommendation to rename toolpaths to make G-code program comments easier to read and understand what the code is doing.
    v. Machining simulation
        1. Use comparison colorization mode under Stock in order to highlight differences between model and actual machined material
    vi. Set-up sheets
    vii. Post processor
        1. Select “Tormach PathPilot” post. No changes are required to the default options.
    viii. Reading the G-code
        1. Safety line
        2. The work offset number for the part program will be located inside the first toolpath (G54, G55, G56…) The safety line will always have G54 so do not look there if you are verifying which coordinate you are using.
        3. N numbers and program comments
        4. M01
        5. T# G43 H# M6
        6. S#. M3 M8
        7. G00 and G01 F#.
        8. Drilling canned cycle (variables)

9. M5 M9
10. G30
11. M30

6. **Session 2.3** (Machine)(Time: 4 hours)
    a. Measure student's material stock size and verify that it matches the setup sheet information

    b. Start Up Procedure

    c. Workholding
        i. If two vises are in the machine, use the vise on the left for the helping hands base. You'll need the lead in/out room on the right side of the part with the fly cutter.
        ii. Use a set of 4" parallels (1.125) that will leave clearance under the part for the drill to go through (avoid the vise) and clearance on top for the chamfer tool (avoid the jaws)
        iii. Tightening the vise
            1. Use the vise speed handle for quickly opening and closing the vise
            2. Don't use a hammer or mallet on the vise handle
            3. After the vise is closed, a rubber mallet can be used to tap down the part against the parallels.

    d. Tool Setting Procedure

    e. Work Offset Procedure
        i. Verify G54 or G55

    f. Running the program
        i. Review "When to use feed hold, stop, reset, and emergency stop buttons" from Workshop 1
        ii. Use the maximum velocity slider to slow down the feed rates when starting the cut
        iii. Use feed hold and verify "distance to go" (DTG) numbers

    g. Shut Down Procedure

Workshop 2 kit - bill of materials		
Qty	Description	Link/Reference
1	6061 aluminum for Base	Stock size to purchase 36x4x5/8 length = 6.06 +.06 -0 (saw cut ends) width = 3.950 +0 -.005 (mill both sides) thickness = ⅝" (stock, no prep)
4	Alligator Clip	https://www.mcmaster.com/7236K86/
4	¼" Segmented Coolant Hose w/ nozzle w/ Base	NEED LINK
4	feet	3D print cheetah, base - feet (.5-20 thread).STL
4	#8-32 x 1 ¼" OH PHL Mach Screw	https://www.mscdirect.com/product/details/67573402 OR https://www.mcmaster.com/91802A201/
8	Heat Shrink Tubing ⅛" dia. ⅝" long	make717 electronics area
1	¼ 2fl carbide ball end mill	https://www.onlinecarbide.com/242500.html
1	1 gallon zip lock bag	

Documents to print and hand out:

1. Workshop 2 Syllabus (Training Workshops)
2. Session 2.1 Prerequisite Quiz ANSWERS
3. Session 2.1 Speeds and Feeds Exercise Setup Sheet
4. Session 2.2 Prerequisite Quiz ANSWERS
5. Helping Hands Base (.5-20 thread) Rev 4 Drawing (Drawings/Models)

Teaching Props:

1. Session 2.2
    a. Single profile threadmill
    b. Machinery's handbook
    c. Fusion 360 setup sheets
        i. Setup1
        ii. Setup2

Wild Apricot Event Description

Title
Tormach Workshop #2 (Helping Hands Base)

Cost
See course outline

Tags, Location, Date/Time
Classroom session weeknight 6-9pm
Machining sessions weekend 10am-3pm

Description
This is the second of four workshops required to be a qualified Tormach user. Class size is limited to two persons. Please make sure you are available for all sessions prior to registering.

Participants completing this workshop will keep the machined base, and will receive all the parts required to assemble it. A new end mill will also be provided.

Prerequisites

1. Successful completion of Workshop 1
2. Complete the Session 2.1 Prerequisite Quiz
3. Complete the Session 2.2 Prerequisite Quiz

Workshop 2 syllabus

Session 2.1 and 2.2 (classroom): learn about “speeds and feeds”, tapping, thread milling and receive a basic introduction to Fusion 360.
Session 2.3 (machine): machine a helping hands base using the instructor’s verified part program.

The classroom session will be taught by Kyle Hossler and the machining session by Mike Ireland.