HSMworks turning step by step.

Open the file and create a new job

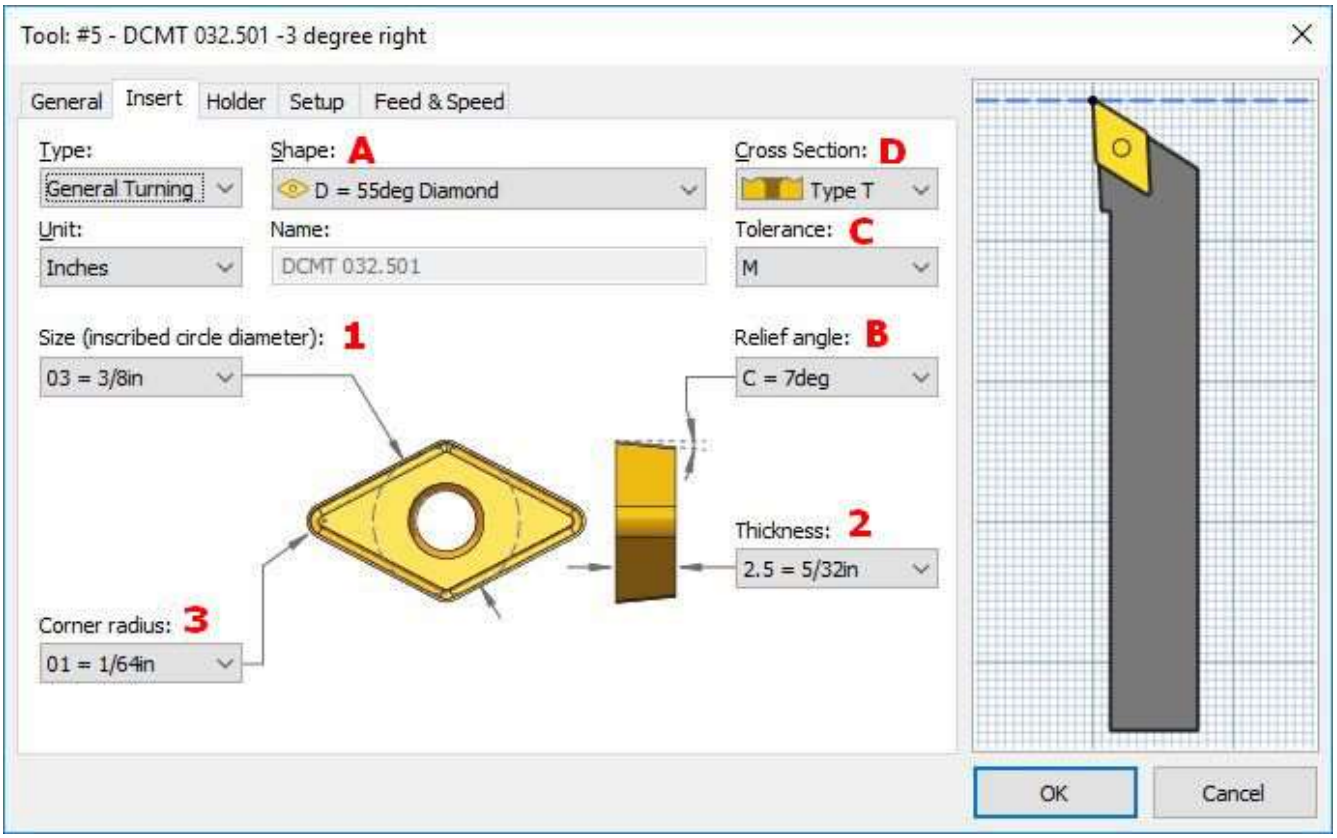
1) Open the “LatheTest.step” file in Fusion 360 (File → Open → Open from my computer...)
2) (Optional) Save a copy of the Fusion file to your local computer (File → Export, use *.f3d)
3) Select the MANUFACTURE tab from the drop down list in the toolbar
4) Select “New Setup” under SETUP dropdown menu
5) Set operation type to Turning or Mill/Turn
6) Select model. Use “Spun Profile” option with models that aren't radially symmetric.
7) Set Chuck Reference to “Model Back” and Offset to -1” (back of part is 1” away from chuck)
8) Click “Stock” tab. Change the stock to Cylindrical if necessary
9) Select Fixed Size Cylinder
10) Enter a diameter of 1”, Length of 6”
11) Model Position should be “Offset from front”
12) Offset should be 0”
13) Leave options under Post-Process tab at default. Click OK

Add an operation and create a tool

1) Right click on Setup1 and select New Operation – Turning – Turning Profile
2) Click “Select...” next to Tool. Tool library opens.
3) Click on New Turn Tool (parting blade icon in upper right corner)
4) Click on the Insert tab
5) We will be using a DCMT 32.51 turning insert

Profiling and boring insert codes are 4 letters and 3 numbers (single digit is ANSI/inches, double digit is ISO/mm). Formatted ABCD123 (ANSI) or ABCD010203 (ISO)

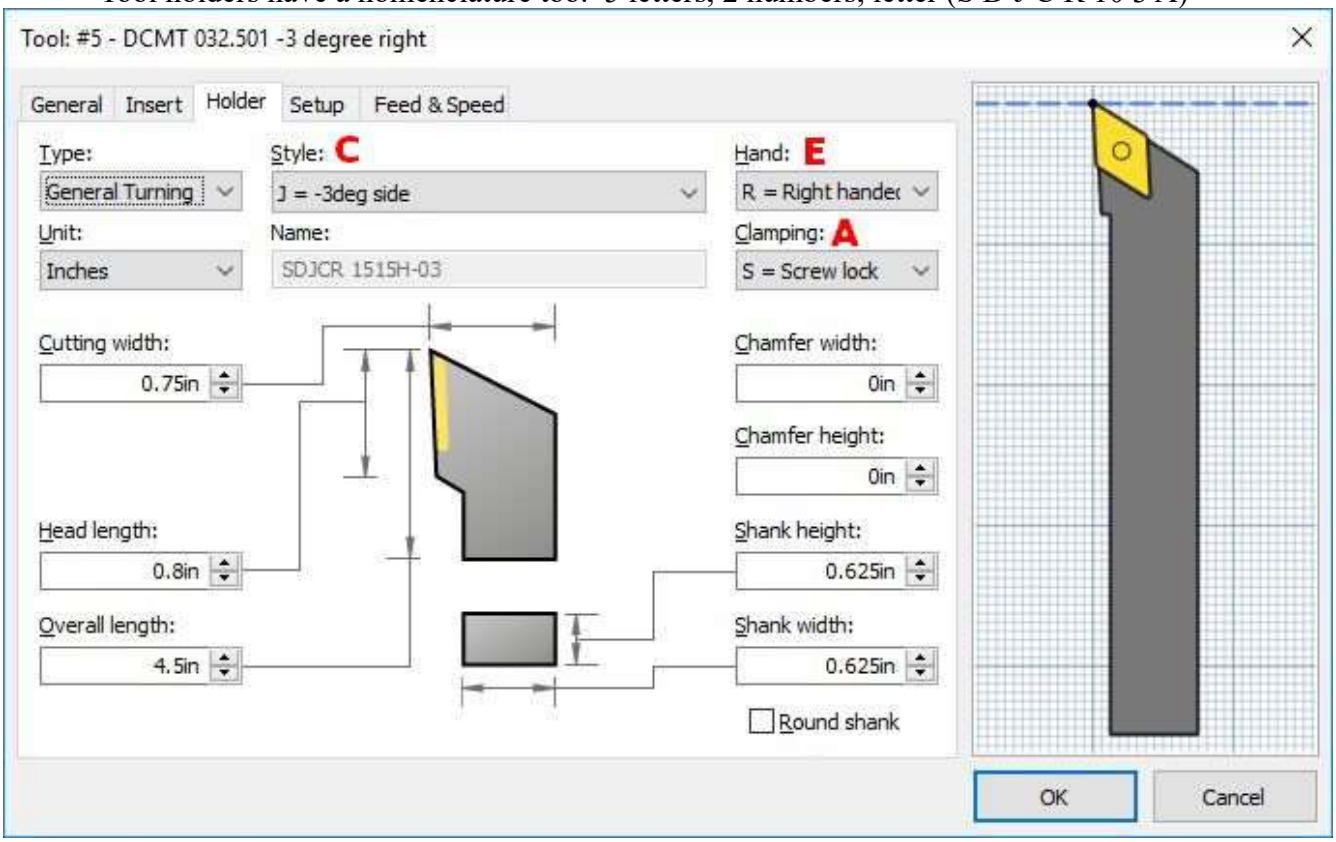
- Example code is D C M T 3 2.5 1
- Letters define shape of insert, numbers define size (not all TCMT inserts the same size)
- http://www.carbidedepot.com/formulas-insert-d.htm for more information



- A Shape code: T for triangular, R for round, many others
- B relief angle: N=0 degree, up to G=30 degree (lookup chart)
- C tolerance: when in doubt use "M" (loosest tolerance inserts)
- D cross section: is mounting hole countersunk? chipbreaker? one or both sides?
- 1 size: english - diameter of inscribed circle in 1/8" (1/32" for IC less than 1/4")
  metric - diameter of inscribed circle in mm
- 2 thickness: english - thickness in 1/16" (1/32" if IC is less than ¼") decimals allowed
  metric - approximate thickness in mm (look up exact values)
- 3 nose radius: english - radius in 1/64" (0 = "sharp" = max radius 0.005")
  metric - radius in 1/10 mm

6) Next click on the Holder tab

Tool holders have a nomenclature too. 5 letters, 2 numbers, letter (S D J C R 10 3 A)



- http://www.carbidedepot.com/formulas-th-d.htm for more information

- first letter (A) is how the insert is locked to the holder

- second letter is insert shape (redundant if you know insert code)

- third letter (C) is insert angle relative to holder (-5 degree, +3 degree, etc...) Note that a 0 degree tool has leading edge perpendicular to rotation axis. (Not a “neutral” tool)
  For Right hand tool: positive rotation is CW, negative CCW from perpendicular.
  For Left hand tool: positive rotation is CCW, negative CW from perpendicular.

- fourth letter is relief angle of insert (again redundant)

- fifth letter (E) is direction (Left, Neutral or Right handed)

- first number is square shank in 1/16” (10 = 5/8” = 10/16”, etc...)

- second number is insert IC size (redundant)

- last letter is overall length (look up or just measure)

- HSMworks doesn't use holder codes for size – just for shape. Measure shank and enter length and size numerically. 3rd and 5th letter most important info

- Tool holder we are using is an SDJCR. The shank is 5/8" square and 4.5" overall length. The head length is 1" and the cutting width is 0.75"

7) Next click on the Setup tab

   - Use 0 degree orientation (boring bars will be 90 degree) and check "Clockwise" rotation

8) Next click on Feeds and Speeds

   - Use a Spindle Speed of 2000 Revolutions per minute
      - lathe supports two speed modes: RPM and SFM
      - SFM mode adjusts RPM as tool moves toward center - maintain SFM across tool nose
      - max RPM limit must be used in HSMworks, based on gear choice. You will also need to calculate the minimum and maximum RPM to determine which gear you need to be in (more on gear selection later)

   - Use a Feed Rate, Lead-in and Lead-out of 15 Inches Per minute
      - also two feed modes. IPM and IPR (aka Spindle Synchronous Feed)
      - Inches Per Revolution syncs feed rate to spindle rotation
      - IPR is primarily for threading operations

9) Click OK. Now select the newly created tool and click OK.

10) Click on Radii tab (red cylinder in middle of tabs)

   - Set Clearance to 0.2" offset from Stock OD
   - Make sure Outer Radius is from Stock OD, Inner Radius from Stock ID (These settings are REVERSED - Outer radius is ID and inner radius is OD - for interior / boring operations!)

11) Click on Passes tab (4th tab – tool with blue lines)

   - Compensation Type should be in computer
   - check Finishing Allowance
   - Set Stepover to 0.04"
   - Check "Roughing Passes", set Maximum Roughing Stepdown to 0.01"

12) Click OK. Wait for toolpath to be generated

Simulate your toolpath

1) Select Setup1 (not the operation)
2) Click Simulation under the ACTIONS dropdown menu in menu bar
3) Check "Stock" box, check "Transparent"
4) Click Play and watch the simulation
5) Drag slider to change speed. Left of center is reverse simulation.
6) Watch for any collisions
7) Also watch for excessively deep cuts (especially at the end closest to spindle)

Exercise #2 – adding a second operation:

Add a second Turning Profile Operation, to clean stock off the back edge of the sphere.

Create a new tool using the following:

- CCMT 32.51 insert
- SCLCL holder, with a 5/8” square shank, 4.5” overall length, 3/4” head length, 3/4” cutting width
- 2000 RPM spindle speed, 15 IPM feed rates

– Use the same clearances and stepdowns as previous operation. Make sure “Leave allowance” is unchecked.

Simulate the job and watch what happens!

Use Confinement to prevent nasty tool crashes

Double click on second Operation to reopen the operation settings
Click the Geometry tab (blue cylinder, right after tool tab), check the Confinement box

Confinement allows us to move the Z boundaries of the tool path

Frontside offset moves the cut range from the end, Backside offset moves the cut range nearest the chuck. NEGATIVE numbers move the range TOWARD the chuck

Set the back side confinement to 0.25

Also check the Rest Machining box. Rest machining tells HSMworks to account for previously removed stock when generating a tool path (as in "machine the rest of it") . This prevents “air cutting”. Make sure Source is “From previous operation(s)”.

Run the simulation again and notice the difference.

Replace the CCMT 32.51 tool with a new tool in second profiling operation.

Notice that HSMworks assigns Tool #3 to this tool. Old tool will be kept in tool list.

DCMT 32.51 insert

SDNCN holder, 5/8” square shank, 4.5” overall length, 5/8” head length

2000 RPM spindle, 15 IPM feeds

Change Confinement in the second profiling operation back to 0 offset. Watch what happens at the far left end of the simulation. This is a general problem with neutral or L/R hand tools with a positive angle. Especially comes up with Threading operations. You can use Confinement to prevent the tool cutting all the way to the end. There are other solutions to this problem, such as extending the model of your part on the back end to remove excess stock, use grooving to remove stock, &c... Removing stock at the back end weakens the material – greater risk of snapping your part off prematurely.

Post Process: Generate your G-code file

1) Make sure Setup1 is selected (not an operation).
2) Select “Post Process” under ACTIONS dropdown menu
3) select the 'Fanuc Turning' post processor
4) Set the Output folder to wherever you want to store your Gcode files
5) This post requires program numbers (set “Program name or number” to 1001)
6) Make sure “Open NC file in editor is checked”
7) In Properties, set Max Spindle Speed to 2000
8) When asked for a file name, use [YourName]LatheTest (doesn't have to be a number)
9) Next, the Brackets editor pops up. We need to make a few changes.
10) remove all G17, G18 and G19 references from code (curve plane – lathe has only one)
11) gear codes: manually insert for speed changes. On the line just before “Sxxxx M3” add ONE of the following M codes:
    - M41 low gear: speeds 10 to 84 RPM (36:1 ?)
    - M42 medium gear: speeds 45 to 380 RPM (8:1 ?)
    - M43 high gear: speeds 240 to 2000 RPM (3:2 spindle/motor ratio?)
    - make sure spindle OFF for gear changes (add M05 if necessary)
    - Spindle control Mcodes: M05 stop, M03 forward, M04 reverse
    - Program will STOP if you are in a gear that can't support selected speed!
    - RPM mode easy: select gear that supports RPM desired
    - if you have a choice: high end of lower gear is usually best (higher torque & Max HP)
    - remember that spindle horsepower is lower at lower speeds within a gear
    - SFM mode harder: have to know starting OD and OD at deepest cut
    - 3.82 * SFM / OD = RPM (3.82 = 12/Pi)
    - find minimum and maxiumum RPM and select gear that covers both speeds
    - be sure to set RPM cap in HSMworks, based on gear selection, when using SFM
    - can use top/bottom to split a cut in to two passes, one at lower gear than the other
12) Save your changes