Fusion 360 Turning -Spin a Creative Workflow with Ease

Speaker/Author: Kevin Lee
LeeVerage Integration Inc.

Co-Speaker: Tim Paul
Autodesk

Learning Objectives

- Learn how to define a turning setup with WCS alignment and coordinate systems
- Apply different toolpath strategies understanding specific parameters
- Validate turning workflows with stock simulation
- Initiate automatic setup sheets and post processing g-code
- Deploy Mill/Turn strategies with live tooling
- NEW! Turning Product Manager Tim Paul shares what is coming in Turning

Description

This class is intended to assist new Fusion 360 CAM users in learning the turning environment interface, and how to create: job setups, define stock, establish work coordinate systems, turning operations, stock simulation, setup sheets, and post processing g-code by exploring the manufacturing workspace. This class is also well suited for existing Fusion CAM users that want to learn tips and tricks with live tooling and how to optimize turning workflows. With an added guest appearance by Tim Paul who can provide a sneak peak of what's being planned for future development this will be a class you don't want to miss.

Class Speaker Kevin Lee

Kevin is an innovative entrepreneur specialized with integrated CAD/CAM manufacturing technology solutions. He graduated with a Mechanical Engineering diploma continuing his education with a tool maker apprenticeship with focus on CNC manufacturing various projects such as wind tunnel model sky scrapers and specialized aeroelastic instruments with the University of Western Ontario for sixteen years.

His continued education with Western Engineering focused on effective leadership led him to a successful part time college teaching career as a CNC manufacturing professor extending his knowledge and experience to many of his students. Using this experience, Kevin provides leadership as managing director of his own company LeeVerage Integration Inc. located in southwestern Ontario, Canada.

Kevin's focus on integrated CAD/CAM solutions provides guidance to the firm and its clients looking for innovative technologies that deliver competitive advantage with long term sustainability. Kevin extends dedicated on-demand software cam support and training services to our clients allowing them to compete at a world class level utilizing Autodesk's HSM cam manufacturing products like Fusion 360, Inventor Cam, and HSMWorks.

Co-Speaker Tim Paul

Tim Paul is currently a Product Manager for Fusion 360 Manufacturing Platform and Turning. Over the previous four years, Tim has worked in various roles including Application Engineer for HSM, Customer Success Manager for Fusion CAM, and Manufacturing Business Strategy Manager.

Before Autodesk, Tim found his passion for manufacturing while traveling the world as a UCI World Cup Mountain Bike Mechanic and was also involved in several bicycle development teams. Tim left the bicycle industry to join the manufacturing industry which fed his passion for developing and making things.

In the past 20 years, Tim has enjoyed working for large and small shops, and owning his own business, and starting a shop for top tier defense contractor L3 Communications. Tim also writes the "CAM/CAM Perspective" column in the CNC West trade magazine.

Fusion 360 Turning - Defining your Turning Setup

Setup Tab

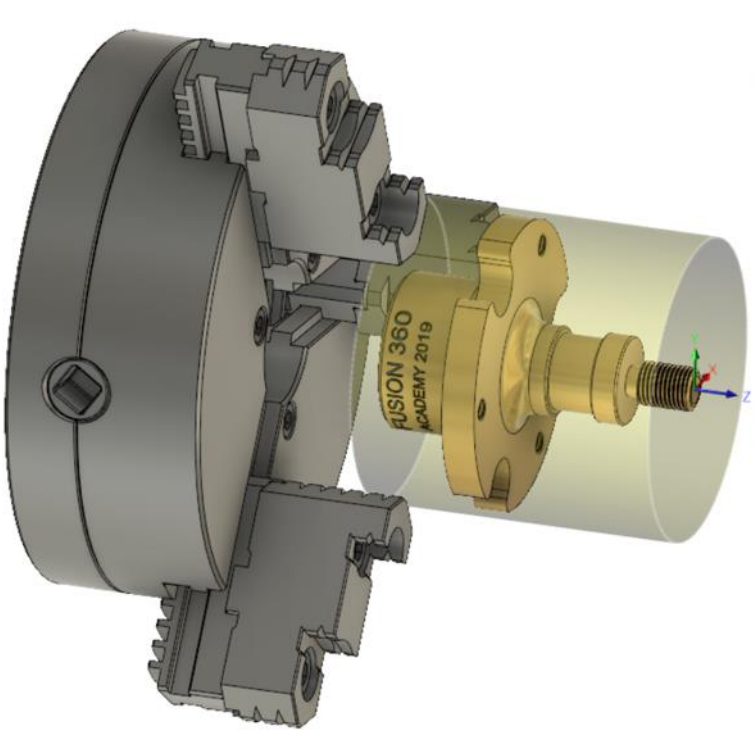
- Ensure Operation Type is set to Turning or Mill/Turn
- Define Z axis (rotary axis) with visual blue arrow direction
- Determine Origin position (program zero)

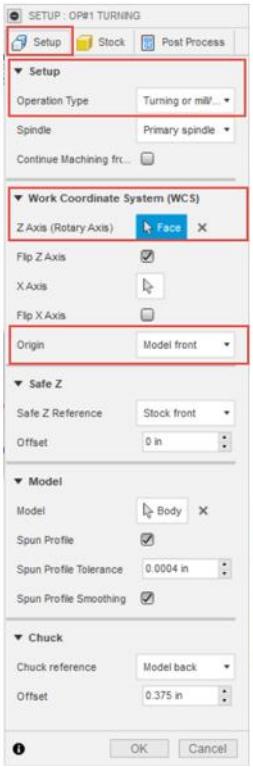
Stock Tab

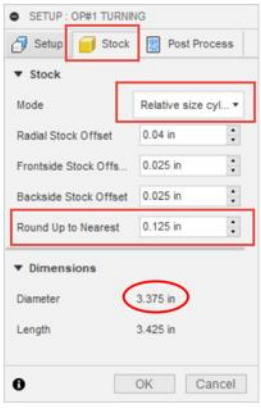
- Establish stock definition/size– relative size cylinder with “round up to nearest” option
- Set Stock offsets; Radial, Front side, Back side

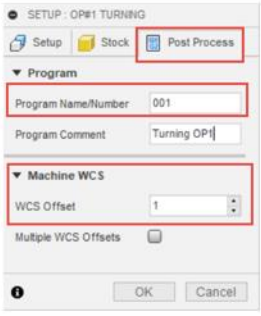
Post Process Tab

- Decide on a program number
- Define Program Comment, program name that can be identified on machine controller
- Set WCS Offset to use in Gcode program (ex. G54, G55, etc.)

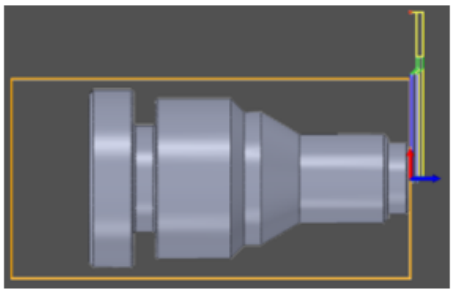






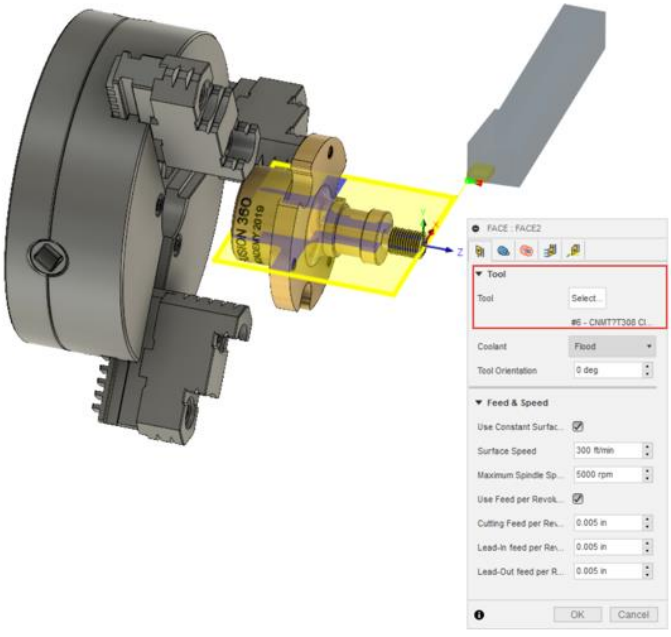


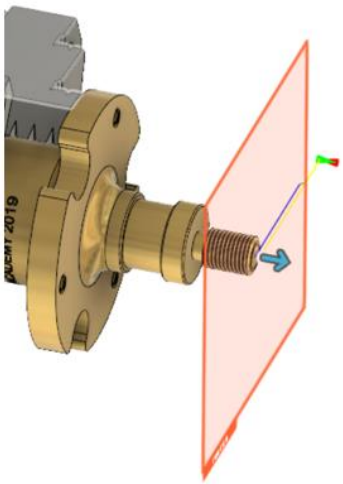
Fusion 360 Turning - Facing Operation

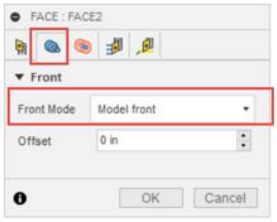


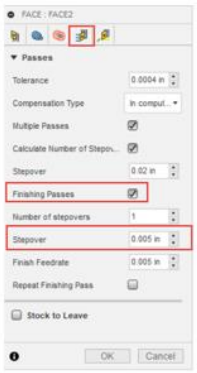
The face strategy is used for machining the front side of the part.

- Select a Facing Tool
- Applies face operation automatically to model front face, no geometry selection required
- In passes tab, enable 0.005” finishing pass



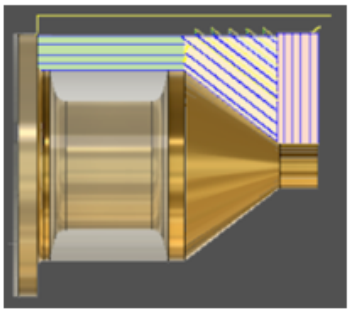






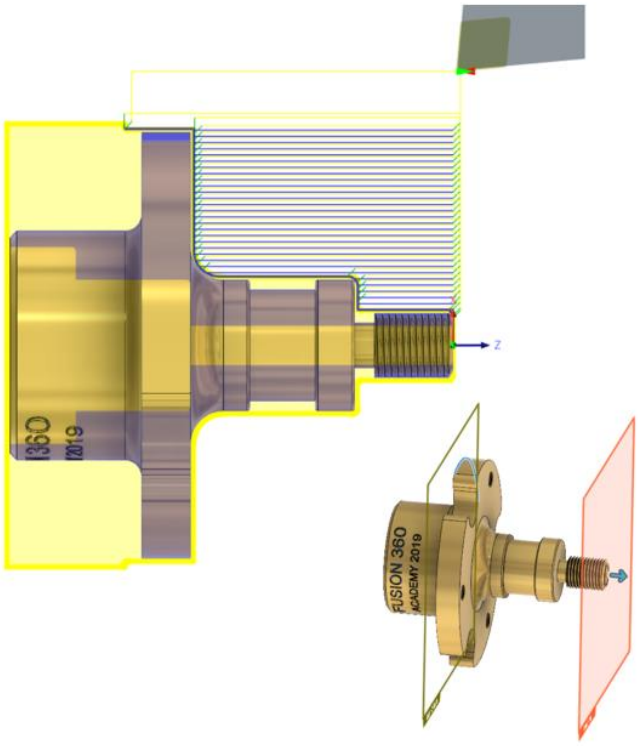
AUTODESK®
FUSION 360™
ACADEMY

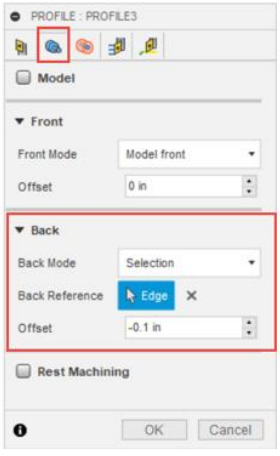
Fusion 360 Turning - Profile (Roughing) Operation

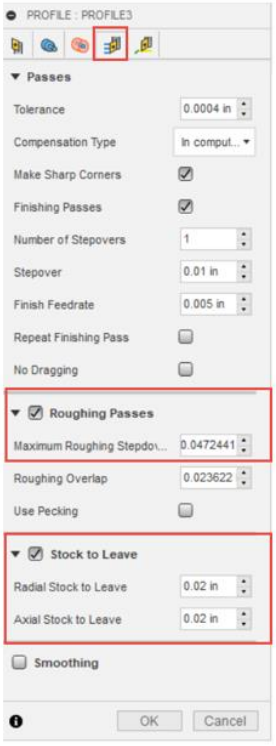


The turning profile strategy is used for both Rough and Finish toolpaths. Roughing can be applied at any angle, shown here as 3 separate roughing toolpaths.

- Select a profiling tool, can use previous facing tool for aggressive material rouging
- On tooling tab, restrict undercut toolpath motion, don't allow grooving
- Adjust back mode reference to define target machining area
- Determine maximum roughing passes
- Enable stock to leave both radially and axially

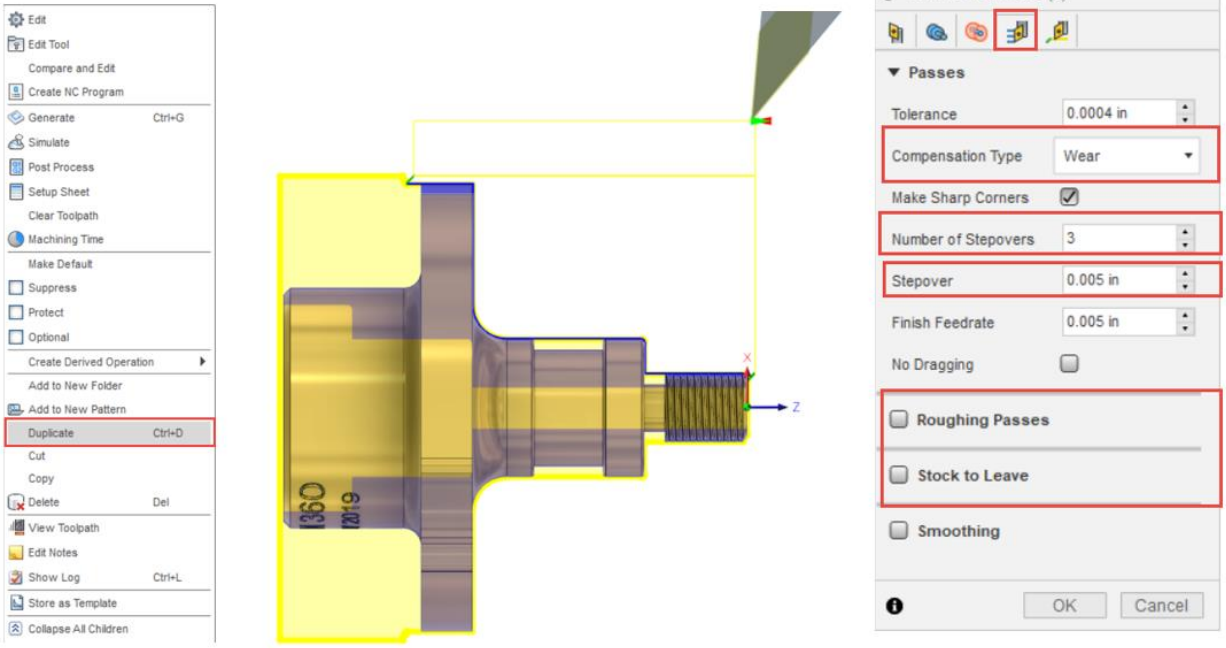




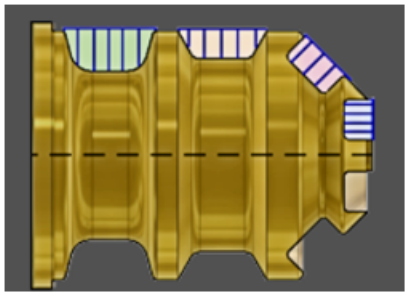


Fusion 360 Turning - Profile (Finishing) Operation

- Right-click on profile roughing operation, Duplicate existing profile roughing operation
- Select a finish profiling tool
- In the passes tab, disable roughing passes, and remove stock to leave
- Allow 3 finish passes with 0.005” stepover
- Activate tool radius compensation for controlling part size on machine controller

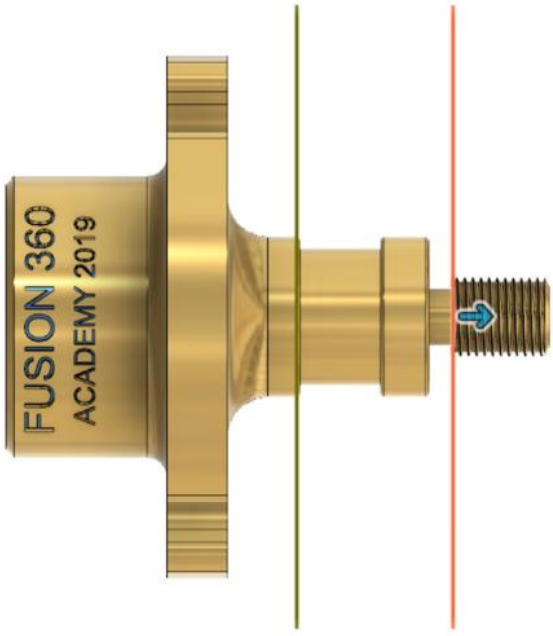


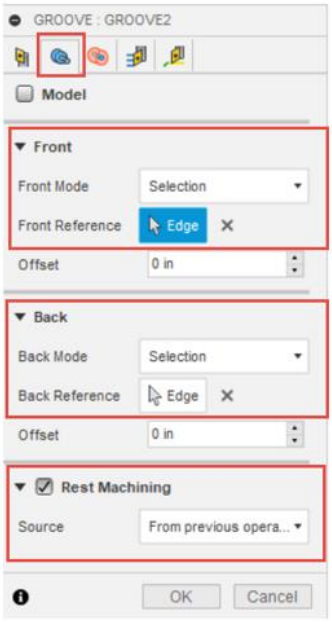
Fusion 360 Turning - Groove Operation

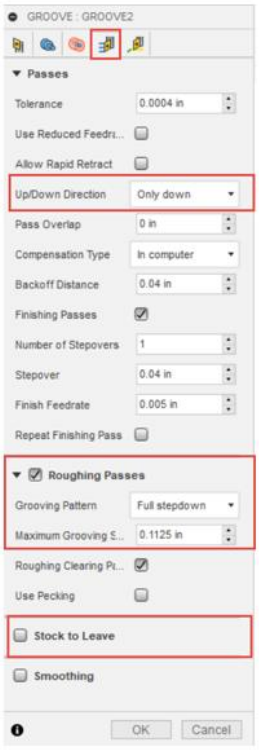


The turning groove strategy is used for both roughing and finishing of the part using groove tools.

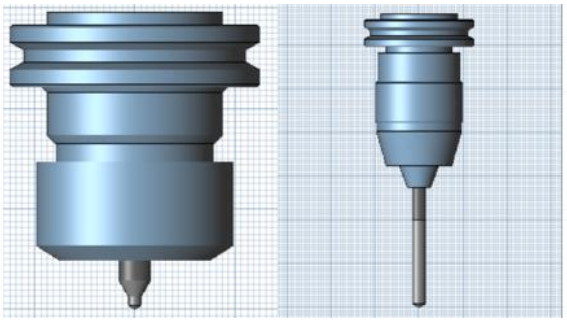
- Select a grooving tool from sample library and adjust insert width to 0.125”
- In geometry tab, enable rest machining from previous operations and hit okay
- Add both Front and Back mode control by selecting edges to confine toolpath area
- In the passes tab, define down direction only for controlling direction of the finish passes





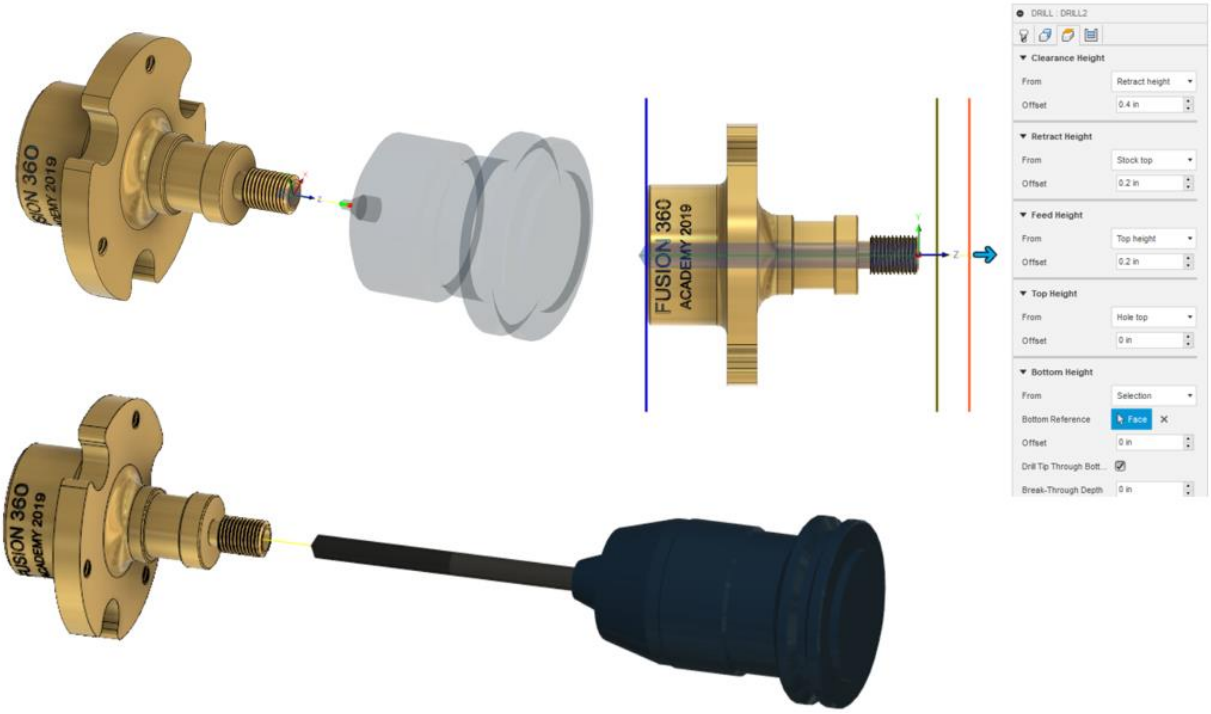


Fusion 360 Turning - Center Drill / Drill Operation



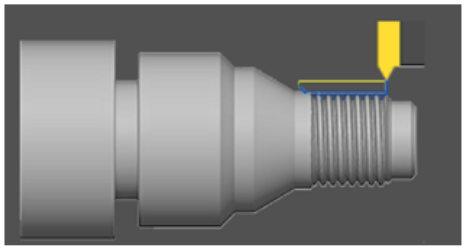
The drilling operation provides access to a wide range of drilling, tapping and hole making operations.

- Create a new drilling operation and select a #4 center drill and set depth to -0.25” from hole top in the heights tab
- Create another drilling operation selecting a 5/16” drill
- Set drilling depth from back face of part enabling drill tip through bottom
- In cycles tab, select a deep hole drilling canned cycle such as G83 peck drilling



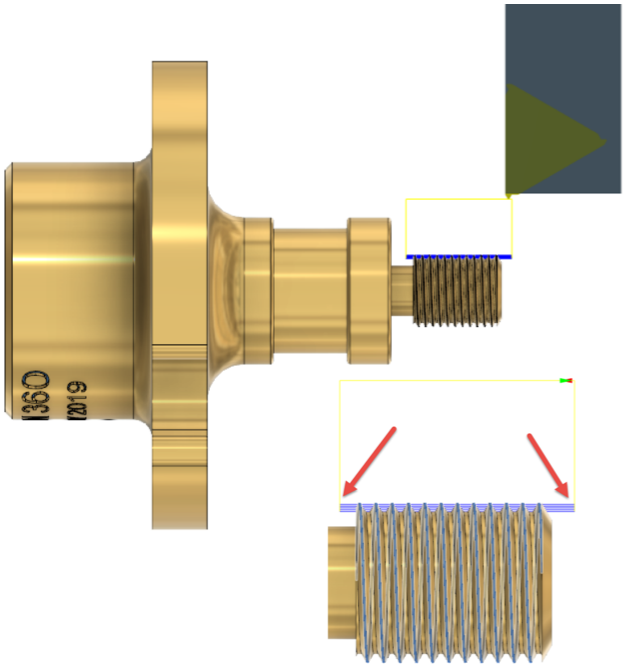
AUTODESK® FUSION 360™ ACADEMY

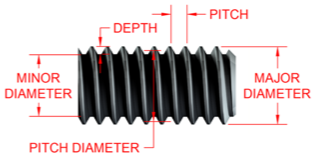
Fusion 360 Turning - Threading Operation

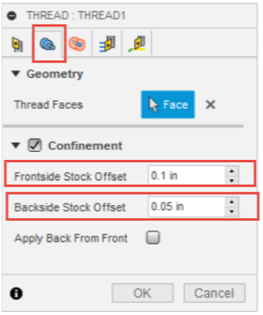


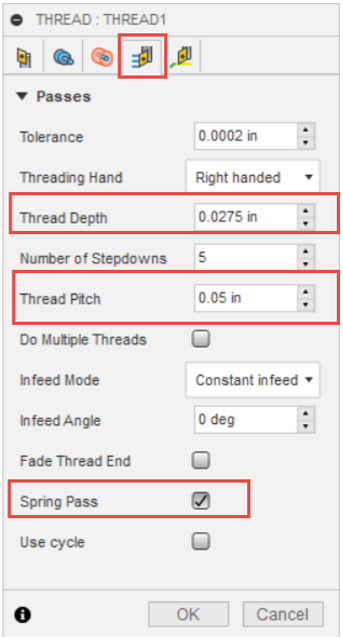
The turning thread strategy is used for turning threads. Both cylindrical and conical threads are supported. The CNC control must have built-in support for synchronizing the spindle and feed.

- Create a new turning thread operation and select a threading insert tool
- Adjust tool insert pitch geometry to 0.050” to cut a ½-20 thread form
- Adjust frontside and backside stock offset confinement to extend thread lead in and out
- Ensure thread pitch is entered to match require thread callout
- Determine thread depth, the difference between major and minor thread diameters
- Activate Spring passes, recommended to reduce tool deflection during thread cutting

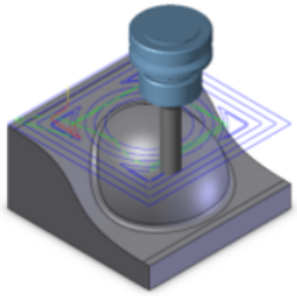








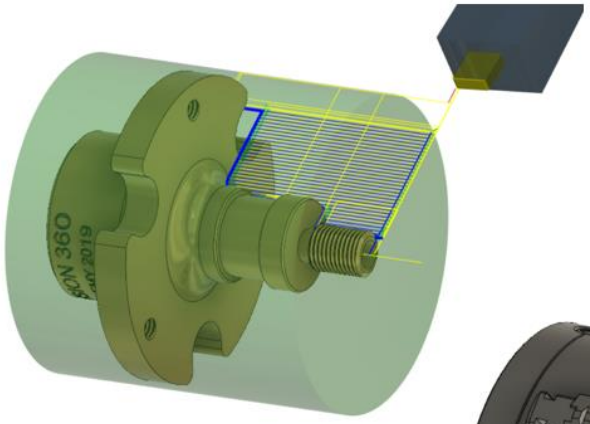
Fusion 360 Turning - Validate with Stock Simulation

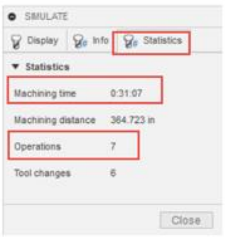


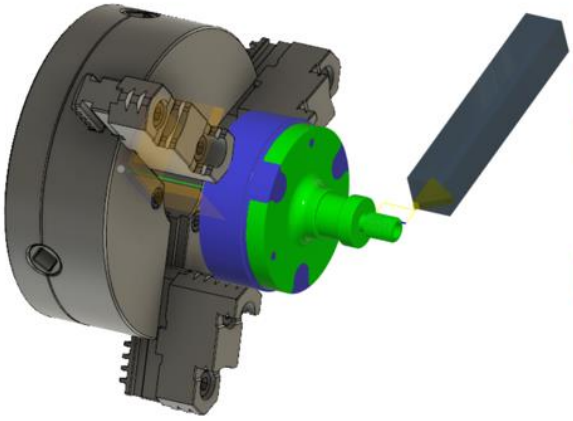
Stock Simulation

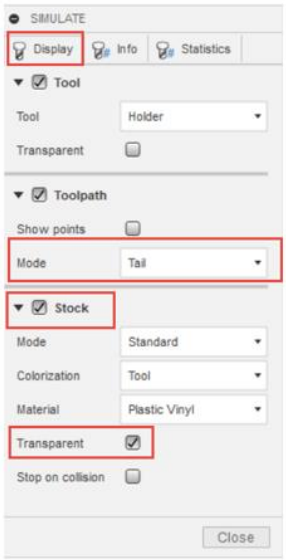
Previews and simulates toolpaths and stock material removal. Various controls include simulation speed and direction, visibility of tool, shaft and tool holder, as well as the coloring of rapid moves, lead moves, and cutting moves.

- Simulate all toolpaths to validate machining process and workflow
- Select full stock check box, and enable transparency
- Adjust toolpath mode to tail









Fusion 360 Turning - Setup Sheet Documentation

Setup Sheet for Program 001

PROGRAM COMMENT: Turning OP1
JOB DESCRIPTION: Op#1 Turning
DOCUMENT PATH: Fusion Academy Turning v20

Setup

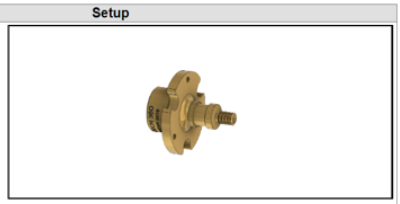
WCS: #1

STOCK:
DX: 3.375in
DY: 3.375in
DZ: 3.425in

PART:
DX: 3.185in
DY: 3.25in
DZ: 3.375in

STOCK LOWER IN WCS #1:
X: -1.687in
Y: -1.687in
Z: -3.4in

STOCK UPPER IN WCS #1:
X: 1.687in
Y: 1.687in
Z: 0.025in



Setup Sheets
Generates an overview of the NC program for the CNC operator. Provides tool data, stock and work piece position, and machining statistics. The setup sheet is fully customizable.

Total

NUMBER OF OPERATIONS: 7
NUMBER OF TOOLS: 6
TOOLS: **T6 T7 T8 T9 T10 T11**
MAXIMUM Z: 0.625in
MINIMUM Z: -3.469in
MAXIMUM FEEDRATE: 29in/min
MAXIMUM SPINDLE SPEED: 3670rpm
CUTTING DISTANCE: 92.986in
RAPID DISTANCE: 271.49in
ESTIMATED CYCLE TIME: 29m:52s

Tools

T6 D0
TYPE: general turning
INSERT: ISO C 80deg
INSCRIBED CIRCLE: 0.381in
NOSE RADIUS: 0.031in
CROSS SECTION: T
TOLERANCE: M
RELIEF: N 0deg
COMPENSATION: Tip tangent
DESCRIPTION: CNMT Right Hand
MINIMUM Z: -2.506in
MAXIMUM FEED: 25in/min
MAXIMUM SPINDLE SPEED: 500rpm
CUTTING DISTANCE: 68.375in
RAPID DISTANCE: 64.951in
ESTIMATED CYCLE TIME: 24m:5s (80.7%)
HOLDER: ISO L Right

T7 D0
TYPE: groove turning
INSERT: Square
WIDTH: 0.125in
NOSE RADIUS: 0.01in
COMPENSATION: Tip
DESCRIPTION: OD Grooving
MINIMUM Z: -1.57in
MAXIMUM FEED: 16.4643in/min
MAXIMUM SPINDLE SPEED: 500rpm
CUTTING DISTANCE: 4.147in
RAPID DISTANCE: 7.107in
ESTIMATED CYCLE TIME: 39s (2.1%)
HOLDER: External Right

T8 D0
TYPE: general turning
INSERT: ISO V 35deg
INSCRIBED CIRCLE: 0.381in
NOSE RADIUS: 0.008in
CROSS SECTION: T
TOLERANCE: M
RELIEF: N 0deg
COMPENSATION: Tip tangent
DESCRIPTION: VNMT Right Hand
MINIMUM Z: -2.483in
MAXIMUM FEED: 13.64975in/min
MAXIMUM SPINDLE SPEED: 500rpm
CUTTING DISTANCE: 11.809in
RAPID DISTANCE: 12.503in
ESTIMATED CYCLE TIME: 3m:14s (10.8%)
HOLDER: ISO L Right

Operations

Operation 1/7
DESCRIPTION: Face2
STRATEGY: Turning Face
WCS: #1
TOLERANCE: 0in
MAXIMUM STEPOVER: 0.02in
MAXIMUM Z: 0.062in
MINIMUM Z: 0in
SURFACE SPEED: 300ft/min
FEEDRATE PER REV: 0.005in
CUTTING DISTANCE: 4.045in
RAPID DISTANCE: 4.598in
ESTIMATED CYCLE TIME: 1m 19s (4.4%)
COOLANT: Flood
T6 D0
TYPE: general turning
INSERT: ISO C 80deg
INSCRIBED CIRCLE: 0.381in
NOSE RADIUS: 0.031in
CROSS SECTION: T
TOLERANCE: M
RELIEF: N 0deg
COMPENSATION: Tip tangent
DESCRIPTION: CNMT Right Hand

Operation 2/7
DESCRIPTION: Profile3
STRATEGY: Turning Profile
WCS: #1
TOLERANCE: 0in
STOCK TO LEAVE: 0.02in
MAXIMUM STEPDOWN: 0.047in
MAXIMUM STEPOVER: 0.01in
MAXIMUM Z: 0.057in
MINIMUM Z: -2.506in
SURFACE SPEED: 300ft/min
FEEDRATE PER REV: 0.005in
CUTTING DISTANCE: 64.33in
RAPID DISTANCE: 60.353in
ESTIMATED CYCLE TIME: 22m:47s (76.3%)
COOLANT: Flood
T6 D0
TYPE: general turning
INSERT: ISO C 80deg
INSCRIBED CIRCLE: 0.381in
NOSE RADIUS: 0.031in
CROSS SECTION: T
TOLERANCE: M
RELIEF: N 0deg
COMPENSATION: Tip tangent
DESCRIPTION: CNMT Right Hand

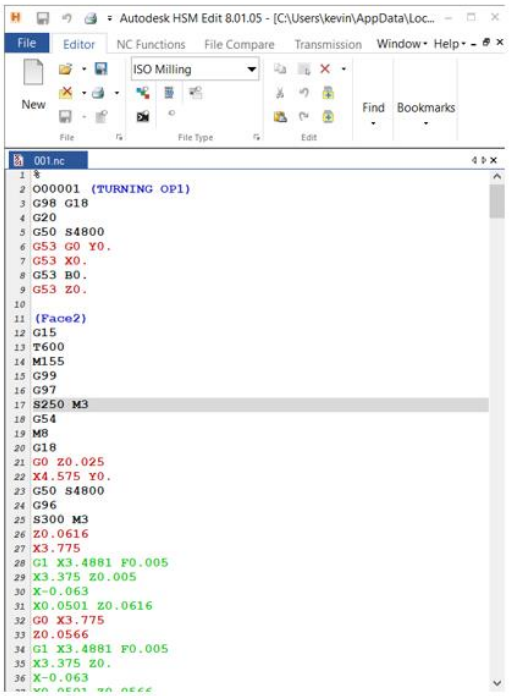
Operation 3/7
DESCRIPTION: Profile3 (2)
STRATEGY: Turning Profile
WCS: #1
TOLERANCE: 0in
STOCK TO LEAVE: 0in
MAXIMUM STEPOVER: 0.005in
COMPENSATION: wear (center)
SAFE TOOL DIAMETER: < 0in
MAXIMUM Z: 0.057in
MINIMUM Z: -2.483in
SURFACE SPEED: 300ft/min
FEEDRATE PER REV: 0.005in
CUTTING DISTANCE: 11.809in
RAPID DISTANCE: 12.503in
ESTIMATED CYCLE TIME: 3m:14s (10.8%)
COOLANT: Flood
T8 D0
TYPE: general turning
INSERT: ISO V 35deg
INSCRIBED CIRCLE: 0.381in
NOSE RADIUS: 0.008in
CROSS SECTION: T
TOLERANCE: M
RELIEF: N 0deg
COMPENSATION: Tip tangent
DESCRIPTION: VNMT Right Hand

Fusion 360 Turning - Post Processing G-Code

Post Processing

Converts the machine-independent cutter location data into machine-specific NC code.

Customizable post processor configurations are provided for all the commonly available CNC controls/machines.



AUTODESK® HSM

Post Library for Fusion 360 and Autodesk HSM

https://cam.autodesk.com/hsmposts

HAAS ST-20Y

DMG Mori NLX Mill/Turn

HAAS Turning

OKUMA LB3000

SIEMENS Mill-Turn

Fusion 360 Turning - Defining your 2nd Turning Setup

Note: You can create a New Setup or Duplicate your 1st setup to reduce selections and the need to re-define stock size information.

Setup Tab

- Ensure Operation Type is set to Turning or Mill/Turn
- Define Z axis (rotary axis) with visual blue arrow direction
- Determine Origin position (program zero)

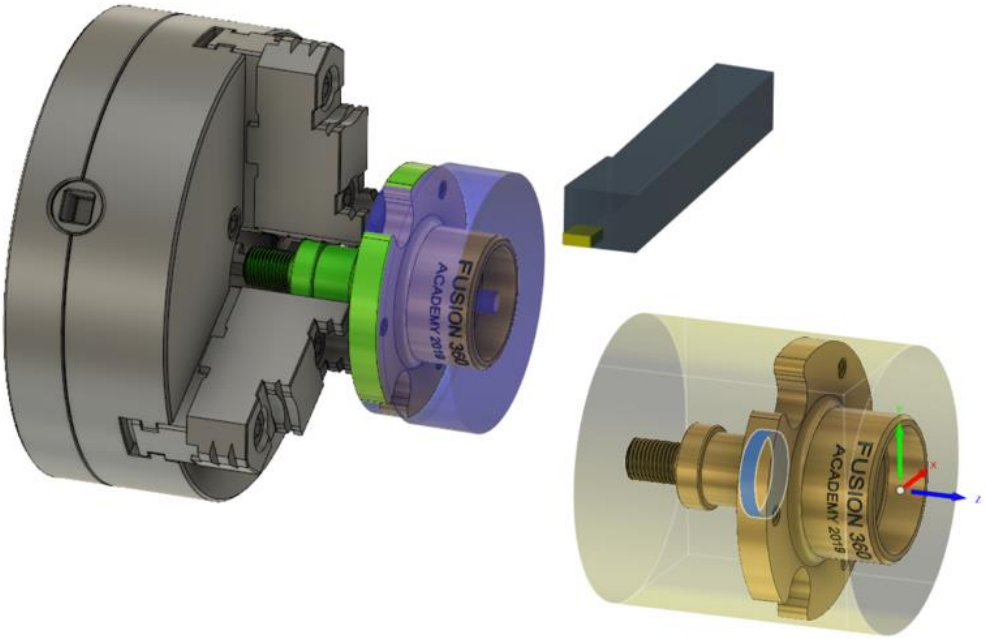
Stock Tab

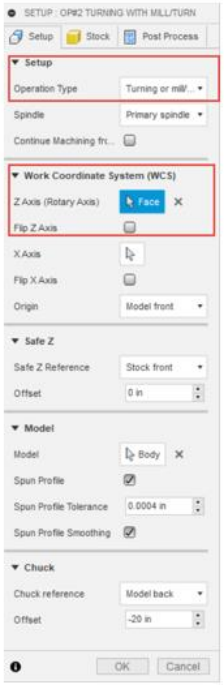
- Establish stock definition/size– relative size cylinder with “round up to nearest” option
- Set Stock offsets; Radial, Front side, Back side

Note: When Simulating 1st side operation, you can save machined state as an STL model that can be used as reference for simulating 2nd side operations.

Post Process Tab

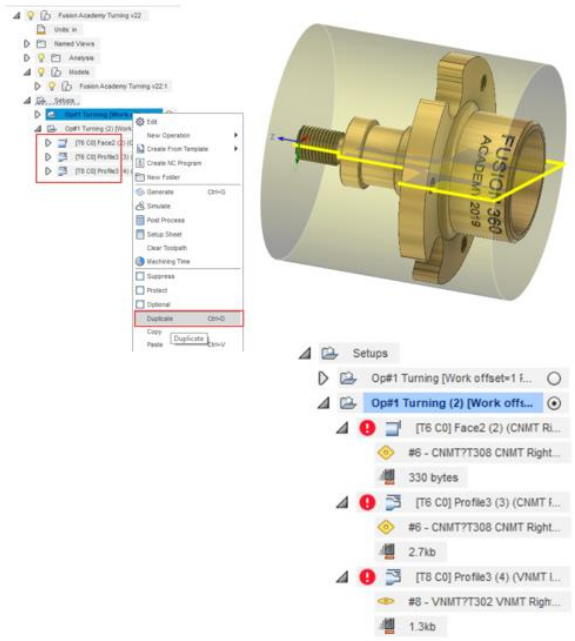
- Decide on a program number
- Define Program Comment, program name that can be identified on machine controller
- Set WCS Offset to use in Gcode program (ex. G54, G55, etc)





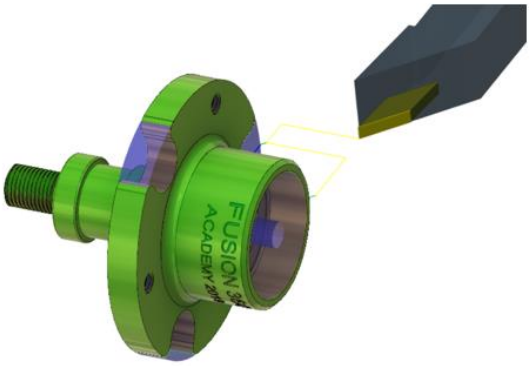
Fusion 360 Turning - Duplicated Setup Advantages

- Right-click, and duplicate Setup1
- Redefine Z axis alignment (blue arrow direction)
- Regenerate Facing, and both profiling operations
- Adjust Back Mode reference on the Geometry tab on both profiling operations



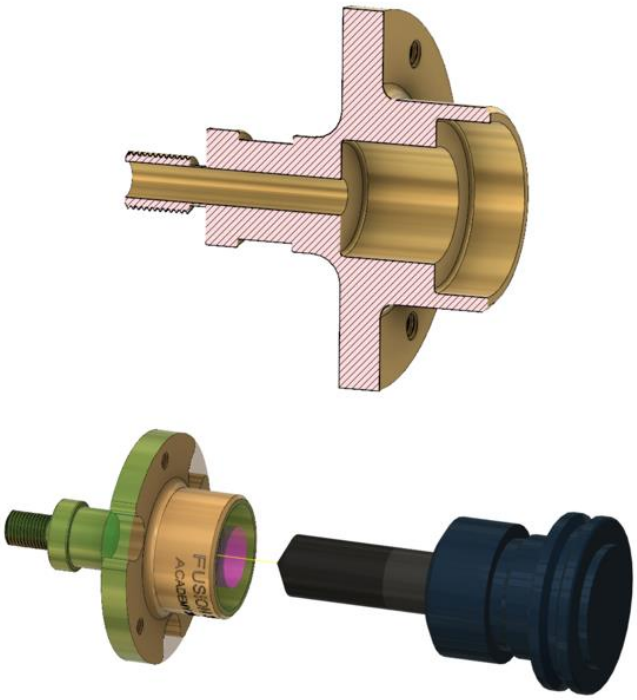
"Right-Click" Duplicate

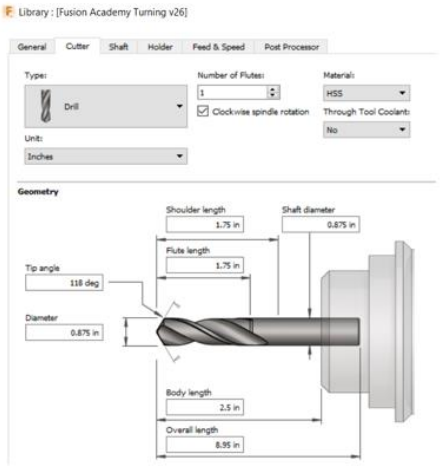
There are several advantages in Duplicating operations such as reduced selections for WCS alignment and stock sizing. In addition, you can quickly reuse deployed toolpaths from first side operations.



Fusion 360 Turning - Pre-Drill Operation

- Adding a pre-drill operation will remove material to allow clearance for a turning bore operation
- Select a 7/8” Drill from library and adjust tool body length to 2.5”
- G73 chip breaking drill cycle is effective material removal caned cycle

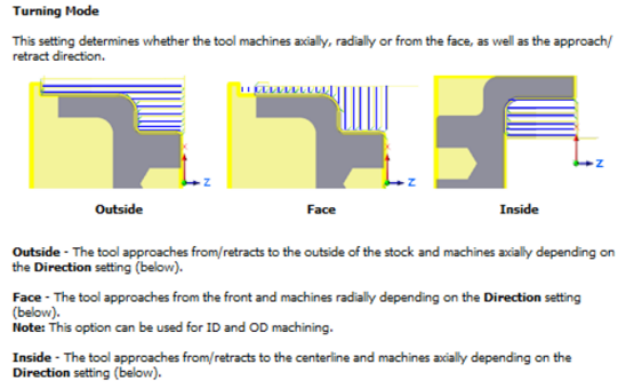


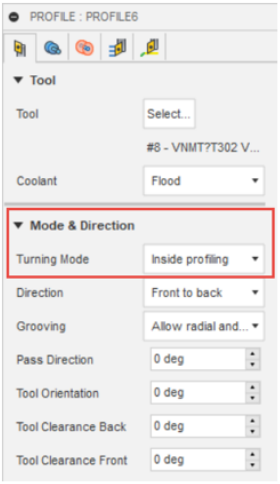


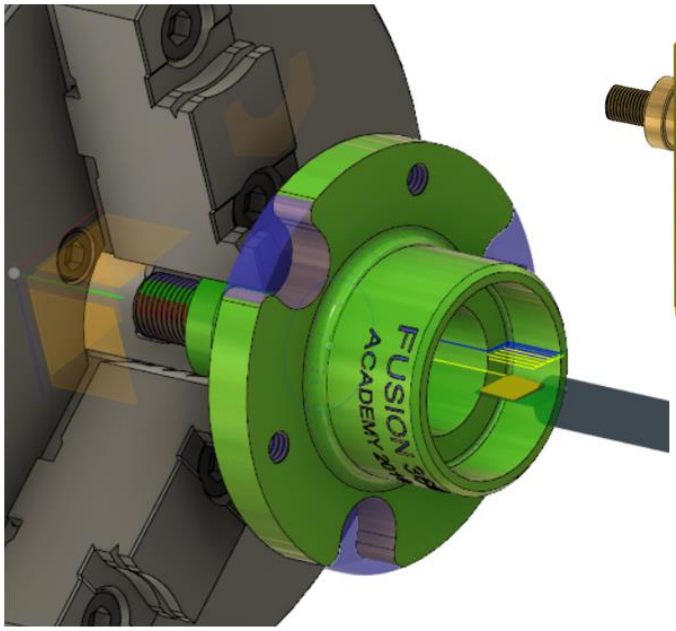
Fusion 360 Turning - Internal Bore Operation

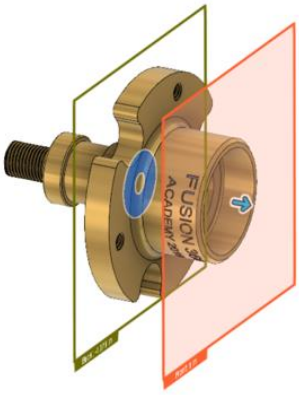
- Select an ID boring tool from library
- Adjust boring tool dimensions to ensure it will fit inside the pre-drilled 7/8” hole
- Change back mode reference to be at bottom of bore feature
- Adjust outer radius selection in heights tab by selecting larger ID bore diameter face
- Ensure inner radius is set to model ID and retract and clearance are set accordingly

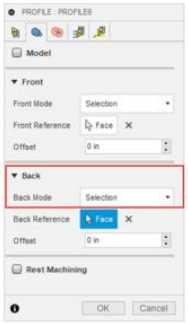
The turning profile strategy can be used for internal bore operations.

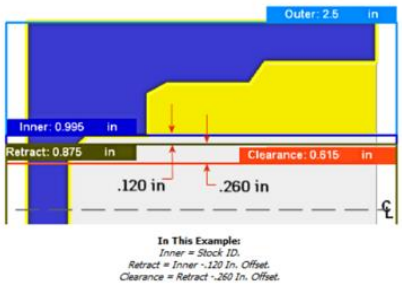








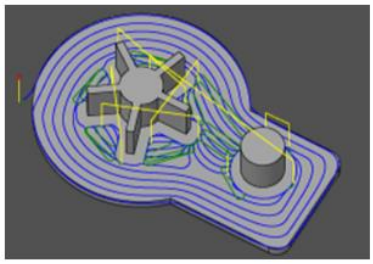




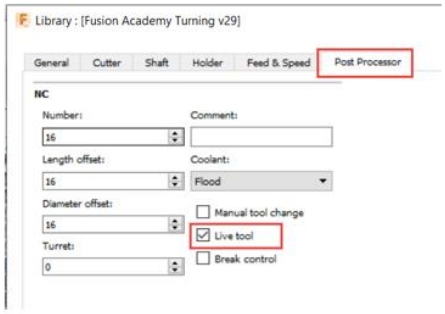
Fusion 360 Turning - Mill/Turn with Live Tooling

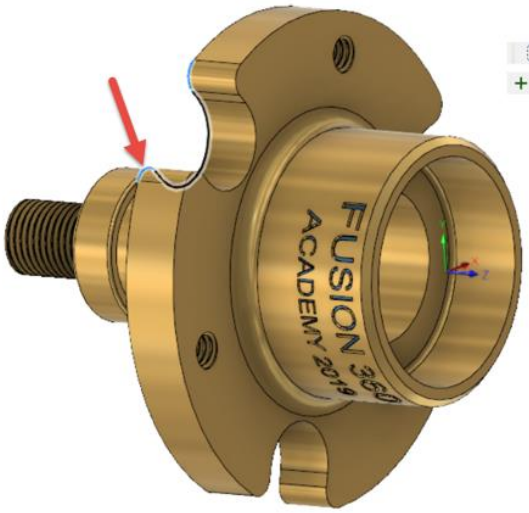
2D Adaptive

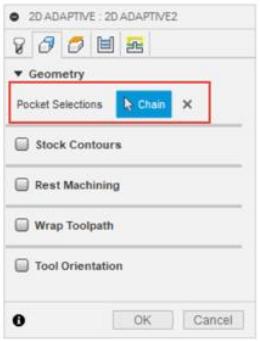
- Choose a 2d Adaptive operation using a 3/8” Endmill from sample library, ensure live tool option is activated
- Select one open edge profile of lobed feature on model
- Enable both ways in geometry tab specifying both climb and conventional milling
- Activate stock to leave allowing 0.020” radial stock to leave
- Adjust bottom height of toolpath with a -0.020” offset

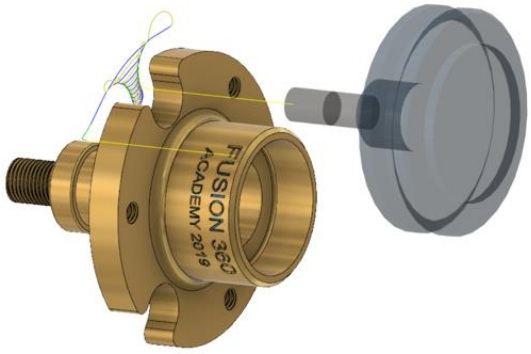


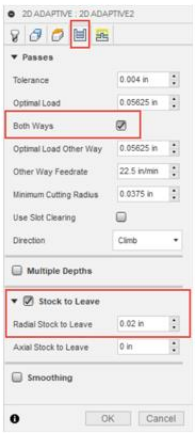
The 2D Adaptive Clearing strategy creates a roughing operation that uses a more optimized toolpath that avoids abrupt direction changes.







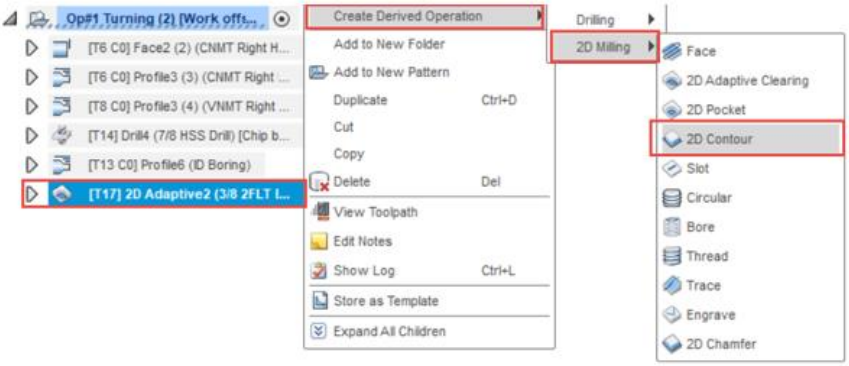


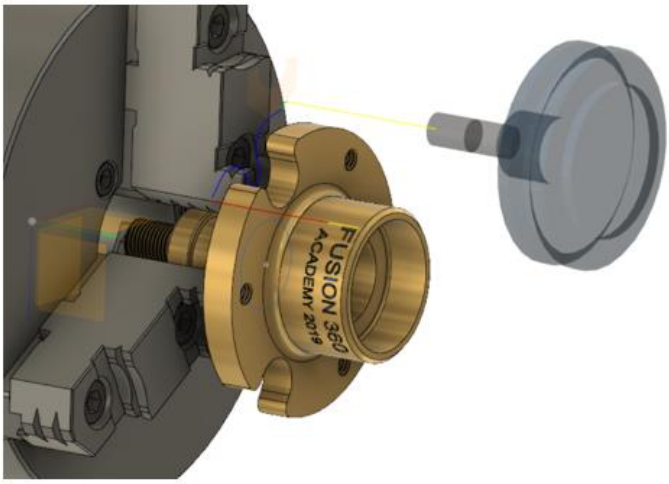


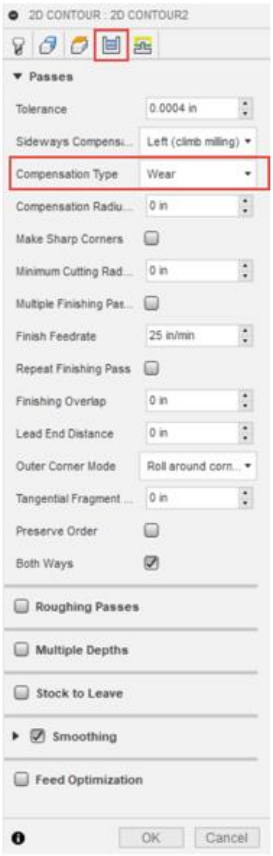
Fusion 360 Turning - Mill/Turn with Live Tooling

2D Contour (Derived Operation)

- Derive a new 2d Contour operation from previous 2d adaptive toolpath
- Derived Toolpaths lets you copy the parameters of an existing toolpath and use it in a different type of toolpath.
- Activate radius wear compensation (G41/G42) in the passes tab to allow for small adjustments on the machine controller.



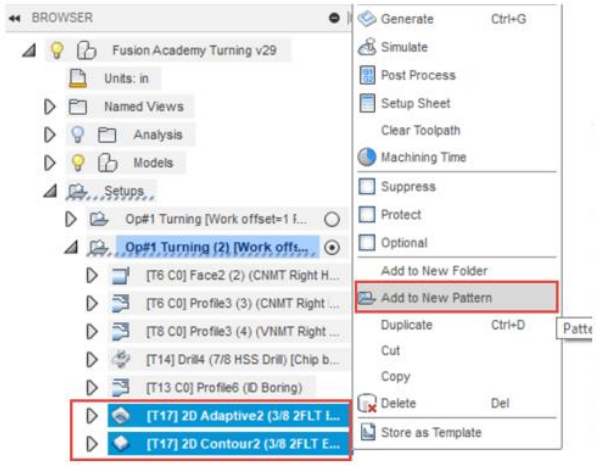


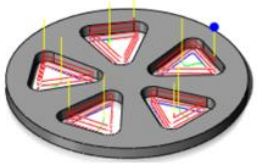


Fusion 360 Turning - Mill/Turn with Live Tooling

Circular Patterns

- Select both 2d Adaptive and 2d Contour, right-click, add to circular pattern
- Option to Order by Tool when posting g-code program





Pattern Type

The pattern type to use for the folder.

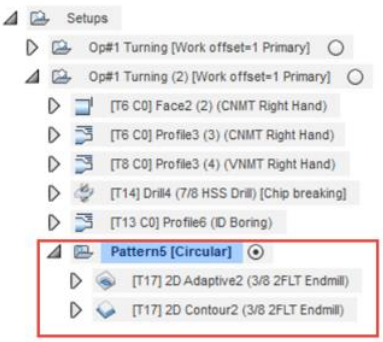
Linear - Patterns operations/toolpaths in a straight line, or grid.

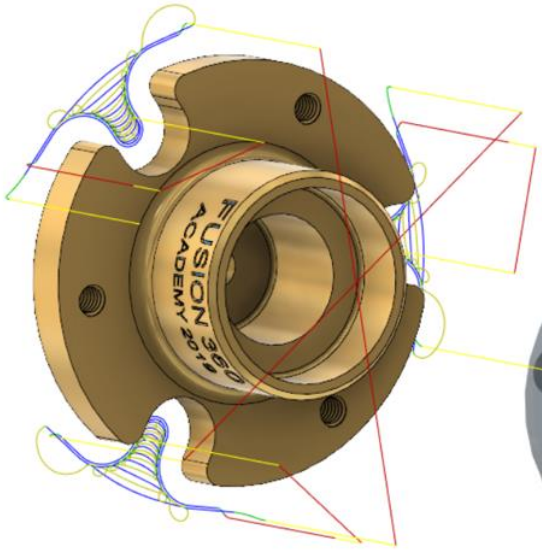
Circular - Patterns operations/toolpaths around a rotational center.

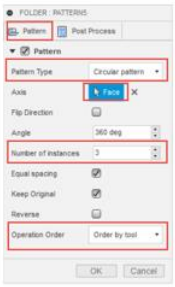
Mirror - Creates mirror-image operations/toolpaths that are flipped about a mirror plane.

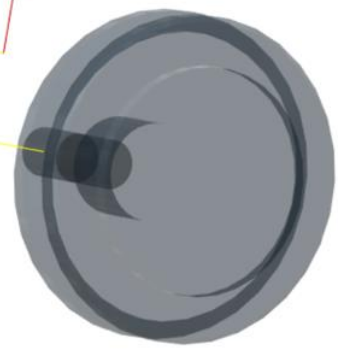
Duplication - Creates a pattern using operations/toolpaths as the source and sketch points as the targets.

Component - Available in *assemblies only*, this option patterns operations/toolpaths from one component to multiple components in an assembly.



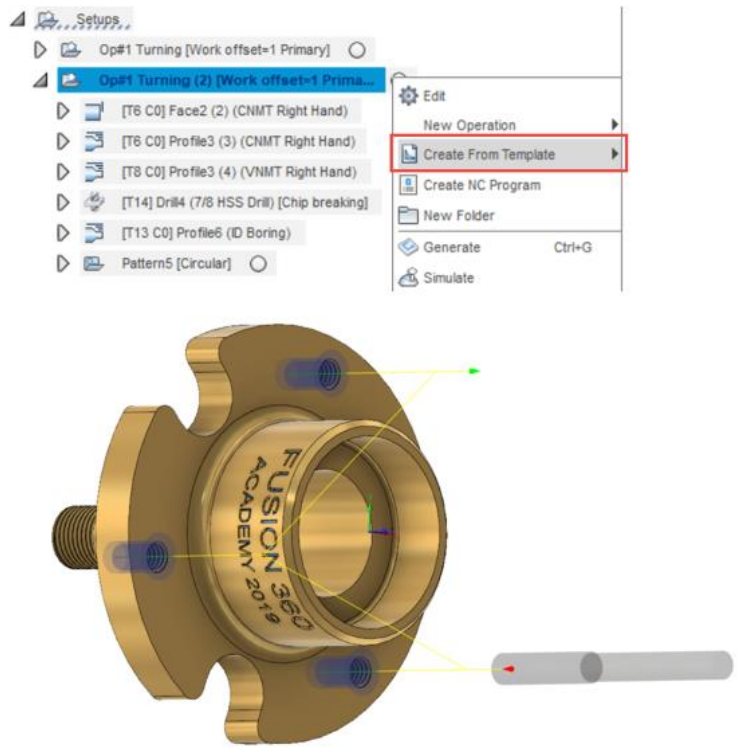


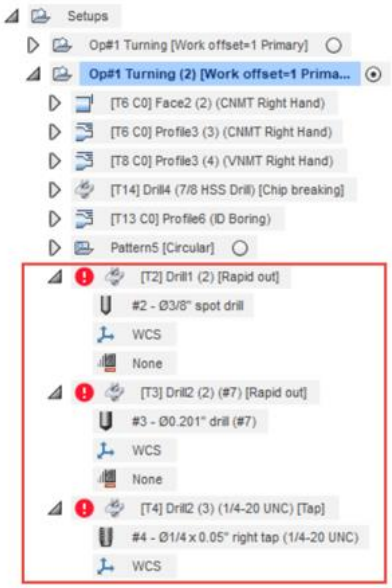




Fusion 360 Turning - Mill/Turn with Live Tooling Template Workflows (Spot, Drill, Tap)

- Applying strategic toolpath templates can save you enormous time
- Right-click on setup and create new toolpaths from a previously recorded operation
- Re-select hole faces on each operation and make any necessary adjustments

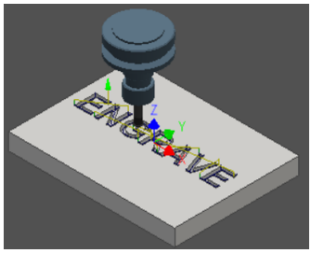




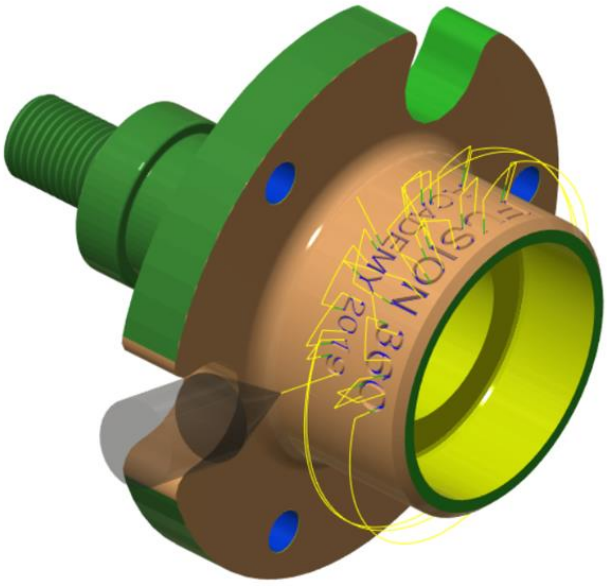
Fusion 360 Turning - Mill/Turn with Live Tooling

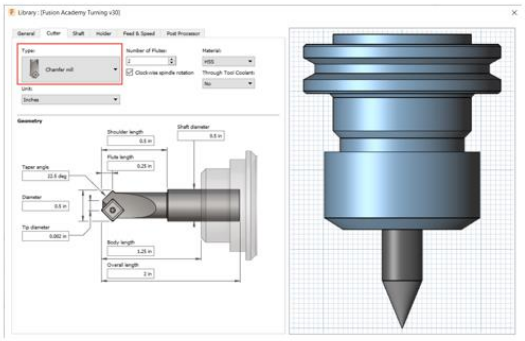
Engrave Text -Wrap Toolpath (Currently Beta)

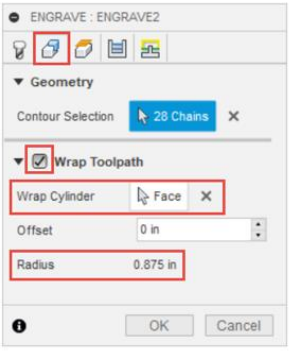
- An introductory look at what's new and coming soon with Fusion 360 Turning
- A sneak peak at Engrave Wrapped Toolpath Strategy
- To enable Beta Mode: File / View / Show Text Commands
- Enter into the text command: cam.betamode/on
- Create a 2d Engrave toolpath operation, and select a chamfer tool
- In Geometry tab, activate Wrap Toolpath toggle and select a wrap cylinder face



The Engrave toolpath strategy machines along contours with V-shaped chamfered walls. Chamfer Mill type tools are required for this toolpath.





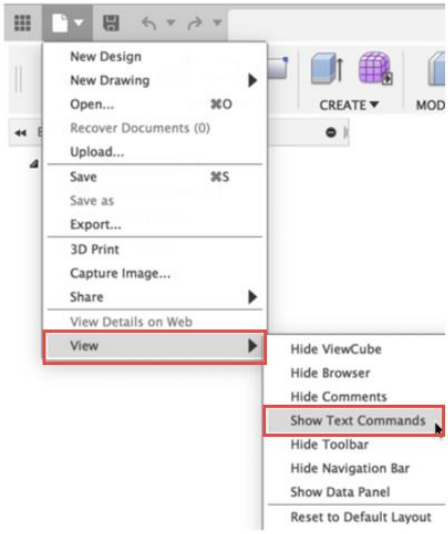


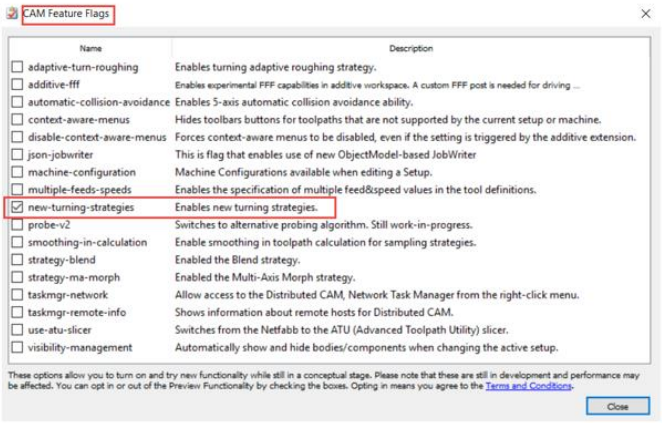
Fusion 360 Turning – What's New in Turning

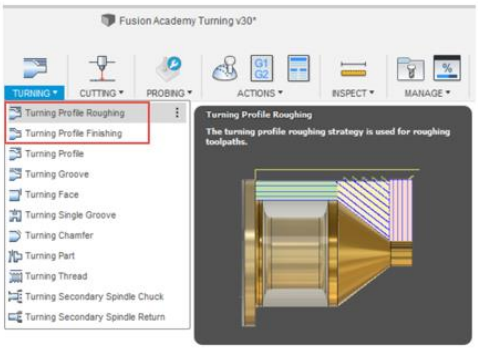
Turning Product Manager – Tim Paul

NEW! – Feature Flags for New Turning Operations

- An overview of what is coming in Fusion 360 Turning
- How to enable feature flags, open the Text Commands under View > Show Text Commands.
- Make sure the Txt radio button is enabled, and then enter: CAM.featureflags
- Activate the button for new-turning strategies







Fusion 360 Turning

- This presentation data set can be accessed with following name and link
- Fusion File Name: FUSION360 TURNING-Lee-F360ACADEMY19
- Fusion Public Link: https://a360.co/2ynSGWG

@leeverageintegration

Follow us on Instagram

@oneeartim

AUTODESK
Make anything.

AUTODESK®
FUSION 360™
ACADEMY

Fusion 360 Turning – Presentation Notes