

Simplify CAM with Fusion 360

Graham Goodier

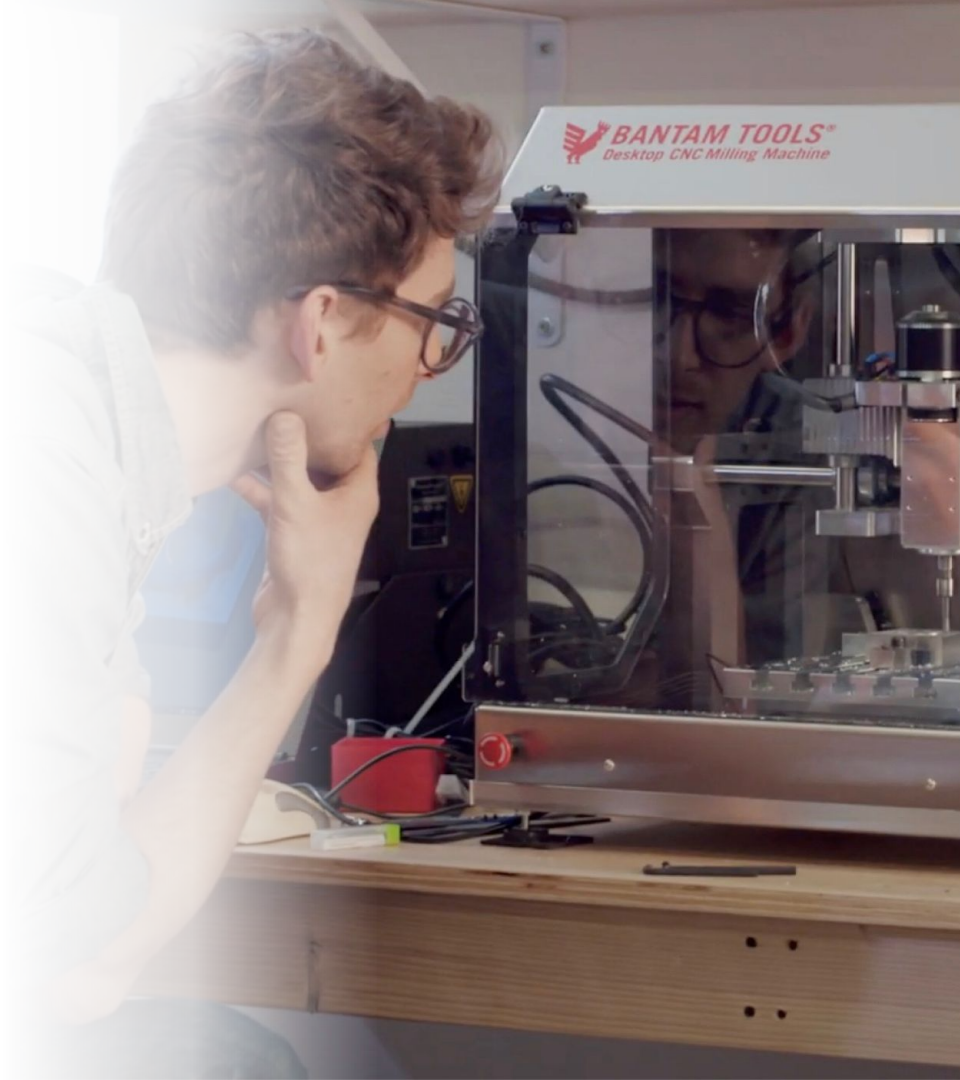
Technical Consultant

 @goodier.cam |  www.linkedin.com/in/graham-goodier

 **AUTODESK.**  **BANTAM TOOLS®**

AGENDA

- How is a Part Programmed?
- Fusion 360 Workflow
- Fusion 360 Strategies
- Machining Parameters
- Software Demo
- Machining Configuration
- Q&A



The background features a series of overlapping, curved, blue and white geometric shapes that create a sense of depth and movement. A white, semi-transparent rectangular box is positioned in the center-left, containing the text. The overall aesthetic is clean, modern, and technical.

How is a Part Programmed?

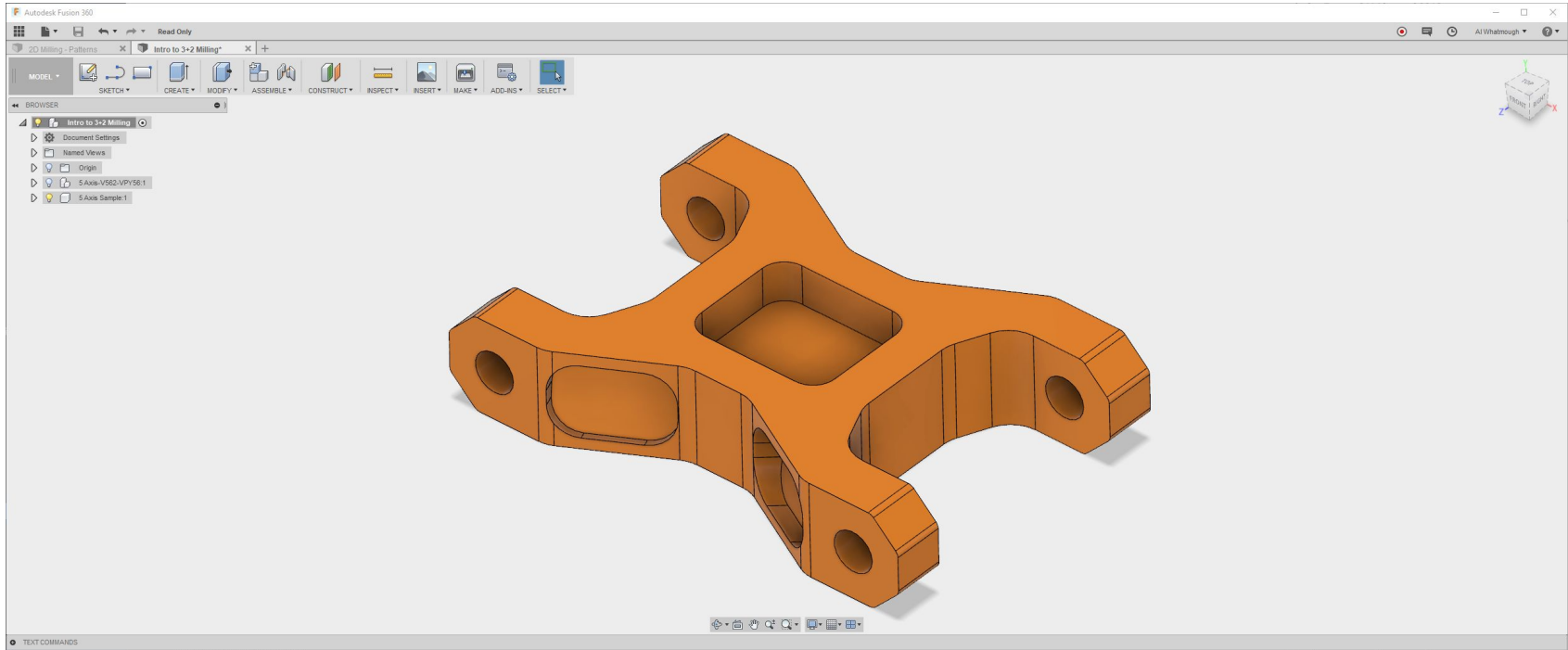
HOW IS A PART PROGRAMMED?

- A **CAD model** is produced, depicting the part to be cut
- **Stock** material is determined from the part size
- A **Setup** (Work Co-ordinate System) is determined that the machine tool uses as positional reference
- **Tools & Machining Parameters** appropriate for the job are selected
- **Toolpaths** are created based on the features of the part, taking into account the stock material
- **Simulation** of toolpath helps avoid gouges/collisions
- **NC Code** is output using a post processor
- NC Code is input into the machine controller and the part is **cut on the machine tool**



The background features a series of overlapping, semi-transparent blue and white geometric shapes, including curved planes and rectangular blocks, creating a sense of depth and movement. The colors range from light sky blue to a vibrant cyan. The shapes are arranged in a way that suggests a modern, architectural or technological environment.

Fusion 360 Workflow



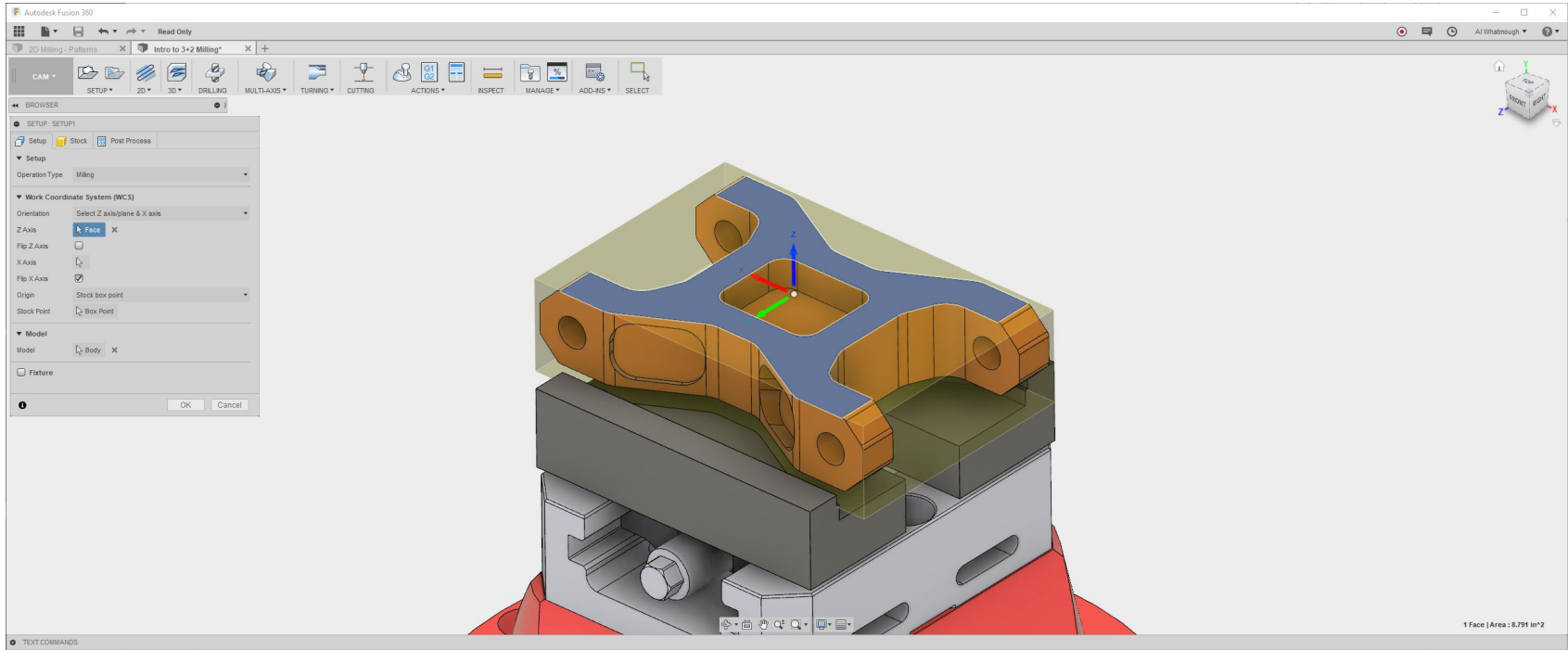
Create/Open CAD
Data

Define Job
Setup

Define
Operations

Simulate
Toolpath

NC Code



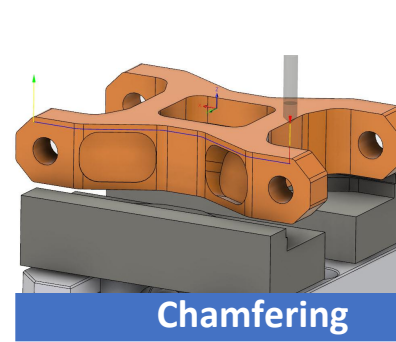
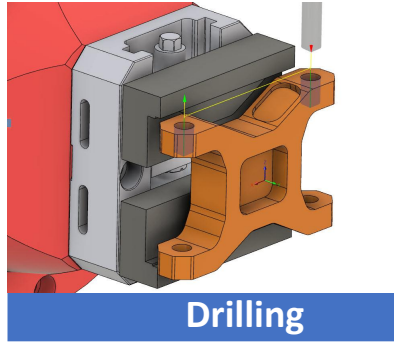
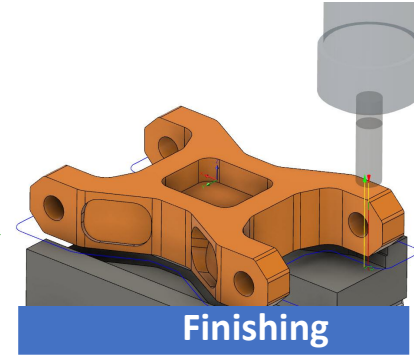
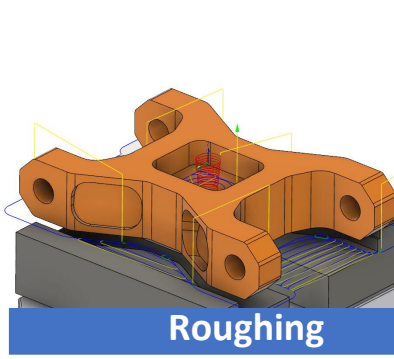
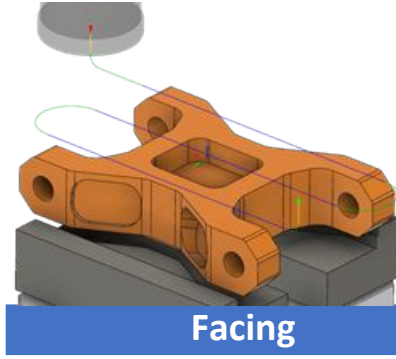
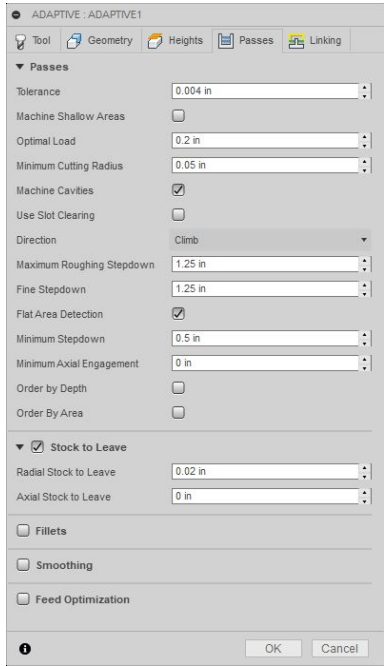
Create/Open
CAD Data

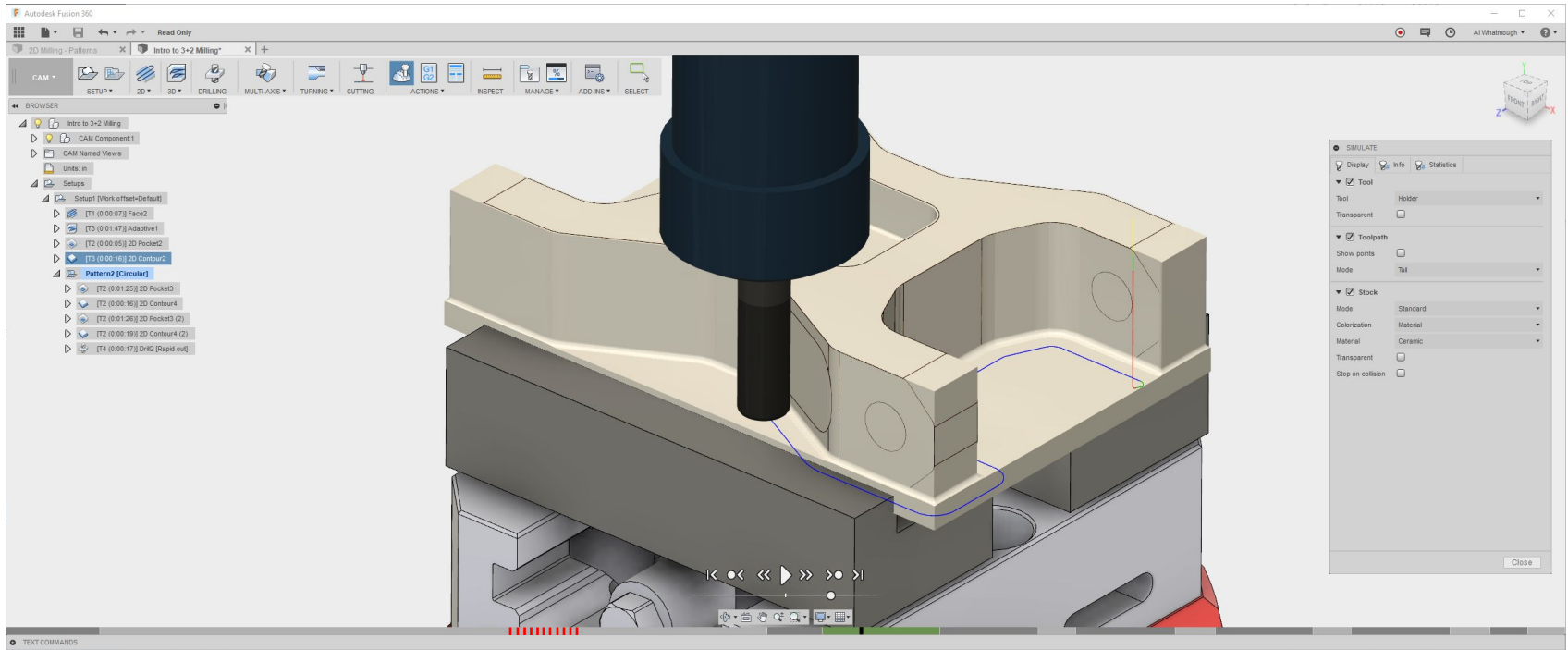
Define Job
Setup

Define
Operation

Simulate
Toolpath

NC Code





Create/Open
CAD Data

Define Job
Setup

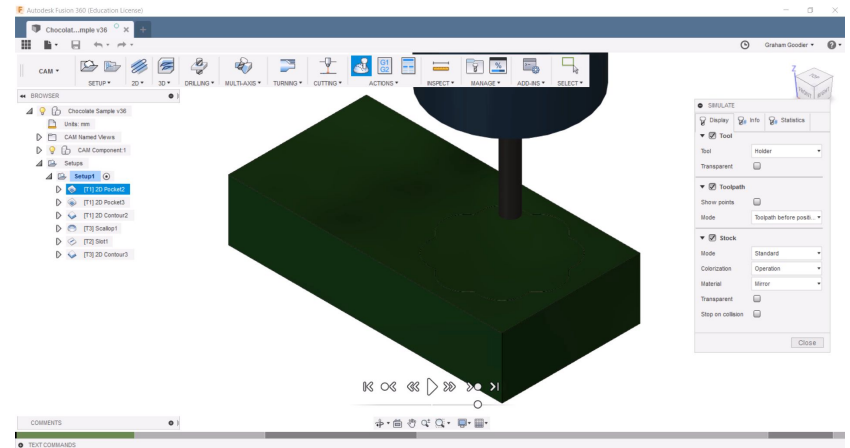
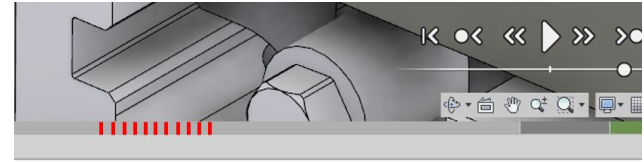
Define
Operation

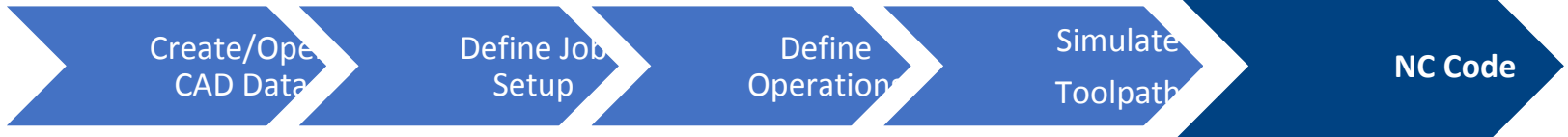
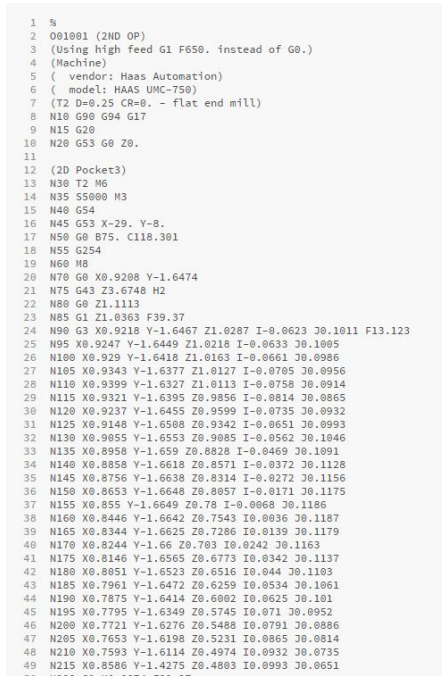
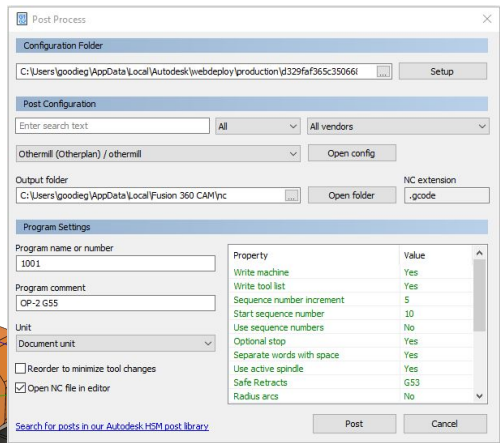
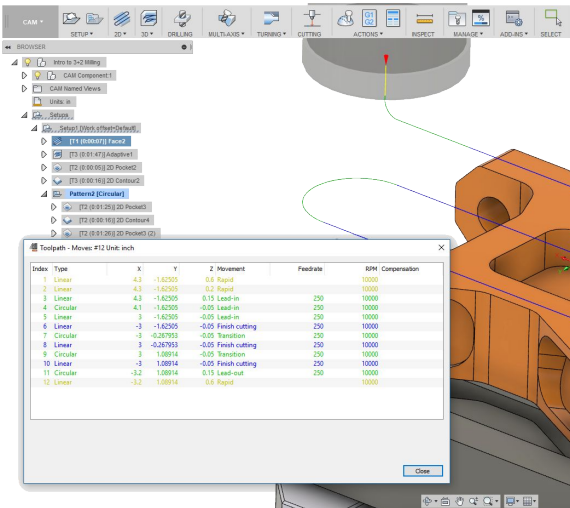
Simulate
Toolpath

NC Code

Simulation

- Before posting, simulate to check for gouges/collisions – which are shown in red
- If no vice/fixture model is present in the software, look out for potential collisions as a result of:
 - Machining the clamped sides
 - Below the bottom of the stock
 - Lead-ins/lead-outs
- Check the lead-in/ramp, along with rapid heights
- Simulation helps avoid collisions with the machine turret as well as the tool itself





Types of code

N## - Line number

G## - Motion

X## - Horizontal position

Y## - Vertical position

Z## - Depth

F## - Feed rate

S## - Spindle speed

T## - Tool selection

M## - Miscellaneous functions

I## / J## - Incremental arc center

R## - Radius of an arc

Examples of NC Code

G00 – Rapid Move from one cutting area to another

G01 - linear cut from current position to programmed X, Y, Z

G20 – Program in mm

G21 – Program in inches

G53 – Machine co-ordinate system

M00 – Program Stop

M02 – End of Program

M06 – Tool Change

M08 – Coolant on (Flood)

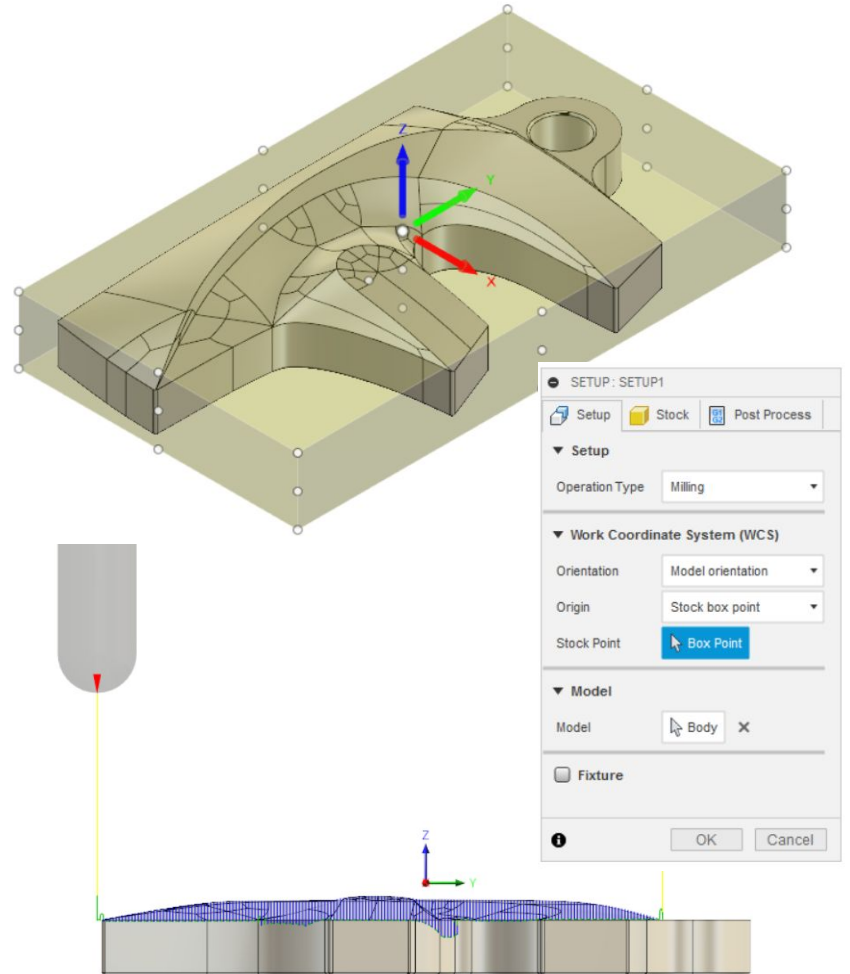


The background features a series of overlapping, semi-transparent blue and white geometric shapes, including curved planes and rectangular blocks, creating a sense of depth and movement. A white, trapezoidal text box is positioned in the center-left of the image.

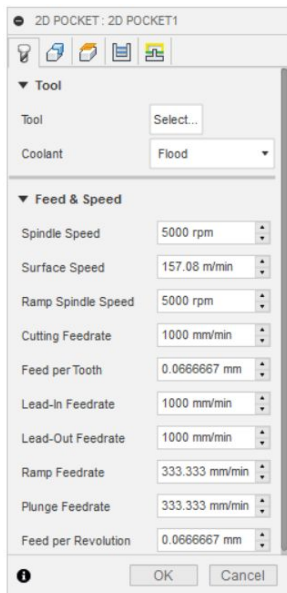
Fusion 360 Setup and Strategies

Setup & Stock

- A setup tells Fusion 360 what type of toolpaths to create (i.e. Milling, Turning)
 - In which the **Work Coordinate System (WCS)** is defined
- Tools descend in Z when approaching the part, therefore the WCS' Z axis must be parallel to the tool's Z axis travel
- More than one setup can be created to machine other faces if the machine is capable of doing so
- Fusion determines the size of your raw stock using the part dimensions, or a specific sized stock can be dictated



CAM Properties



2D POCKET : 2D POCKET1

▼ Tool

Tool: Select...

Coolant: Flood

▼ Feed & Speed

Spindle Speed: 5000 rpm

Surface Speed: 157.08 m/min

Ramp Spindle Speed: 5000 rpm

Cutting Feedrate: 1000 mm/min

Feed per Tooth: 0.066667 mm

Lead-In Feedrate: 1000 mm/min

Lead-Out Feedrate: 1000 mm/min

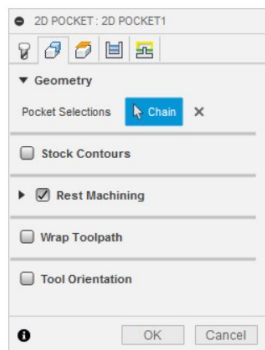
Ramp Feedrate: 333.333 mm/min

Plunge Feedrate: 333.333 mm/min

Feed per Revolution: 0.066667 mm

OK Cancel

**Tool
Properties**



2D POCKET : 2D POCKET1

▼ Geometry

Pocket Selections: Chain X

Stock Contours

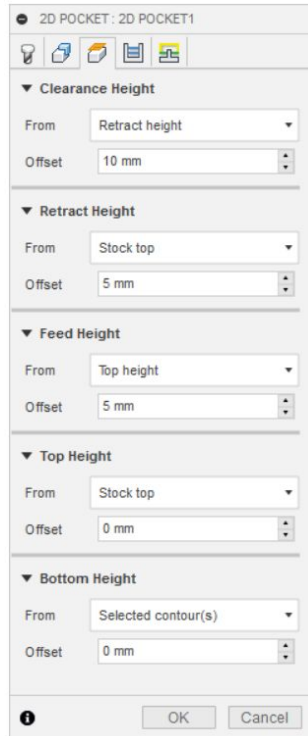
Rest Machining

Wrap Toolpath

Tool Orientation

OK Cancel

**Geometry
Properties**



2D POCKET : 2D POCKET1

▼ Clearance Height

From: Retract height

Offset: 10 mm

▼ Retract Height

From: Stock top

Offset: 5 mm

▼ Feed Height

From: Top height

Offset: 5 mm

▼ Top Height

From: Stock top

Offset: 0 mm

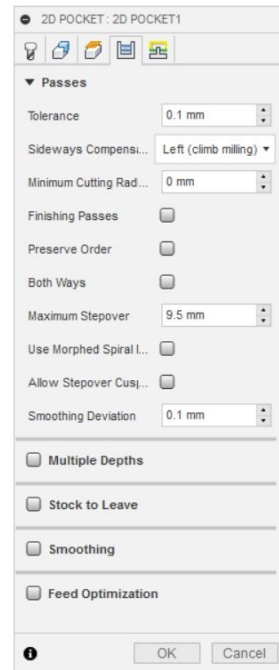
▼ Bottom Height

From: Selected contour(s)

Offset: 0 mm

OK Cancel

**Heights
Properties**



2D POCKET : 2D POCKET1

▼ Passes

Tolerance: 0.1 mm

Sideways Compens...: Left (climb milling)

Minimum Cutting Rad...: 0 mm

Finishing Passes:

Preserve Order:

Both Ways:

Maximum Stepmover: 9.5 mm

Use Morphed Spiral L...:

Allow Stepmover Cus...:

Smoothing Deviation: 0.1 mm

Multiple Depths

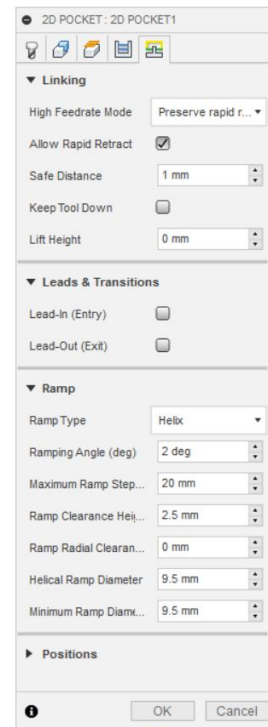
Stock to Leave

Smoothing

Feed Optimization

OK Cancel

**Passes
Properties**



2D POCKET : 2D POCKET1

▼ Linking

High Feedrate Mode: Preserve rapid r...

Allow Rapid Retract:

Safe Distance: 1 mm

Keep Tool Down:

Lift Height: 0 mm

▼ Leads & Transitions

Lead-in (Entry):

Lead-out (Exit):

▼ Ramp

Ramp Type: Helix

Ramping Angle (deg): 2 deg

Maximum Ramp Step...: 20 mm

Ramp Clearance Hei...: 2.5 mm

Ramp Radial Clearan...: 0 mm

Helical Ramp Diameter: 9.5 mm

Minimum Ramp Diam...: 9.5 mm

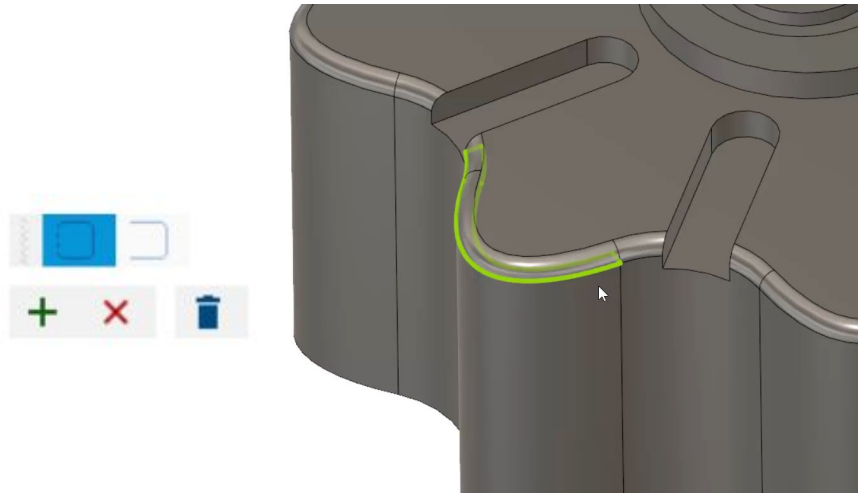
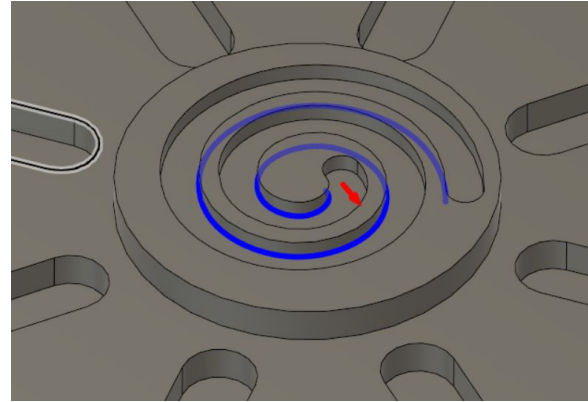
► Positions

OK Cancel

**Linking
Properties**

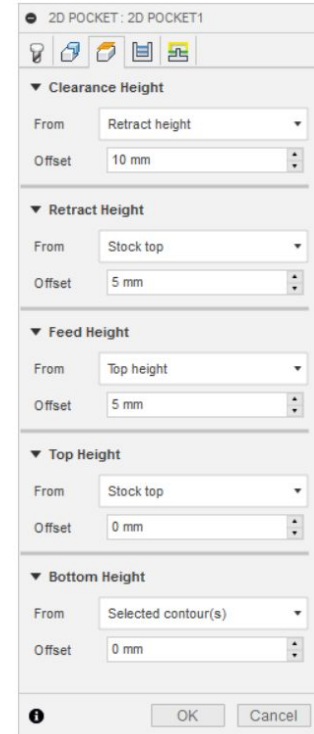
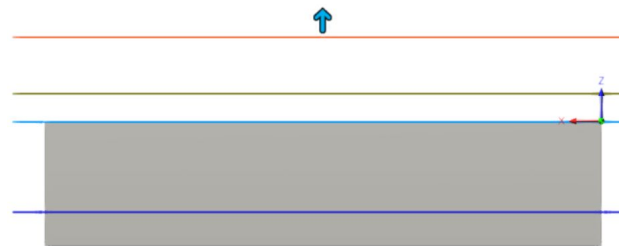
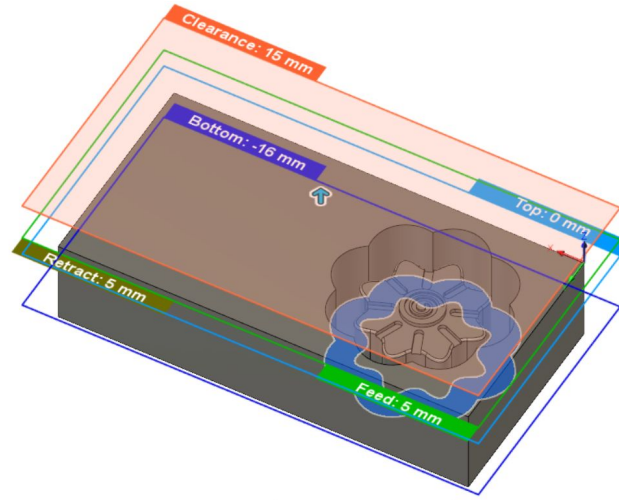
Boundary Selection

- Strategies can use machining boundaries based on points, contours and surfaces as reference in their programming
- Double clicking a contour allows you to edit the boundary – selecting whether it's open or closed
- Clicking arrows switches the direction of the machining side of the boundary
- Multiple boundaries can be selected in the same strategy – even at different Z levels for 2D operations



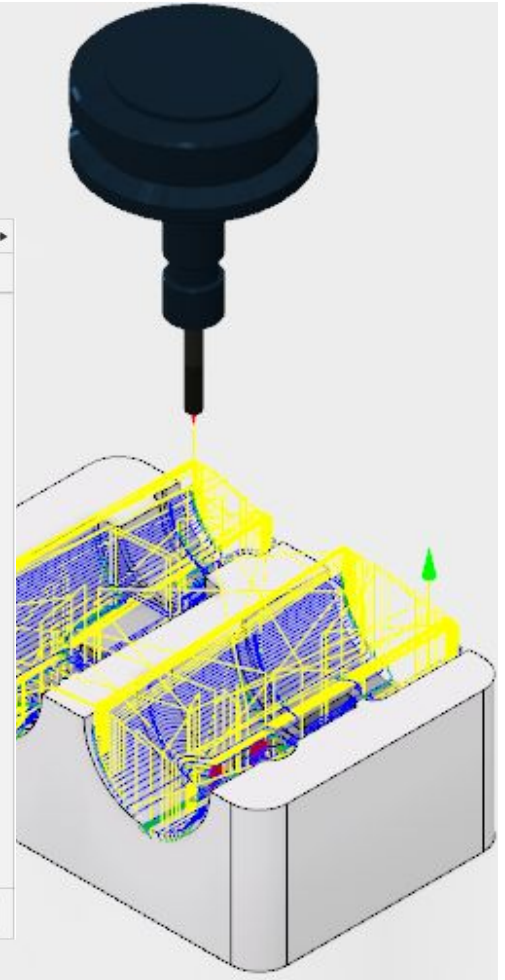
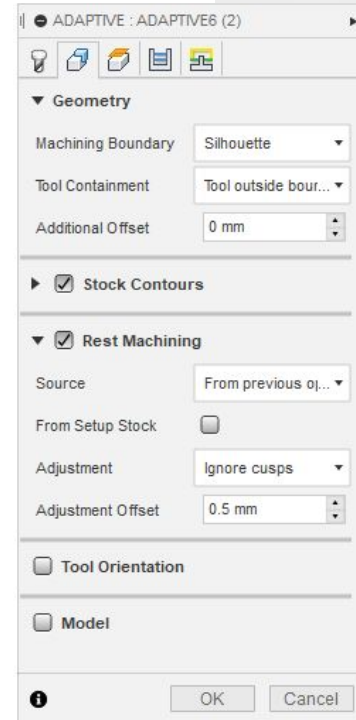
Heights

- Allocate the height of each tool move:
 - Clearance** - first height the tool rapids to on its way to the start of the tool path
 - Feed** – sets height that the tool rapids to before changing to the feed/plunge rate to enter the part
 - Retract** - height that the tool moves up to before the next cutting pass
 - Top** - sets maximum cutting height
 - Bottom** - sets minimum cutting height



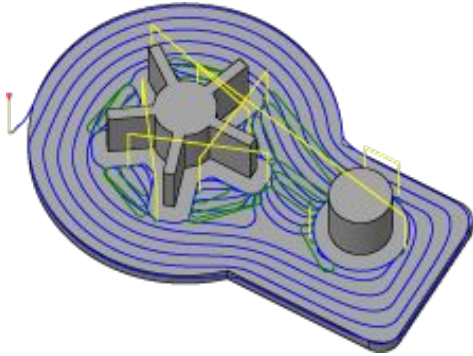
Rest Machining

- Rest machining is used when a larger tool access can't access certain areas because it's too big
- A smaller tool can then be specified to perform rest machining to clear the remaining rest materials
- The process instructs the tool to only cut the parts that couldn't be reached with X size of tool
- Opened with a tick box
 - Specify tool diameter and corner radius



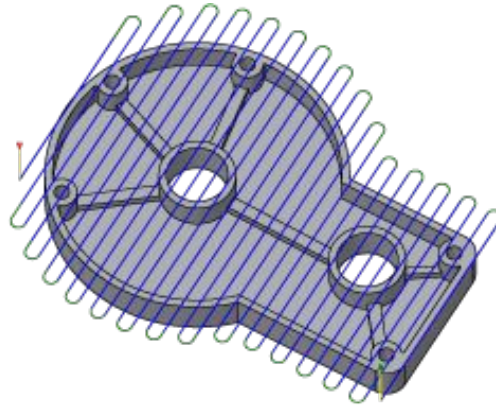
2D Strategies

2D Strategies



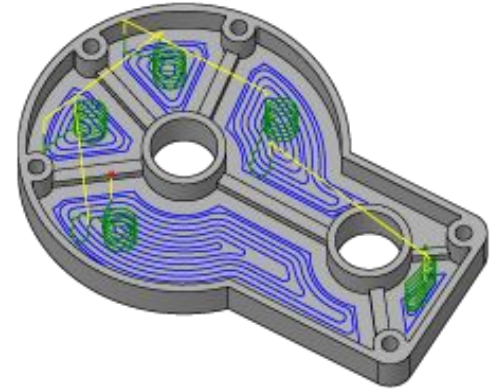
2D Adaptive clearance

- A roughing operation using a toolpath that "flows"
- You can clear a cavity, open pocket or the area around a boss.



Facing

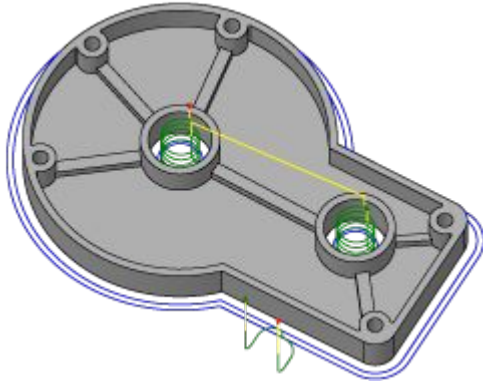
- Removes stock quickly from the top surface of the part
- Generally prepares the part for further machining - used for clearing flat areas



2D Pocket

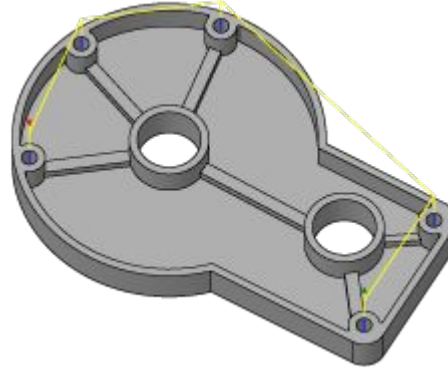
- A roughing and finishing operation - clears a cavity, open pocket or the area around a boss
- The toolpath resembles the shape of the boundary being machined

2D Strategies



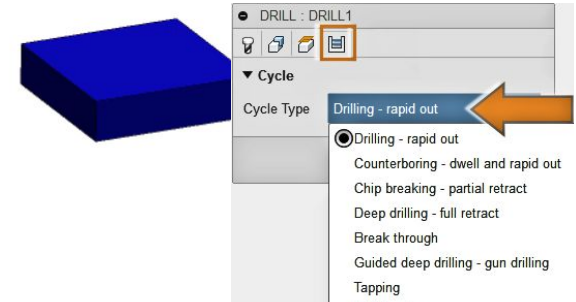
2D Contour

- Machines open/closed profiles with corner smoothing
- Typically a finishing operation – can use multiple cuts



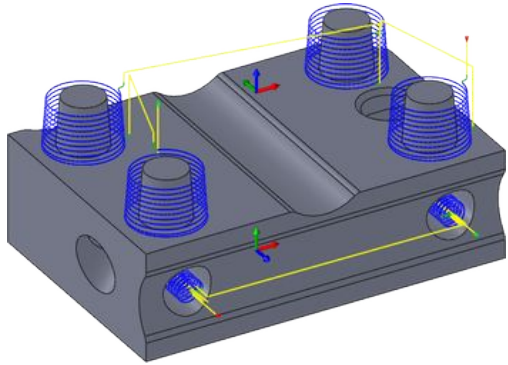
Drilling & Hole Making

- Drilling is a common machining task for creating holes in the work piece
- Cycle types can be selected within the operation itself

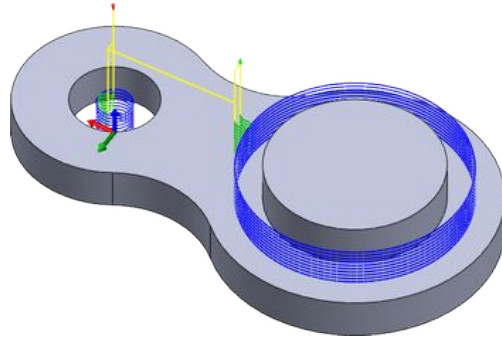


- All operations are optimized to minimize tool travel and cycle time
- All cycles are supported, including spot-drilling, deep drilling with chip break, etc.

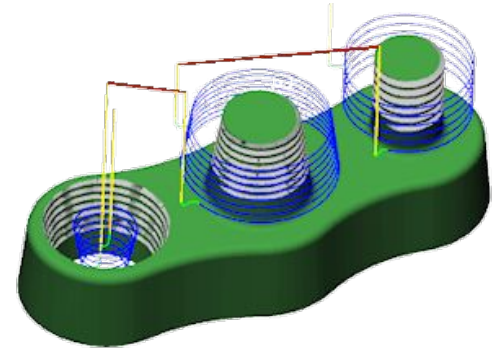
Circular Strategies



- The **Bore** operation lets you to bore mill cylindrical pockets and islands by selecting the cylindrical geometry directly

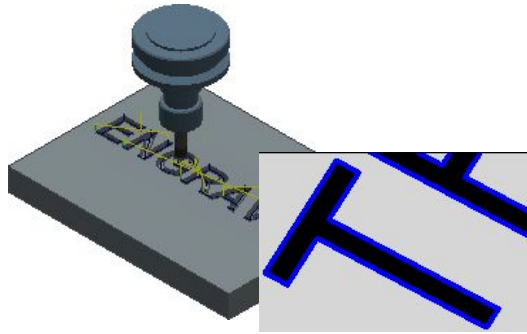


- The **Circular** strategy is used for milling cylindrical pockets and islands



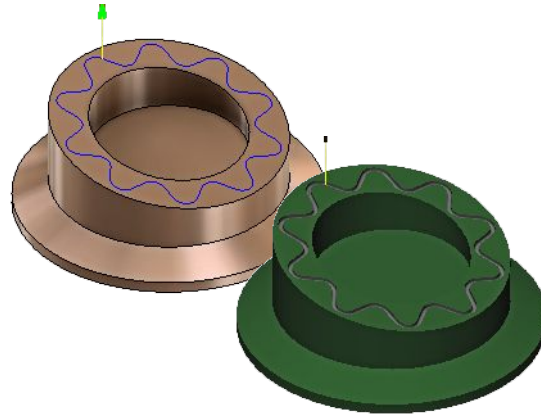
- The **Thread** operation is used for thread milling cylindrical pockets and islands

2D Strategies



Engrave

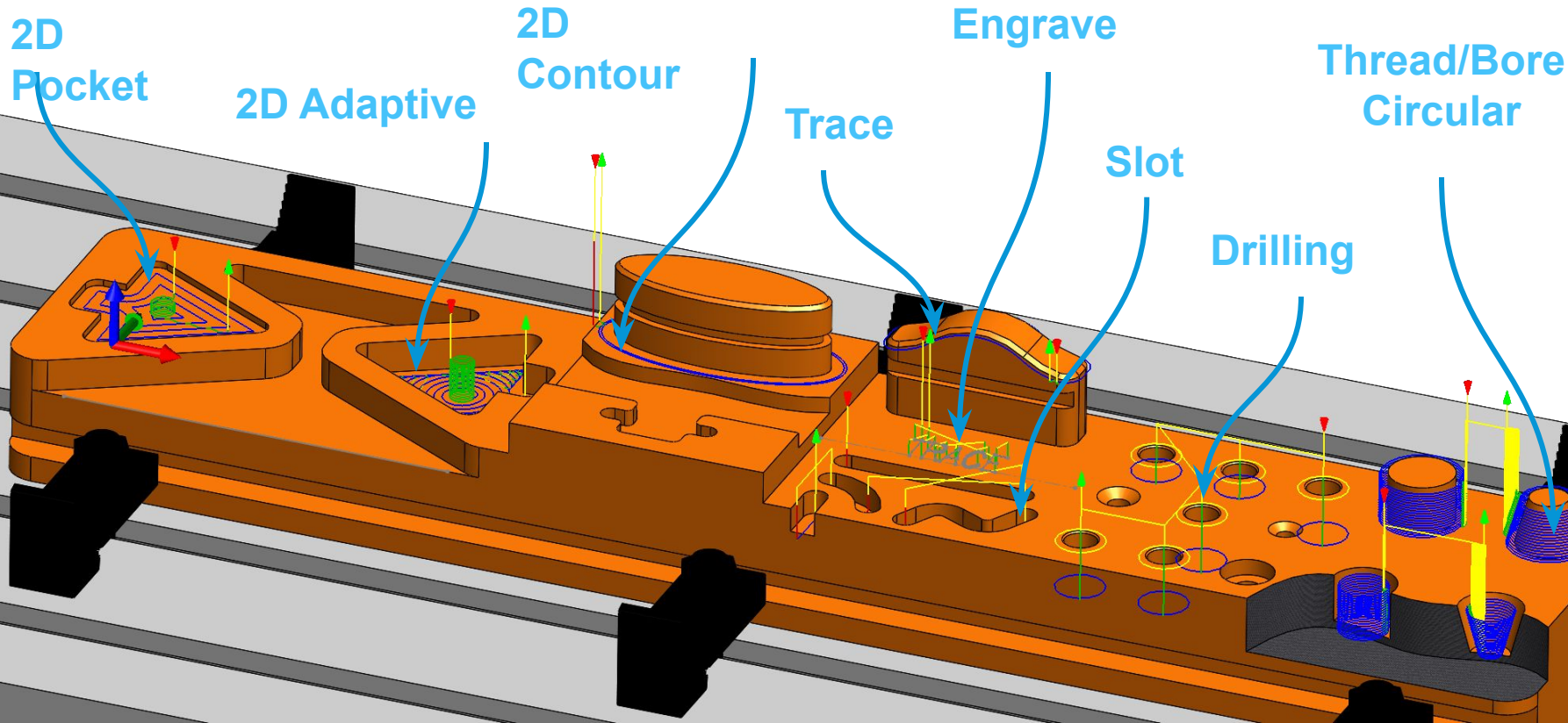
- Engrave machines along the contours with a chamfer tool
- Typically a finishing operation – can use multiple cuts



Trace

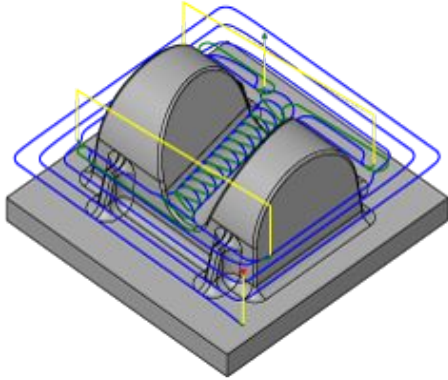
- Allows you to machine 2D/3D Edge contours defined in the sketch
- Single line engraving can be used for scroll work or text

All 2D Strategies



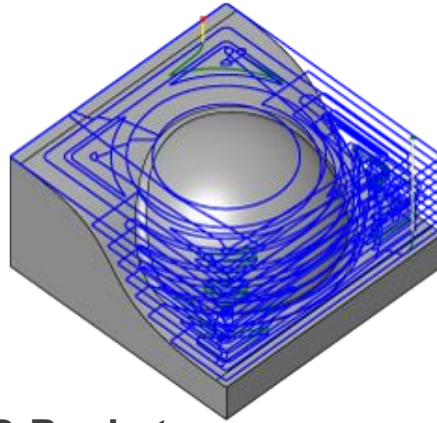
3D Strategies

3D Strategies



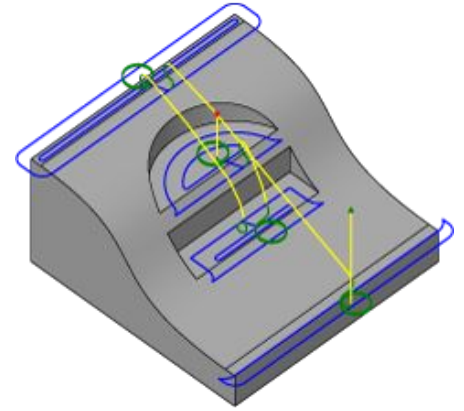
3D Adaptive Clearing

- Roughing strategy that avoids full-width cuts by progressively shaving material off the stock
- Cutting conditions remain constant with a stable load on the tool



3D Pocket

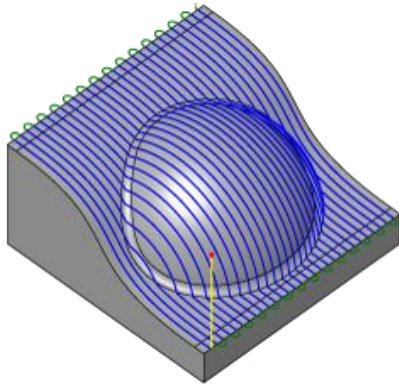
- Conventional roughing strategy for clearing large quantities of material
- The part is cleared layer by layer with smooth offset contours maintaining climb milling



Horizontal Clearing

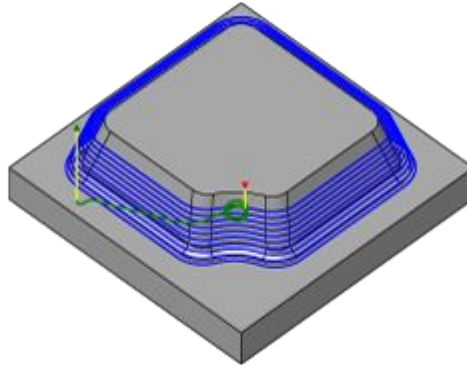
- Roughing/semi-finish/finishing operation
- Automatically detects all the flat areas of the part and clears them with an offsetting path and machines in stages

3D Strategies



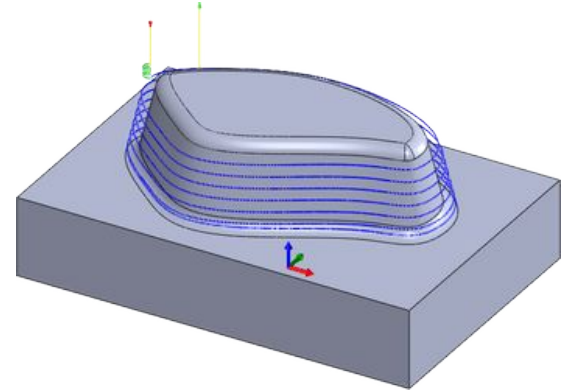
Parallel

- One of the most widely used finishing strategies – best suited for shallow areas and down milling
- Passes are parallel in the XY plane, follows the surface in the Z direction



3D Contour

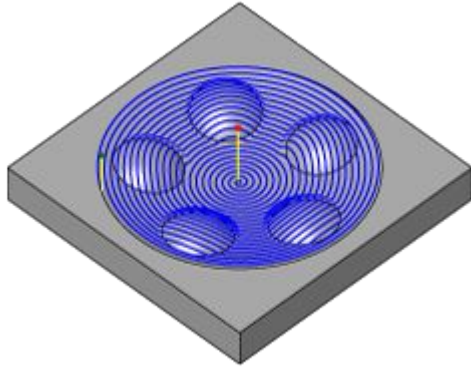
- Creates toolpaths with constant Z along contours of a part dependent on slope angle
- Suited for finishing steep walls - can be used for semi-finish / finishing



Ramp

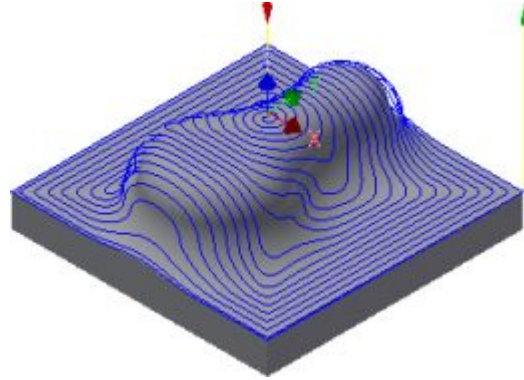
- Intended for steep areas
- Ramps down walls rather than machines with a constant Z
- Ensures that the tool is engaged at all times

3D Strategies



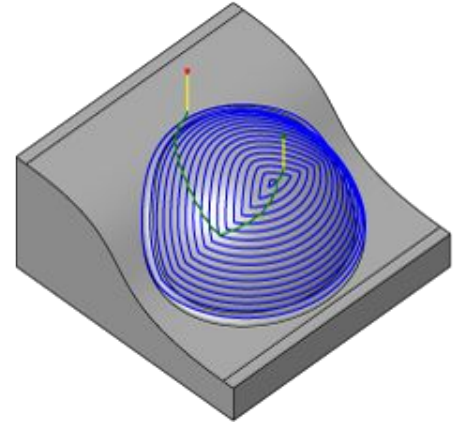
Spiral

- Creates a spiral toolpath from a given center point - machines within a given boundary
- Ideally suited for round shallow parts using tool contact angles up to 40°



Morphed Spiral

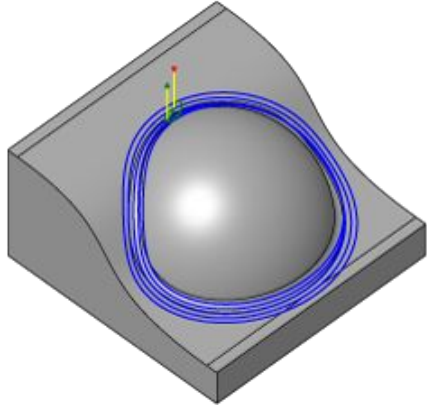
- Generates a spiral from a selected boundary for additional surfaces for which **Spiral** isn't suitable
- Useful when machining free-form/organic surfaces



Scallop/Constant Stepover

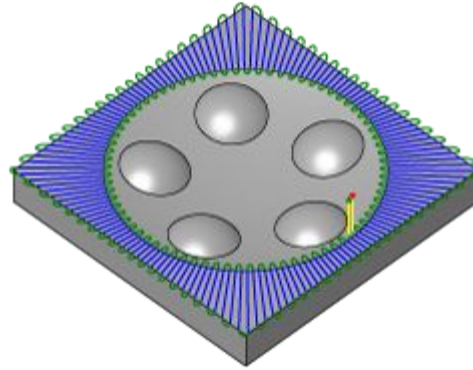
- Creates passes for sloping/vertical walls at a constant stepover from each other by offsetting inwardly
- Commonly used in rest machining

3D Strategies



Pencil

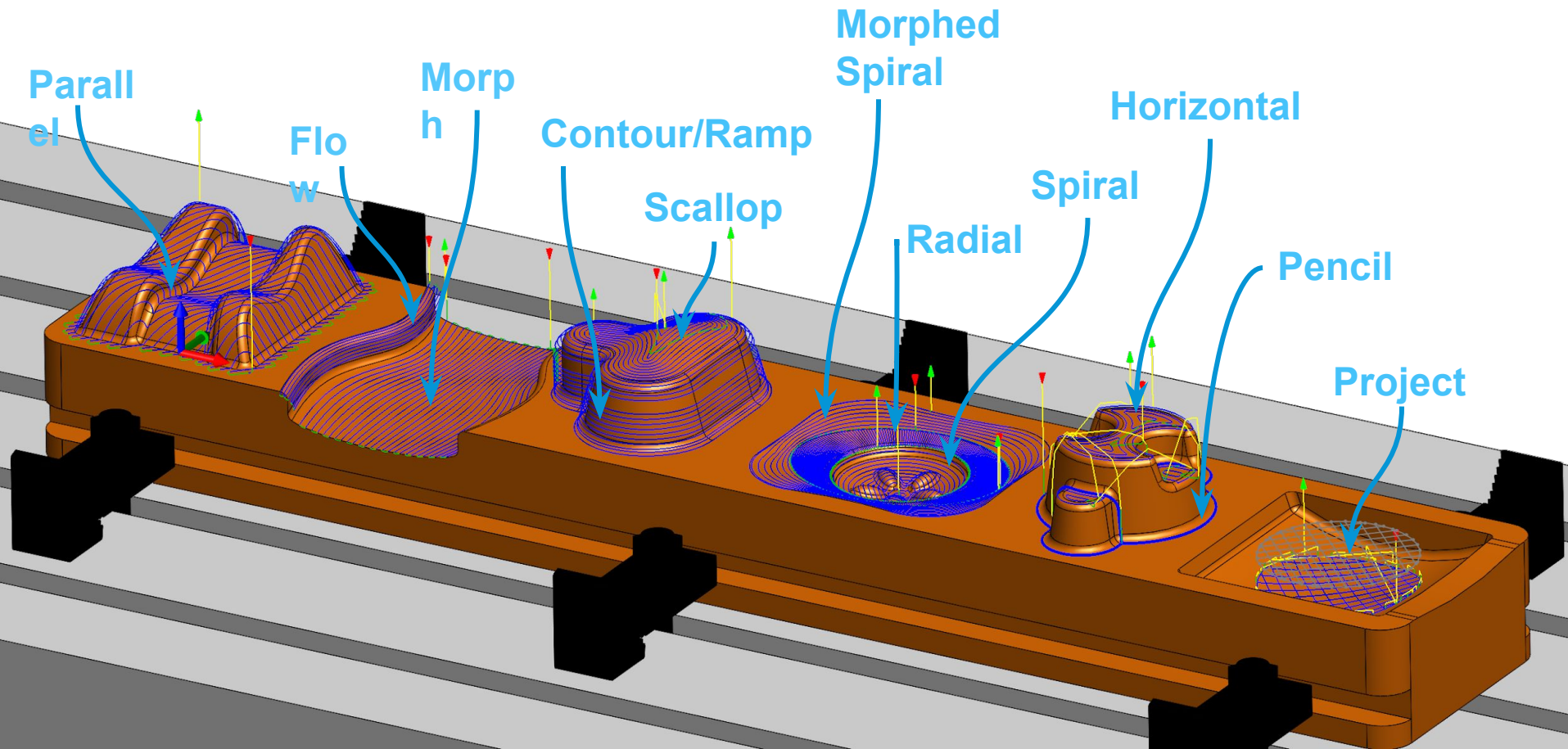
- Creates toolpaths along internal corners/fillets, removes material that other tools can't reach
- Suited for 'cleaning up' after other finishing strategies



Radial

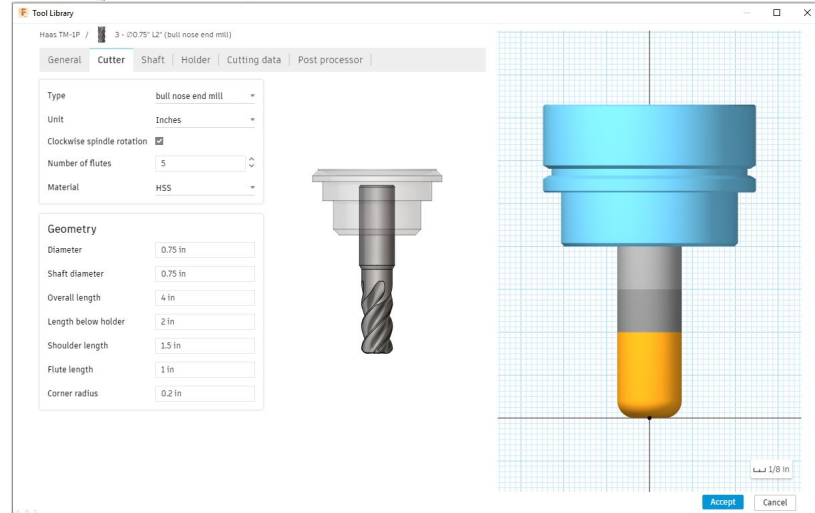
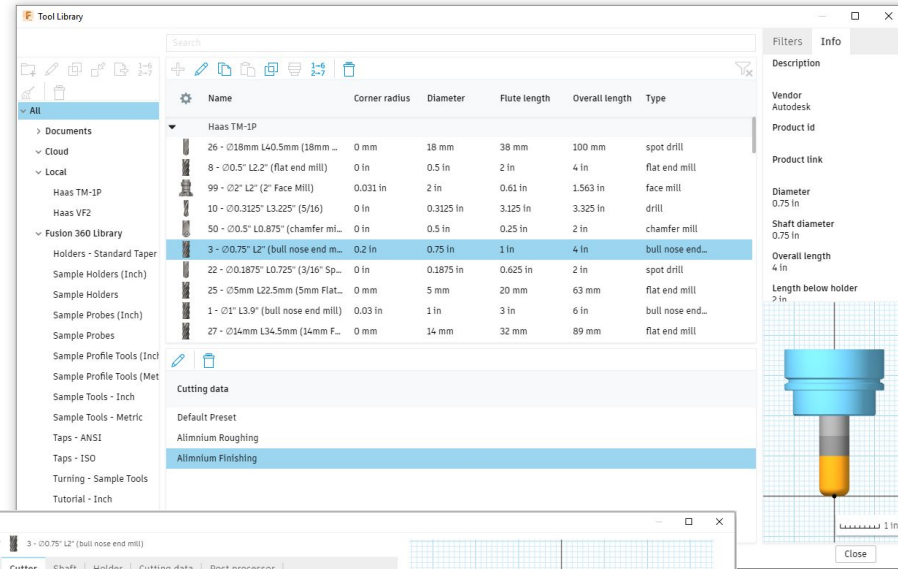
- Starts from center point, provides the ability to machine radial parts
- Gives the option to stop short of the center of the radial passes, where they become very dense

All 3D Strategies



Tool Library

- Tools are selected in the tool library
- Default tools can be chosen from the tool library or custom tools can be created
- Each type of tool can be selected and each element of a tool defined
- Default and custom tool holders are also available for selection
- Feeds and speeds associated with a tool and a tool's position in the magazine can also be defined
 - Presets for material cutting data can be applied for quick selection

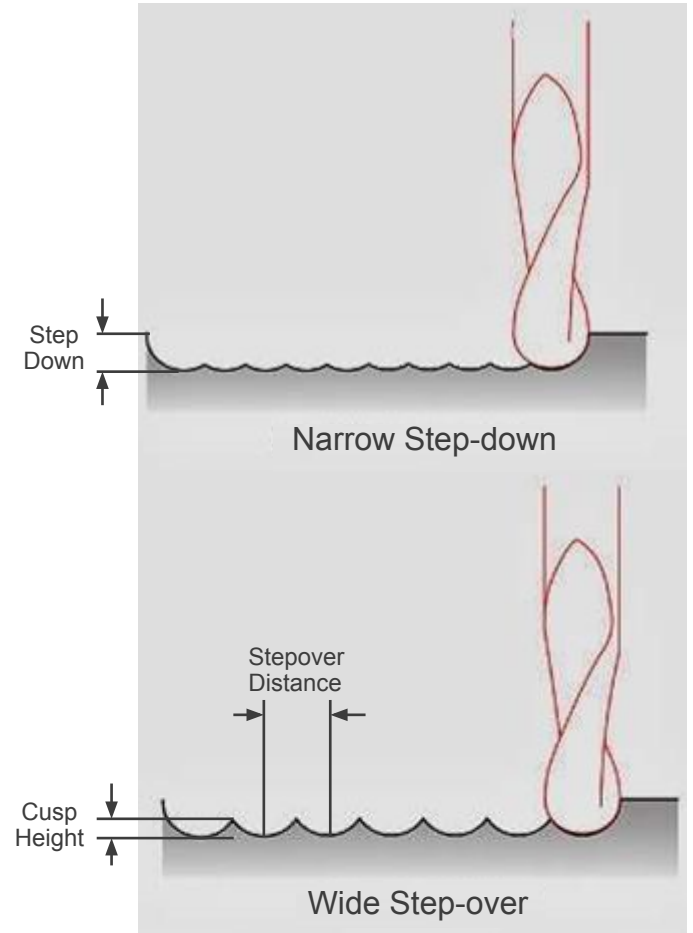


The background features a series of blue, curved, 3D-like shapes that resemble a stylized landscape or architectural elements. A prominent white diagonal shape, resembling a large 'Z' or a stylized '7', is overlaid on the scene. The overall color palette is light blue and white, with a clean, modern aesthetic.

Machining Parameters

Step-down & Step-over

- **Step-down** – the distance between each depth of cut in Z
 - Stepdown not always performed in constant Z
 - Too large a stepdown can increase tool wear
 - Should not surpass flute length
- **Step-over** - the distance between each machining pass at a constant Z height
 - Different strategies are available: Parallel, Spiral, etc
 - Can produce cusping if too large a step-over is chosen
 - Should not be more than tool diameter



Feeds and Speeds

Feed Rate - the speed the tool travels

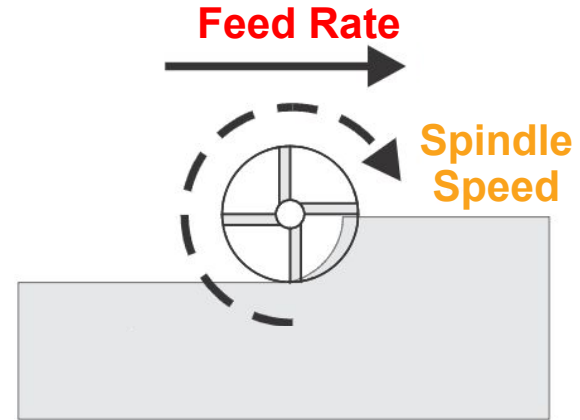
- Measured in distance per time unit – such as mm/min

Spindle Speed - the speed at which the tool turns

- Measured in revolutions per time unit such as rpm

- Tables of data specific to a tool can be used as a guide

- Getting the feeds and speeds right is crucial, if they are wrong serious damage can be caused



Milling Maching Cutting Speeds- *Terminology of Machine Tools (Krar, Oswald)* Table 61-1

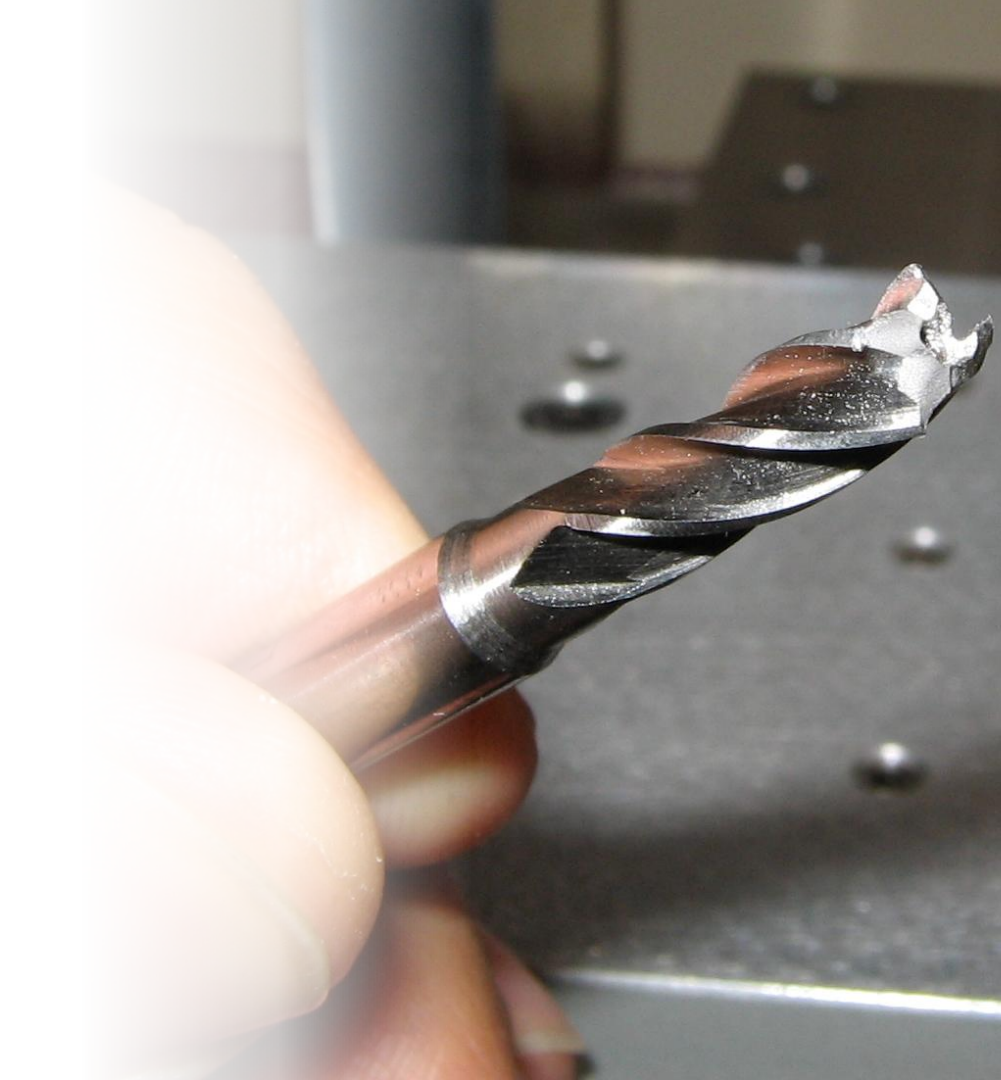
Material	High Speed Steel Cutter		Carbide Cutter	
	sfm	m/min	sfm	m/min
Machine Steel	70-100	21-30	150-250	45-75
Tool Steel	60-70	18-20	125-200	40-60
Cast Iron	50-80	15-25	125-200	40-60
Bronze	65-120	20-35	200-400	60-120
Aluminum	500-1000	150-300	1000-2000	300-600

Recommended Feed per Tooth High Speed Steel Cutters *Terminology of Machine Tools (Krar, Oswald)* Table 61-2

Material	Face Mills		Helical Mills		Slotting & Side Mills		End Mills		Form Relieved		Circular Saws	
	in.	mm	in.	mm	in.	mm	in.	mm	in.	mm	in.	mm
Aluminum	.022	.55	.018	.45	.013	.33	.011	.28	.007	.18	.005	.13
Brass & bronze (medium)	.014	.35	.011	.28	.008	.20	.007	.18	.004	.10	.003	.08
Cast iron (medium)	.013	.33	.010	.25	.007	.18	.007	.18	.004	.10	.003	.08
Machine steel	.012	.30	.010	.25	.007	.18	.006	.15	.004	.10	.003	.08
Tool steel (medium)	.010	.25	.008	.20	.006	.15	.005	.13	.003	.08	.003	.08
Stainless steel	.006	.15	.005	.13	.004	.10	.003	.08	.002	.05	.002	.05

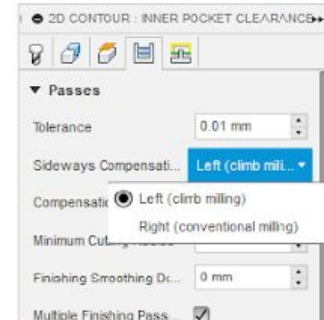
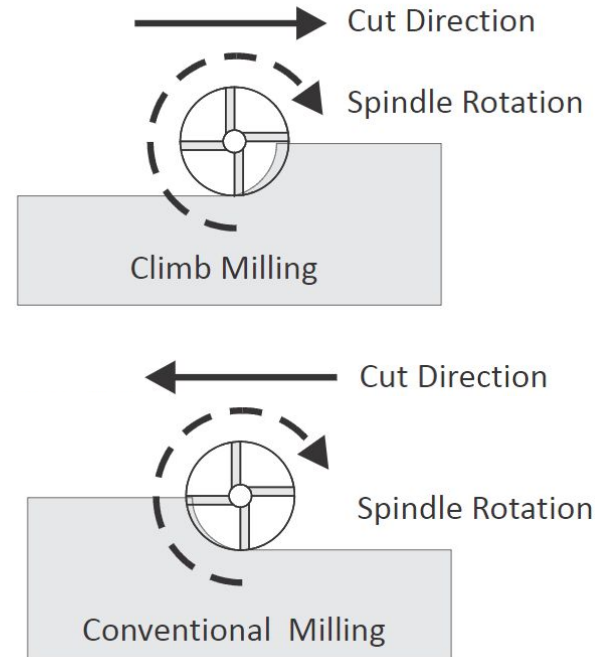
Feeds and Speeds

- For the best surface finish we run a moderate feed rate and high spindle speed
- For the best material removal rate we run a fast feed rate and a high spindle speed
- Too high a feed rate with a high chip load can lead to the tool breaking
- Too high a spindle speed and the tool overheats, which blunts the tool and can even melt the material



Climb vs Conventional Milling

- **Climb Milling** - flutes engage the material at maximum thickness and then decreases to zero
 - Unless recommended by manufacturers, always use climb milling
 - Climb milling produces less cutting pressure and heat, leaves a better surface finish
- **Conventional Milling** - causes the tool to scoop up the material, from zero to maximum
 - Often used on manual machines
 - Generates heat, increases wear, produces a poorer surface finish



CAM IN FUSION

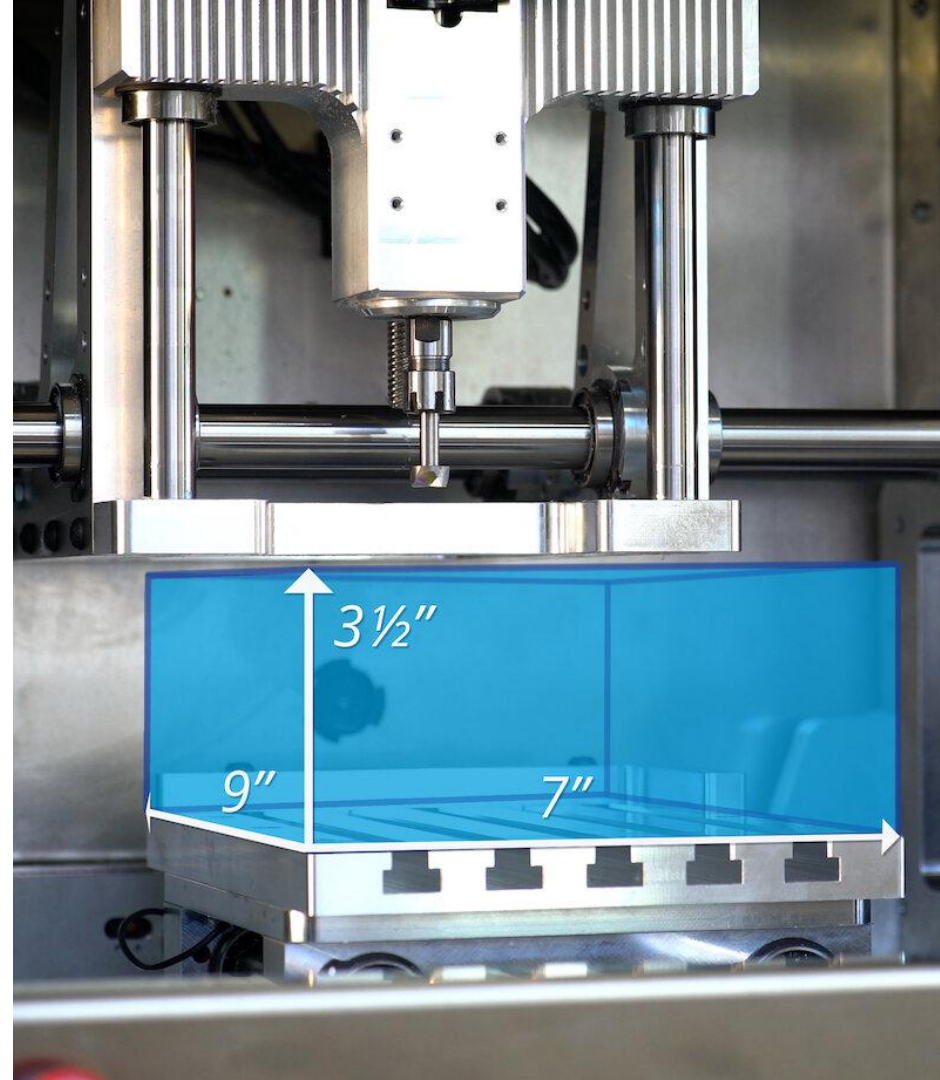
360 DEMO

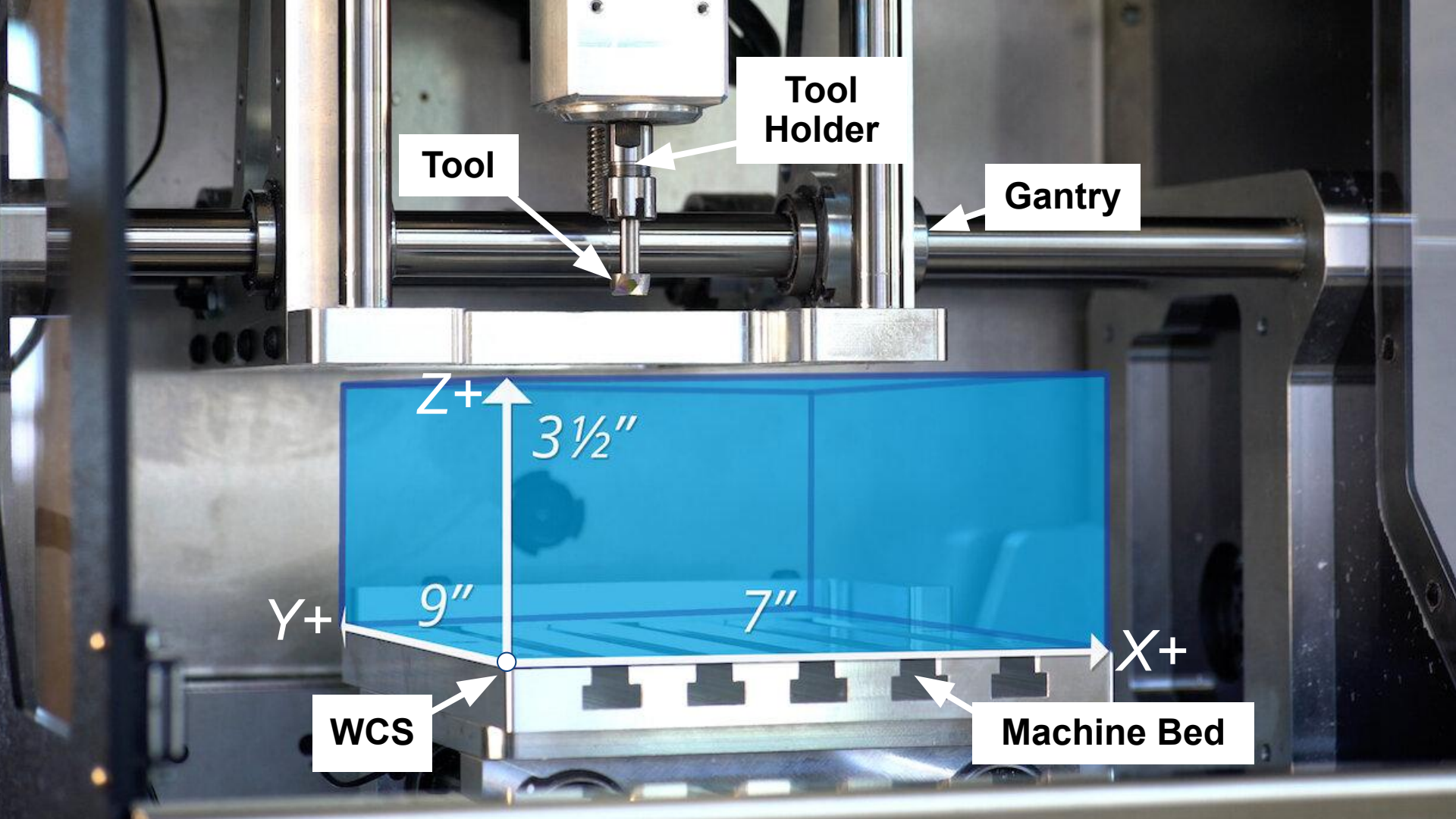
The background features a series of overlapping, semi-transparent blue and white geometric shapes, including curved planes and rectangular blocks, creating a sense of depth and modern design. A white, trapezoidal text box is positioned in the center-left of the image.

Bantam Tools Desktop CNC Milling Machine Configuration

Setting up the Machine

- The fixture needs to be clamped securely to the machine bed – above the maximum machine limit
- The stock axis/clamp must be parallel to the machine bed
- Each tool must be measured and the respective lengths added to the controller
- Each tool must be safely secured in the tool holder
- The user co-ordinate system must be configured to establish the location of the setup





Tool

Tool Holder

Gantry

WCS

Machine Bed

Z+

3 1/2"

Y+

9"

7"

X+

The part must fit within the machine build space

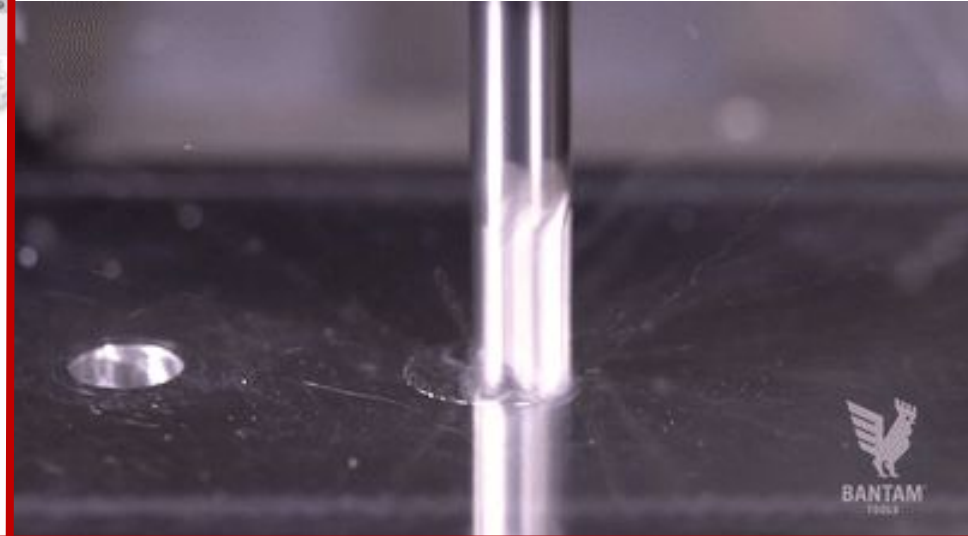
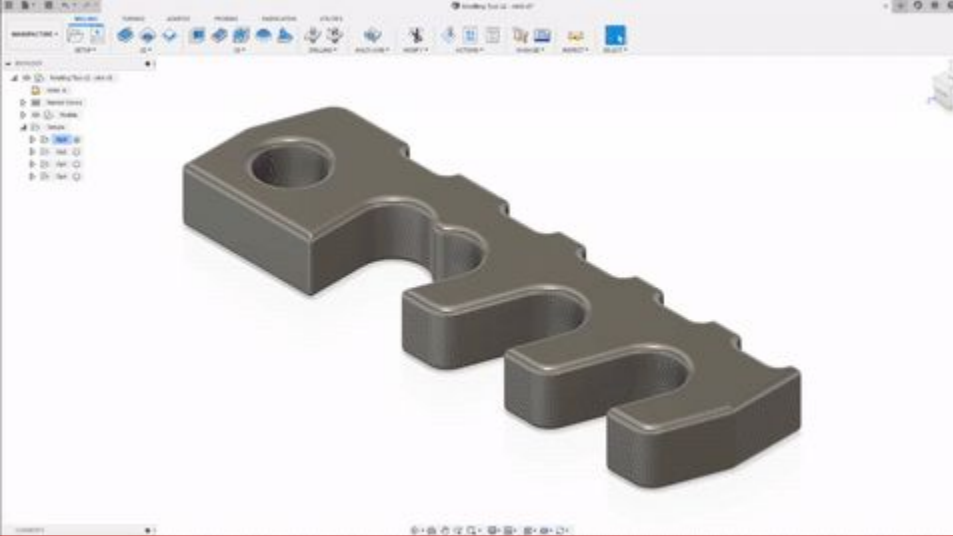
Tools must be safely secured in the tool holder

Each tool must be measured and the respective lengths added to the controller



The Stock axis must be parallel to the machine axes

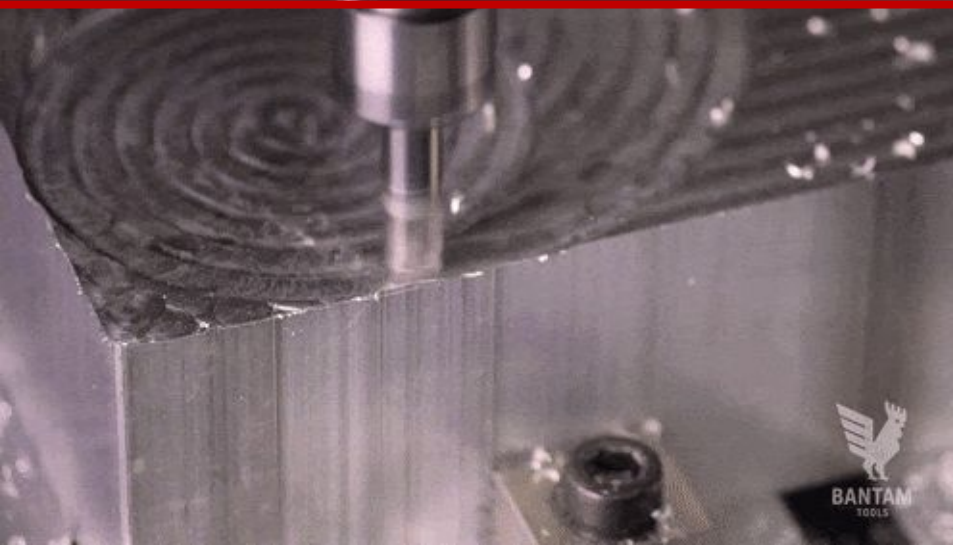
The user co-ordinate system must be configured to establish the location of the setup



Bantam Tools Desktop CNC Milling Machine

- Setup parts quickly with Automatic Stock Location
- Quickly produce prototypes / parts
- Machine Aluminium
- 28K RPM Spindle
- 180° Window visibility
- Drag in SVG files and use the built-in auto-CAM
 - Quickly adds engravings to your parts or mills simple 2D objects
- Use the built-in previewer to get real-time machining information







AUTODESK®

Make anything™