



Haas Desktop CNC Learning Machine
CAD/CAM/CNC
Training Guide



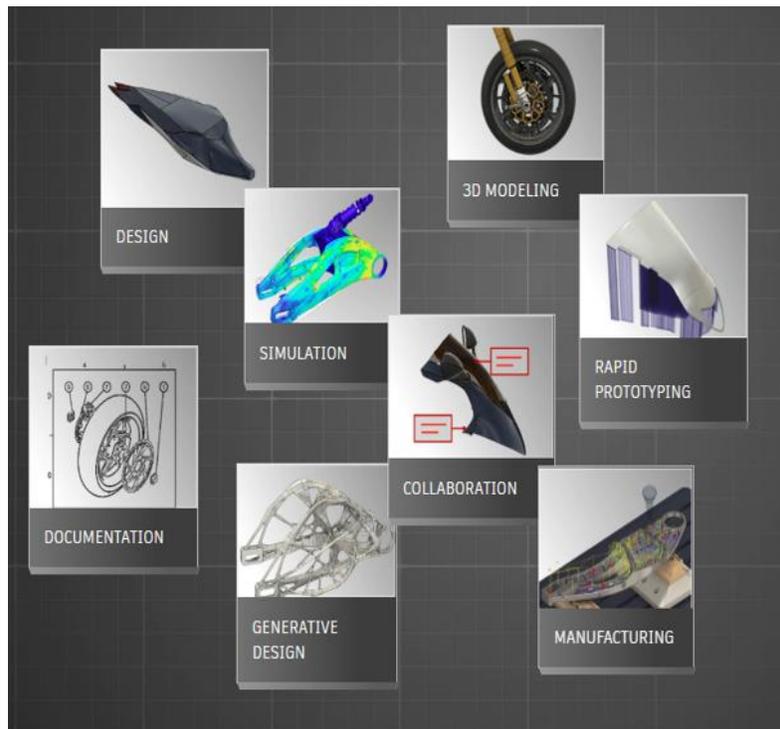
Introduction

This training guide is intended to assist educational users through the process of 3D CAD modeling techniques, manufacturing CAM Setups using Autodesk Fusion 360 software and suggested safe operation of the Haas CNC desktop milling machine.

Fusion 360 is the first 3D CAD, CAM, and CAE tool of its kind and has been chosen to be the best fit for this fidget spinner design to manufacturing project; as it is a collaborative product development tool that combines industrial design, mechanical engineering, and machine tool programming into one software solution.

Fusion 360 connects your entire product development process in a single cloud-based platform that works on both Mac and PC and is also available on Chromebooks or any device via browser for members of the Education Community.

Fusion 360 is a great easy to use tool for any type of educational platform and maker space and is freely available to all educational facility's, schools, colleges, and universities.

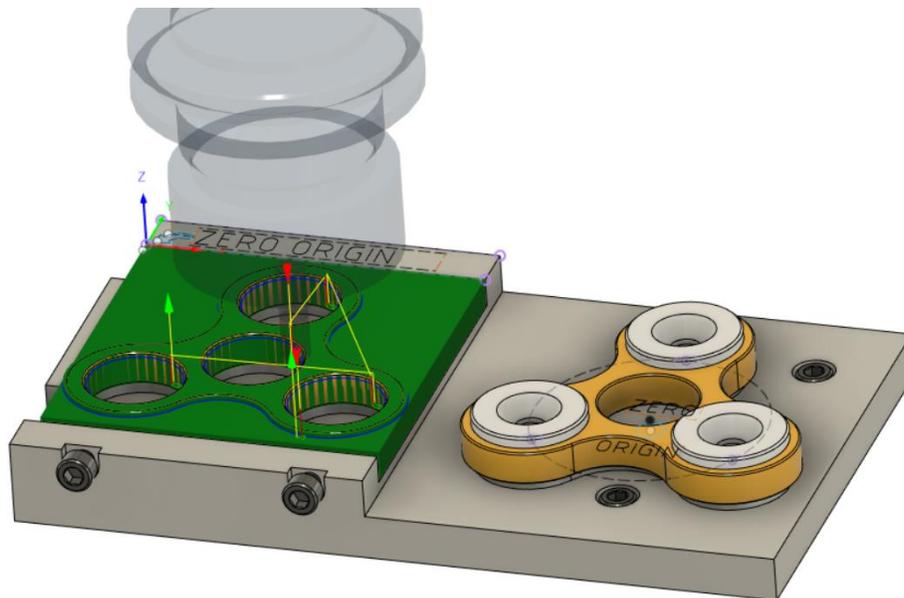


Fidget Spinner Project

This model design was chosen to illustrate Fusion 360 CAD 3D modeling techniques and CAM manufacturing workflow.



This unique project provides learners an opportunity to parametrically CAD model a design of a simple fidget spinner device with learning downstream manufacturing workflows that include CAD manufacturing assemblies, CAM setups, CNC toolpaths and post processing g-code. With understanding some fundamental steps in this process, users can apply their knowledge by safely setting up a Haas desktop CNC milling machine to manufacture their fidget spinner project.



ABOUT THE AUTHOR

Kevin Lee 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 skyscrapers and specialized aeroelastic instruments with the University of Western Ontario for sixteen years.

His continued education with Western Engineering focused on effective leadership leading 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 and founder of his own company LeeVerge Integration Inc. located in southwestern Ontario, Canada.

LeeVerge Integration is partnered with Autodesk as an Authorized Fusion 360 Reseller, Authorized Training Center and very active with engaging the educational community as a recognized Autodesk Learning Partner.

Kevin has proudly presented Fusion 360 classes at Autodesk University from 2017, 2018, 2019 and was chosen to present a manufacturing class in 2019 at the first ever Autodesk Fusion 360 Academy in Portland Oregon. Recently, Kevin was honored with presenting at the 2021 ASEE Virtual Conference showcasing proactive engineering design using Fusion 360 Generative Design technology.

Kevin's focus on integrated CAD/CAM solutions with educational collaborative learning provides guidance to the firm and its clients looking for innovative technologies that deliver competitive advantage with long term sustainability. Kevin extends on-demand Fusion 360 consulting support and training services to our clients allowing them to compete at a world class level utilizing Autodesk Fusion 360 software.



ABOUT THE CO-AUTHOR

Keith Rude is passionate about furthering manufacturing education and making advanced technology relatable. He has over 16 years of industry experience, beginning with studies in Aerospace technology at the University College of the Fraser Valley and then relocating to Calgary and undertaking a four-year machinist apprenticeship at Southern Alberta Institute of Technology.

As a Red Seal Machinist, Keith has worked with a range of manual and CNC machines, and as also mentored and trained others in the trade. He moved on to Thomas Skinner and Sons to work in the service department repairing CNC equipment, and then to his current role as an Applications Engineer.

Through his 9+ years with Thomas Skinner, Keith assists manufacturers with adapting new technology, as well as training end users on new equipment.

Keith is recognized as a subject matter expert and was honored to be involved in bringing this project to fruition.

His hope with this educational initiative is that it excites a new generation of students about the manufacturing industry by providing hands on experience in the classroom.



Applications Engineer



CONTENTS

Introduction.....	3
About the Author.....	4
About the Co-Author.....	5
Introduction to 3D Modeling.....	9
LESSON 1: Understanding Fusion 360 Workspaces.....	10
LESSON 2: Fusion 360 Data Management.....	15
Step 1: Understanding Fusion 360 Projects.....	15
Step 2: Creating a New Project.....	16
Step 3: Setting an Active Project.....	17
Step 4: Creating and Saving a New Design.....	18
Step 5: Fusion 360 file Versions.....	19
LESSON 3: Modeling from 2D sketches to a 3D CAD Model.....	20
Step 1: Creating your First Design.....	21
Step 2: Sketching Geometry.....	22
Step 3: Sketch Dimensions and Constraint Relationships.....	25
Step 4: Modeling 3D Features	30
Step 5: Inserting Assembly Referenced Components.....	32
Step 6: Adjusting Component Appearances.....	36
LESSON 4: CAD Modeling Manufacturing Assemblies.....	38
Step1: Discover Fusion 350 Sample Files.....	39
Step 2: Understanding Fusion 360 Assemblies.....	41
Step 3: Assigning Component Rigid Groups.....	42
Step4: Create a Manufacturing Assembly.....	43
Step 5: Capture Design Intent with Joint Relationships.....	46
LESSON 5: Define Manufacturing CAM Setups.....	52
Step 1: Create a New Manufacturing Setup.....	53

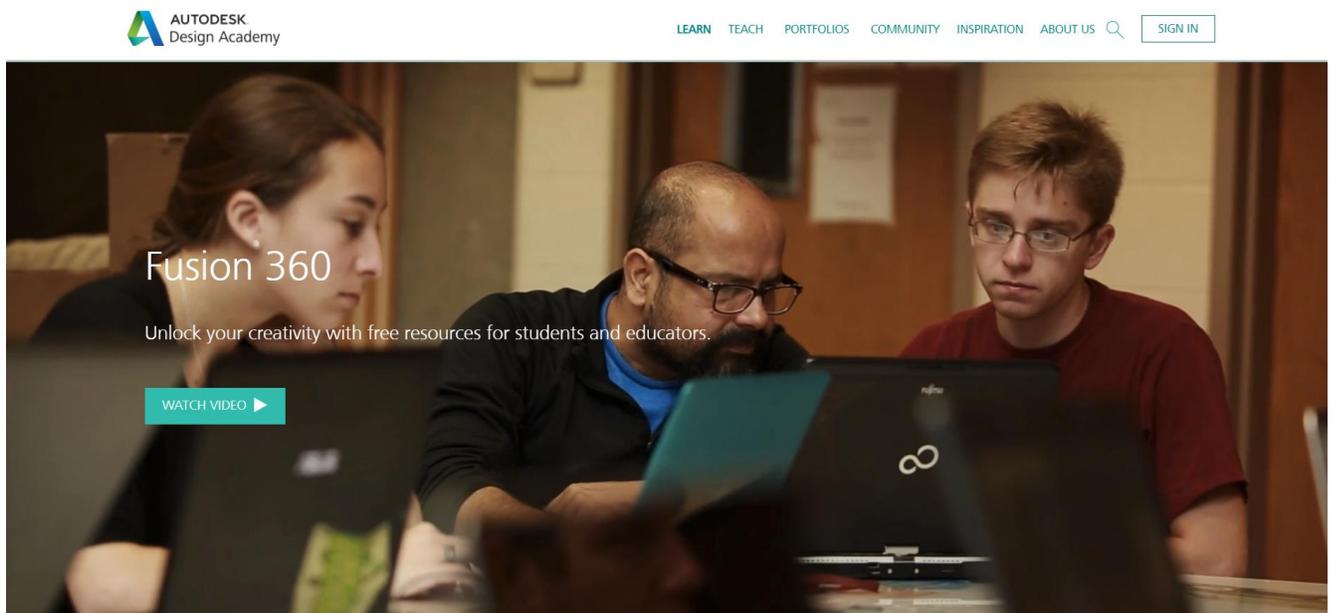
Step 2: Define Work Coordinate System (WCS).....	54
Step 3: Specify Program Zero Position.....	56
Step 4: Defining Stock Material Size.....	57
Step 5: Establish Post Processing Outputs.....	58
Step 6: Create a Secondary CAM Setup.....	59
LESSON 6: Manufacturing CAM Setup#1.....	70
Step 1: Create a Facing Operation.....	71
Step 2: Understanding High Speed Adaptive Roughing.....	74
Step 3: Finish Features with 2D Contour.....	77
Step 4: Duplicating Operations.....	79
Step 5: Deburring with Chamfer Toolpaths.....	81
Step 6: Verify Toolpaths with Stock Simulation.....	85
LESSON 7: Manufacturing CAM Setup#2.....	90
Step1: Copy and Paste Toolpaths.....	91
Step 2: Reapply Toolpath Geometry Selections.....	92
Step 3: Setting Chamfer Tip Offset Amount.....	95
Step 4: Verify Operations with Toolpath Simulation.....	97
Step 5: Create Setup Sheet Documentation.....	98
LESSON 8: Haas Desktop Machine Setup.....	100
Step 1: Secure Work Holding Fixture.....	102
LESSON 9: Qualify Tool Lengths & Work Offsets.....	103
Step 1: Understanding Machine Offsets.....	104
Step 2: Tool Length Offsets.....	105
Step 3: Qualify G54 Fixture Work Offsets X & Y axis.....	108
Step 4: Qualify G54 Fixture Work Offset Z axis.....	111
Step 5: Calculate G55 Fixture Work Offsets.....	113
LESSON 10: Post Processing G-Code.....	116

Step 1: Understanding NC Program.....	117
Step 2: Selecting a Post Processor.....	118
Step 3: Adjusting NC Program Settings.....	119
Step 4: Loading NC Programs to Machine Controller.....	123
Additional Learning Resources:	124
Step 1: Self-paced Learning Pathways for Fusion 360.....	124
Step 2: Autodesk Learning Partner Training Workshops.....	125
Step 3: Autodesk Fusion 360 Certifications.....	126
Step 4: Haas Operator Certification Program.....	127

Introduction to 3D Modeling

Fusion 360 enables you as a designer, engineer, entrepreneur, creator if you will; work seamlessly in one cloud enabled platform. The main focus of this is to help ease the learning transition from 2D design to 3D modeling. 3D modeling using CAD is an outstanding additional skill set for any product designer, engineer, or machinist and being able to use Fusion 360 CAD/CAM will enable you to take an excellent idea from a concept, through to a working 3D CAD model for manufacture, testing and production.

Fusion 360 and Autodesk Design Academy provide extensive learning resources which make it incredibly easy to use CAD and learn how to 3D model. Furthermore, Fusion 360 is free to students and educators, works on Apple and PC operating systems, on multiple devices and platforms.



The image shows a screenshot of the Autodesk Design Academy website. At the top left is the Autodesk Design Academy logo. To the right is a navigation menu with links for LEARN, TEACH, PORTFOLIOS, COMMUNITY, INSPIRATION, ABOUT US, and a search icon. A SIGN IN button is located on the far right. The main content area features a large photograph of three people (two men and one woman) looking at a laptop. Overlaid on the photo is the text 'Fusion 360' in a large white font, followed by the subtitle 'Unlock your creativity with free resources for students and educators.' Below this is a teal button with the text 'WATCH VIDEO' and a play icon.

www.academy.autodesk.com

LESSON 1

Selecting the right Fusion 360 Workspace

After completing this lesson, you will be able to:

- Understand the different workspaces in Fusion 360
- Select the appropriate Fusion 360 Work space

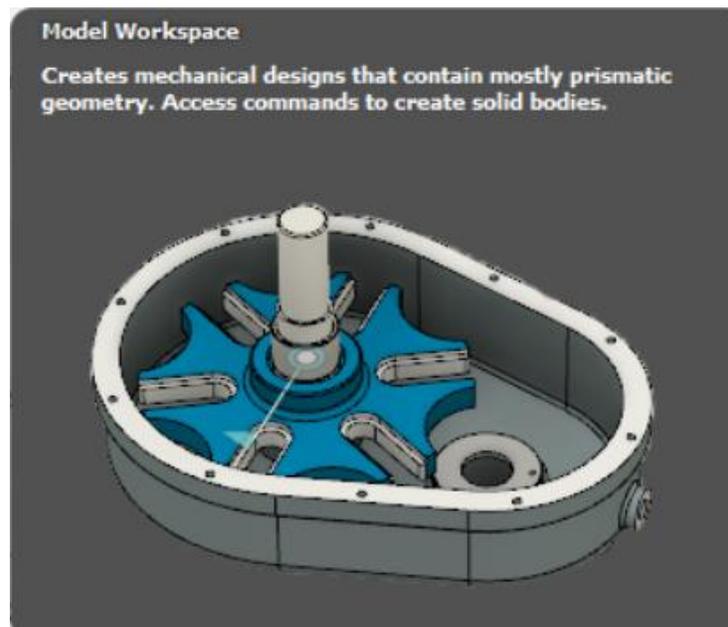
Workspaces in Fusion 360

Fusion 360 capabilities are grouped into purpose-driven workspaces. The tools in each workspace are organized in the toolbar according to particular design objectives. Certain menus and commands are available across multiple workspaces.

The following workspaces are available in Fusion 360.

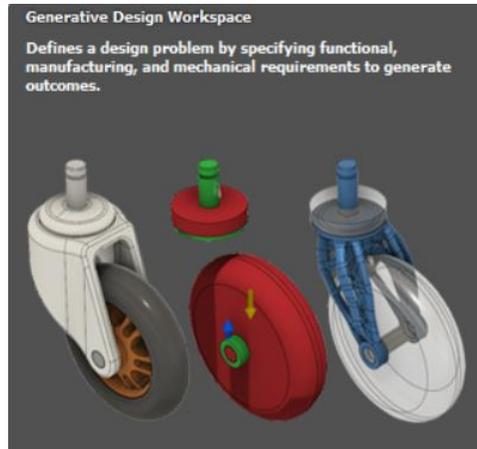
Design Workspace

The **Model workspace** lets you create and edit solid 3D geometry. This workspace is most like a traditional 3D CAD environment, where you can create history-based features (i.e. extrude, revolve, loft, sweep) that update with design changes and is a traditional CAD environment that supports parametric and direct modeling.



Generative Design Workspace

The **Generative Design** workspace lets you simultaneously generate multiple CAD-ready design solutions based on manufacturing and performance requirements. It contains tools that let you set up design studies with defined goals, constraints, materials, and manufacturing options. Generate and explore multiple process and performance-aware design options in the cloud. Evaluate each design option based on trade-off's that are most critical to your needs. Then export your optimal design using cloud credits, and import the CAD-ready geometry into Fusion 360.



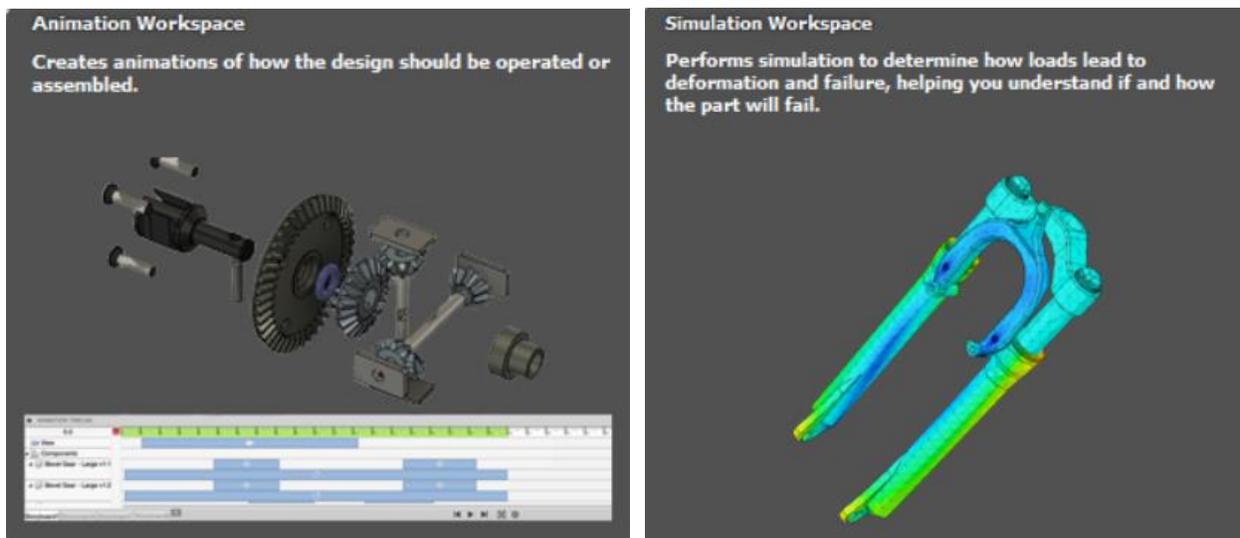
Render Workspace

The **Render** workspace lets you generate photo-realistic images of your design. Adjust lighting and materials, add decals, and leverage local or cloud computing to communicate your design realistically.



Animation Workspace

The **Animation** workspace helps you communicate your design with 3D exploded views and animations to show design assembly. Share videos with collaborators and clients to help them understand and evaluate your design. The Animation workspace lets you create automatic or manual exploded views, as well as to have direct control over unique animations of parts and assemblies. Animations can be used to evaluate and communicate designs and functionality of your designs including assembly or repair operations.

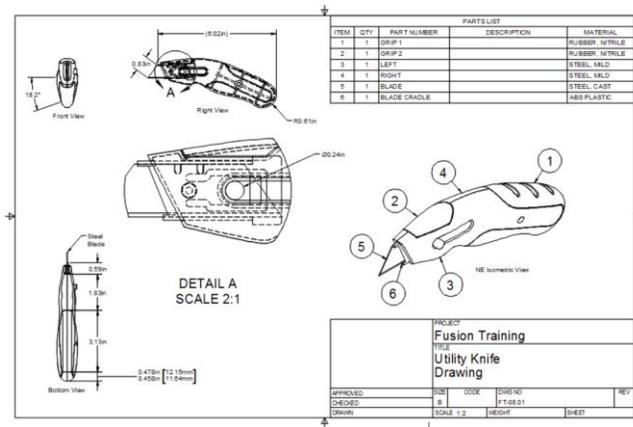
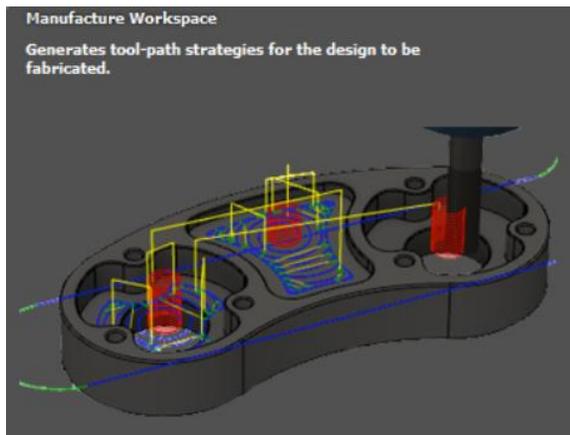


Simulation Workspace

The **Simulation** workspace lets you set up studies to test your design using finite element analysis (FEA). Simulate how your design performs under various loads and conditions. Analyze the results to understand the physical limitations of your design. Explore design alternatives and make informed decisions about design changes. The Simulation workspace enables you to perform stress (Static Stress, Nonlinear Static Stress, and Event Simulation), Modal Frequencies, Structural Buckling, Thermal, Thermal Stress and Shape Optimization analyses.

Manufacture Workspace

The **Manufacture** workspace lets you create toolpaths to machine your components using Computer-Aided Machining (CAM) or use the 3D printing workflow to create a prototype. CAM stands for **Computer Aided Manufacturing**. CAM is the step where you convert the part model into a language that can be used for the manufacturing process (usually G code). This process might be for milling, turning or laser cutting and might be done in wood, plastic or metal, depending on the application. The CAM environment in Fusion 360 is designed from the ground up to work inside Fusion 360.



Design Workspace

Drawing Workspace

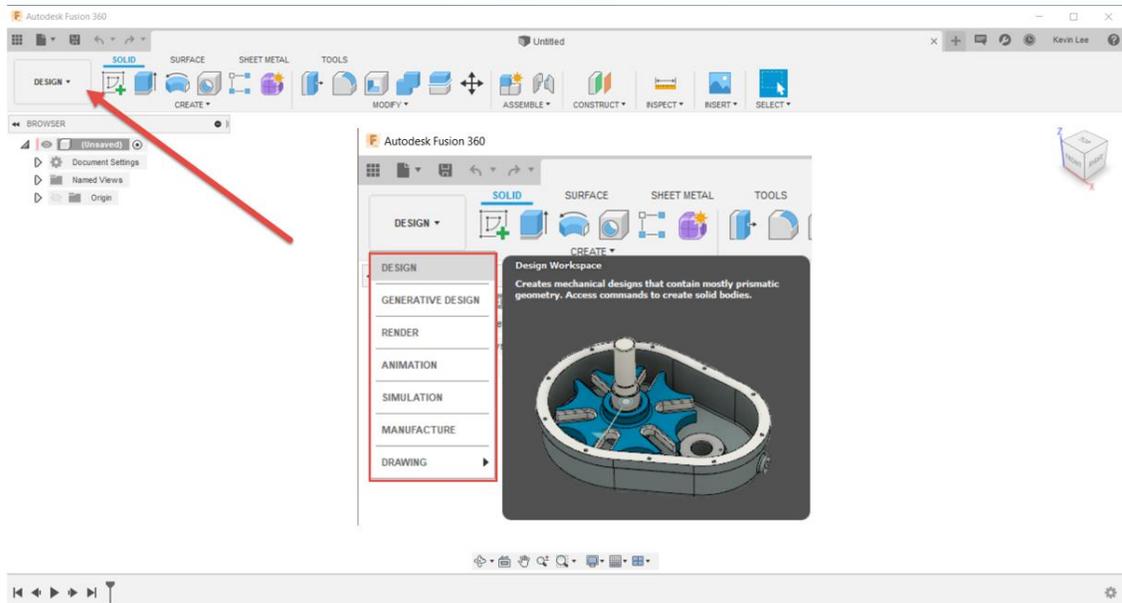
The **Drawing** workspace lets you document manufacturing specifications using integrated, associative drawings and animations for parts and assemblies. When you create 2D drawings from your Fusion 360 designs, it enables you to generate PDF and DWG documentation of your Fusion 360 model. When you create a drawing, it is generated as a derived document of a Fusion 360 model, and it shows up in the Data Panel as a unique item in the active project.

When you create a drawing from the Fusion 360 modeling environment, the system generates a 2D projection of the components you select. The drawing view thus generated is referred to as a base view. Once you place the base view in the drawing, you can generate orthogonal and isometric projected views from it containing dimensional references and part specifications.

Selecting the Design Modeling Workspace

Upon opening Fusion 360 by default, you will begin with a blank untitled document and will automatically start you inside of the Model Workspace to begin designing your first project.

You can easily move into other Fusion 360 workspaces by selecting the drop down arrow in the top left-hand corner of the user interface.

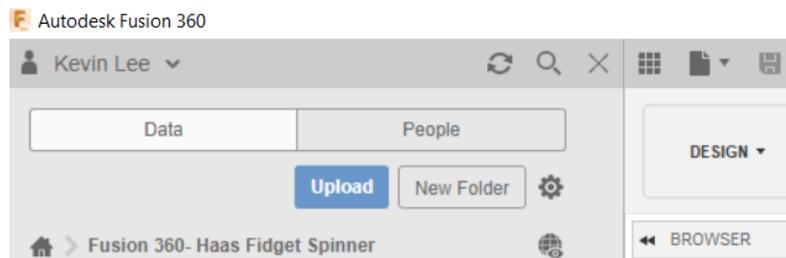


LESSON 2

Fusion 360 Data Management

After completing this lesson, you will be able to:

- Understand what is a Fusion 360 Project
- Create a new design project
- Set the Active Project
- Create and save a new design
- Understand Fusion 360 file versions



Step1: Understanding Fusion 360 Projects

Fusion 360 brings design teams together for collaborative product development. You can manage the design process and collaborate with your team, clients, and partners by sharing and reviewing designs, managing versions, and reusing designs.

Product development typically requires multiple people to take a product to market.

In Fusion 360, a project is a shared workspace for teams of people working together on a project to store, organize, and manage all related design data.

Projects let you control who has access to specific design data. Each project contains its own project members who are assigned project roles. Project members can share and access design data, discuss challenges and successes, and stay current with project activities.

Step2: Creating a New Project

In Fusion 360, you create projects to contain and organize your design data. You can invite other people to a project to collaborate on a design, discuss challenges and successes, and stay current with project activities. Each project has its own data and people.

There are two ways to create a project:

- Use the **Data Panel**
- Save a new design
- *Create a new Fusion 360 project to familiarize yourself with projects organization.*

Create a new project using the Data Panel

1. Click the **Show Data Panel** icon  to open the **Data Panel**, which displays your projects and designs.
2. Click the **New Project** button.
3. Specify a name for the project. Name your first project, **Fusion 360-Haas Fidget Spinner**.
4. Double-click the project to open it.

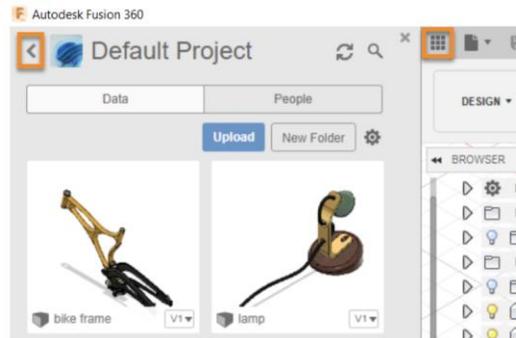
Tip: To return to the list of projects, click the **Leave Data Details** icon  at the top left of the **Data Panel**.

Note, you can also create a new project when you save a new design

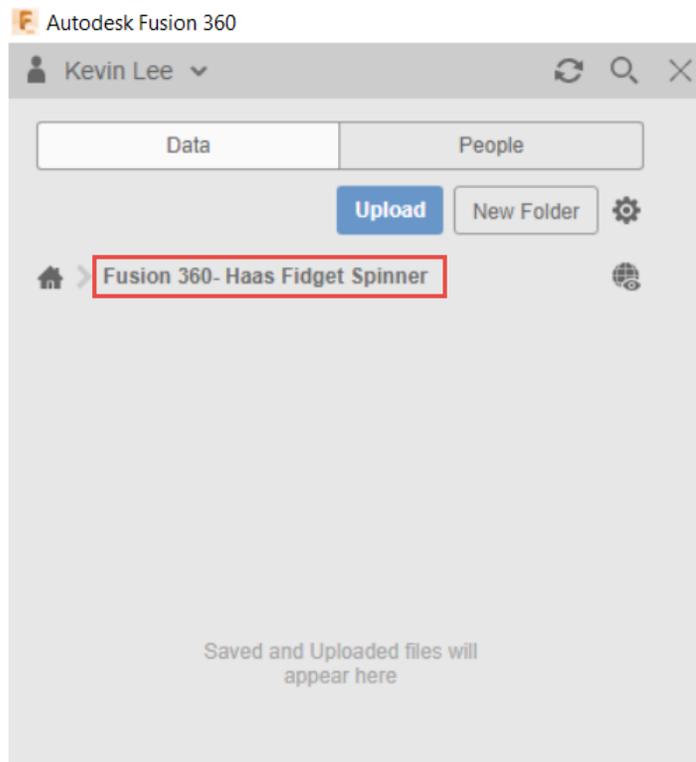
1. In the **Application Bar**, select **File > Save As**.
2. Specify a name for the design in the **Name** field.
3. Next to the **Location** field, click the arrow to expand the options.
4. Click the **New Project** button to create a new project.
5. Specify a name for the project.
6. Select the new project in the list of projects.
7. Click **Save**.

Step3: Setting the Active Project

1. Click the **Data Panel** icon  in the upper left to display the **Data Panel**.
2. Click the back button next to the active project in the **Data Panel**.

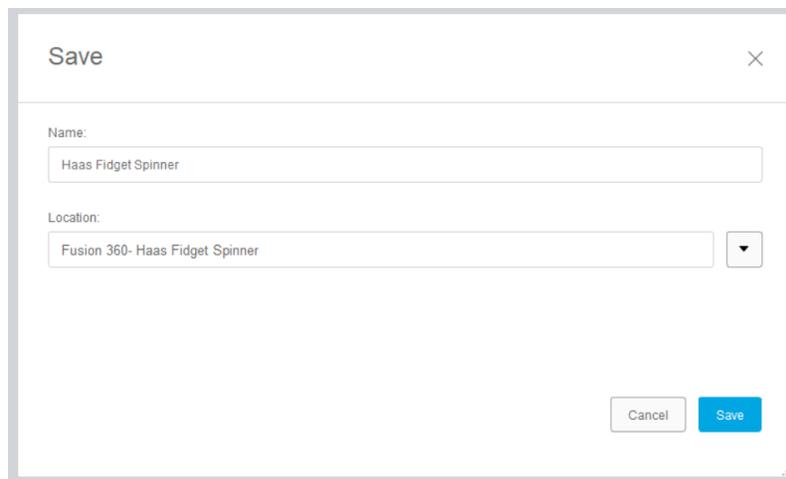


3. Double-click your newly created project **Fusion 360 -Haas Fidget Spinner** to make that project active.

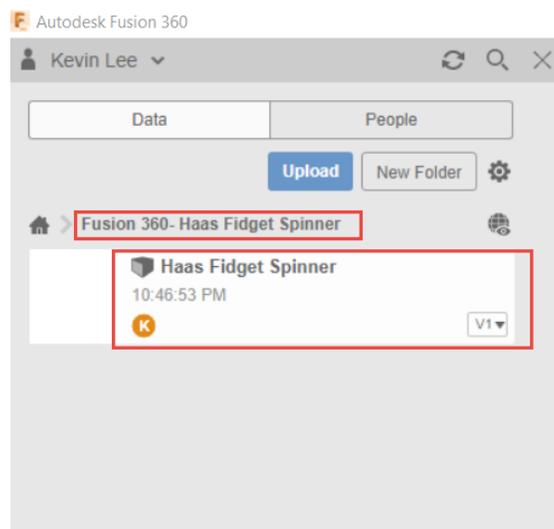


Step4: Creating and saving a new design

1. Click the **File** icon  > **New Design**.
2. Click the **Save** icon  to save the design.
3. In the **Name** field, enter **Haas Fidget Spinner**.
4. Set **Location** to store your new file. Ensure that it sets the location to save your new design into your new created **Fusion 360 -Haas Fidget Spinner** project.
5. Click **Save**.



6. Once your new design saves into your project your data panel should look like the following.



Step5: Fusion 360 file Versions

In Fusion 360 all versions of your document are stored, and you can choose which version you want to use. You also have an option to open a desired read-only version, or promote a version directly from the Data Panel.

After we complete a few more lessons we will be saving our work. Every time you save your file it creates a version of your file. Follow these steps below later on to better understand file version control in Fusion 360.

1. In the **Data Panel**, click the version icon  for the design.
2. If all versions are not listed, click **Show all versions**.
3. Hover over the desired version, and click **Open Version**  .
 - Note: This version is read-only. The comments pane is disabled for read-only documents.
4. Edit the file as needed.
5. To save the file, click **File > Save as**.
6. In the **Save** dialog box, specify the file name and the location.
7. Optional: In the **Add Version Description**, enter a description.
8. Click **OK**, and a new version of your design is created.

Promote a previous version of a design

1. In the Data Panel, click the version icon  for the design.
2. If all versions are not listed, click **Show all versions**.
3. Hover over the desired version, and click **Promote**  .

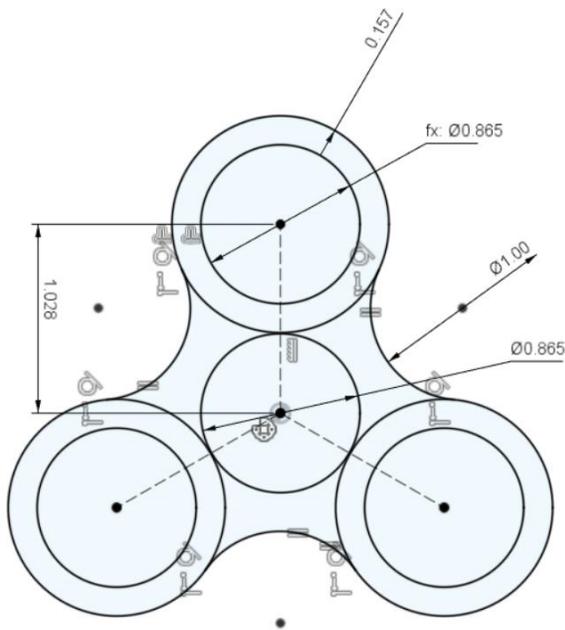
Note: Promote is not available for versions that are currently open.

LESSON 3

Modeling from 2D sketches to a 3D CAD Model

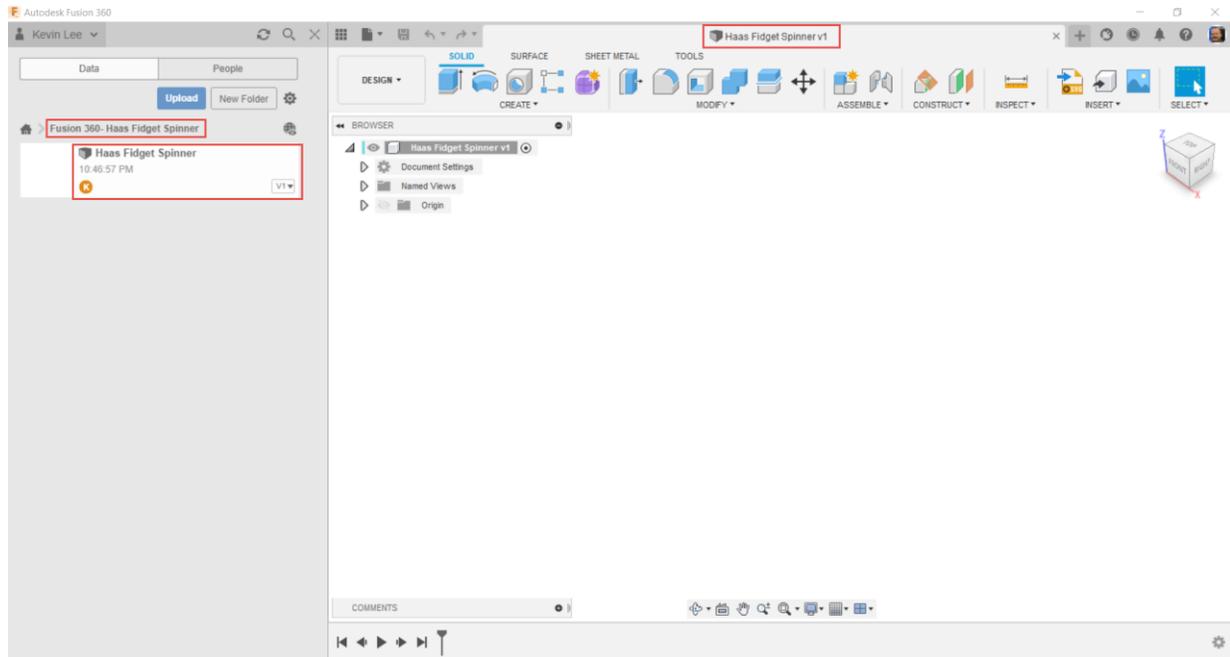
After completing this lesson, you will be able to:

- Create your first design
- Sketch geometry
- Define sketches with dimensions and constraint relationships
- Create 3d modeling features
- Insert Assembly Referenced components
- Adjust Component Appearances

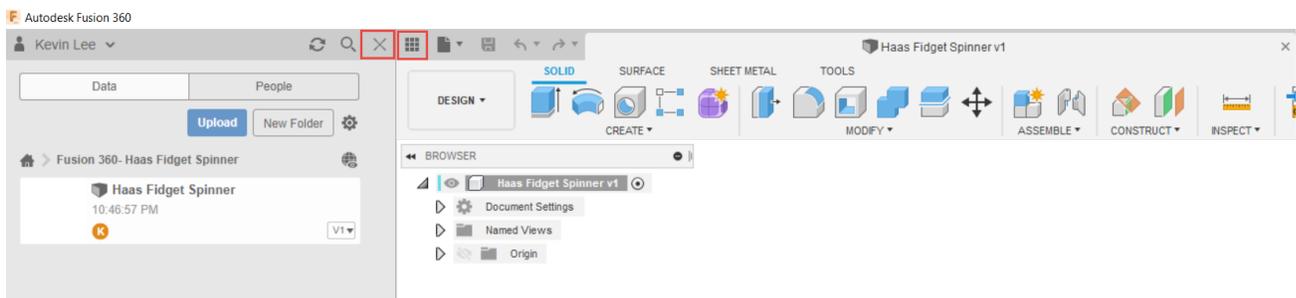


Step1: Creating your First Design

1. Note, your screen should look like the following.

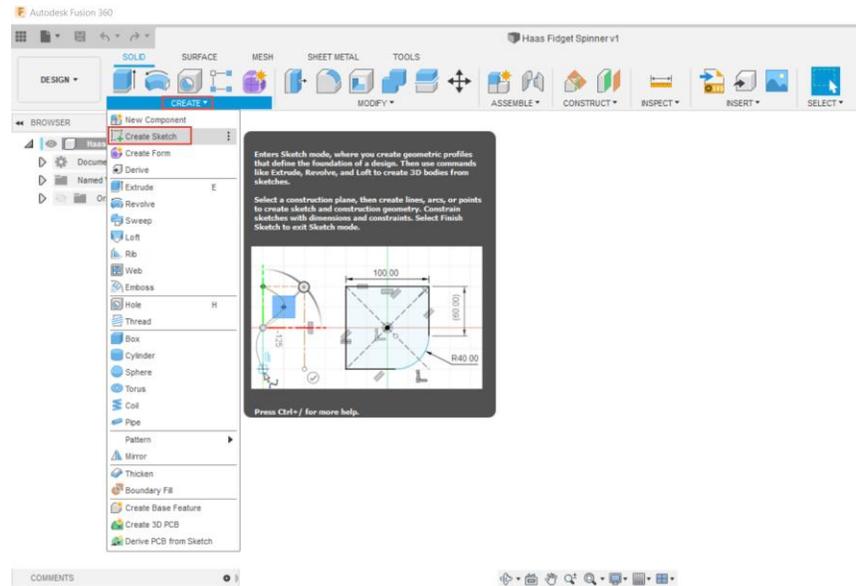


2. You can close the data panel to allow for space to create your first design. Close the data panel by selecting either the X close Button, or the data panel icon.

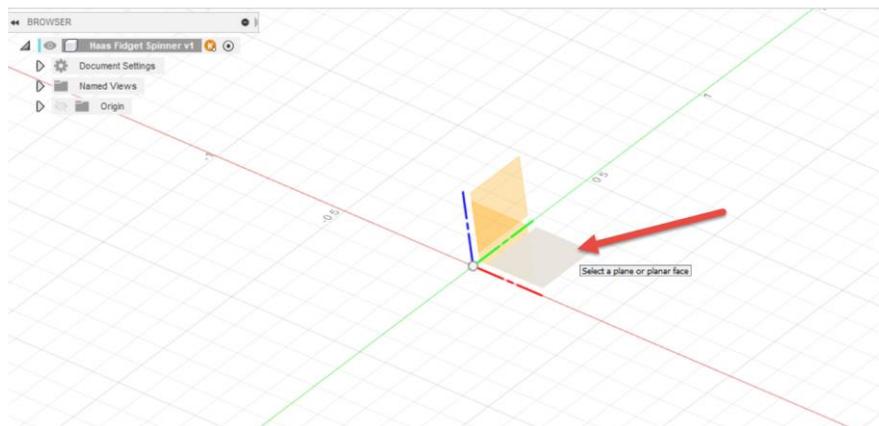


Step2: Sketching Geometry

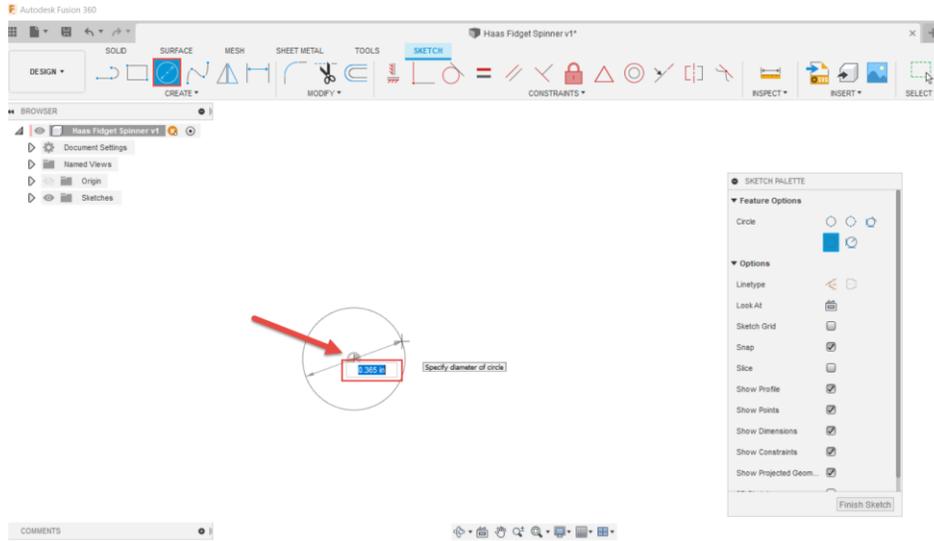
1. Working from the Design workspace, create a new Sketch by selecting the Create drop down menu, then select create sketch.



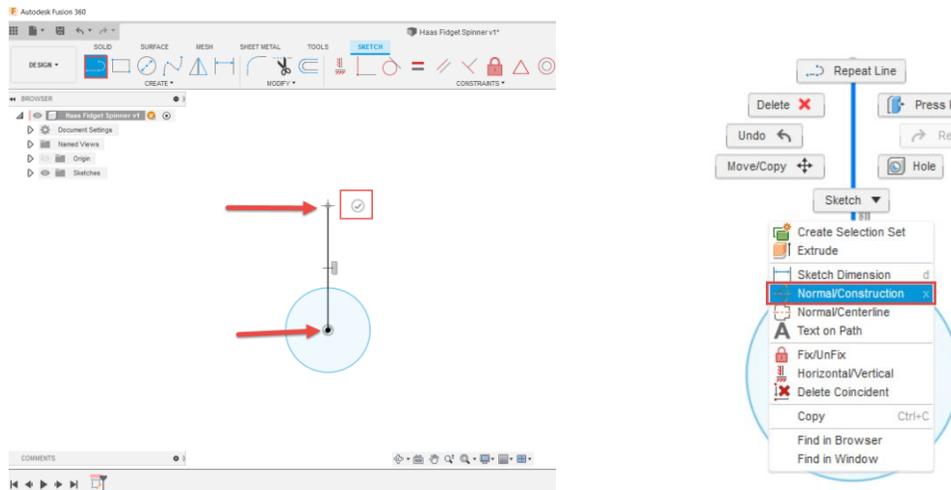
2. Select the top plane shown with the arrow below.



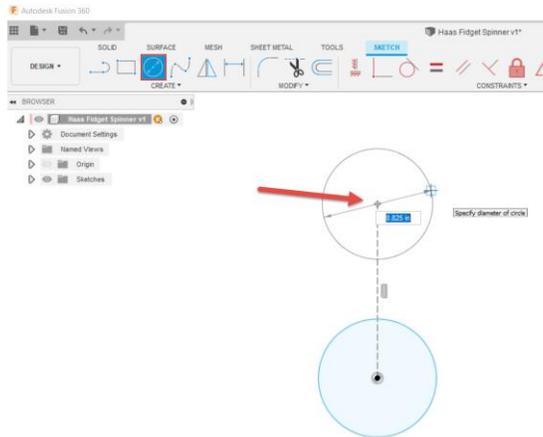
3. We will begin by sketching a few circular shapes by activating the circle command found in the ribbon bar.
4. Define the center point of the circle by selecting the design origin as shown with arrow below. Stretch out the circle and click a point to specify the diameter of the circle.



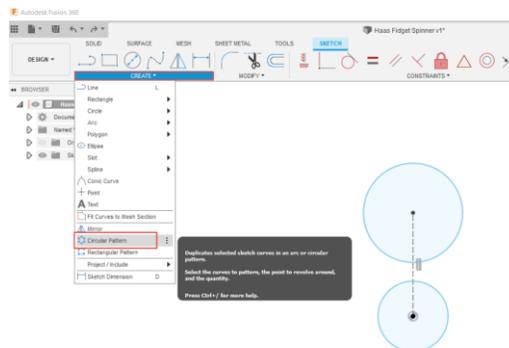
5. While the circle command remains active, press escape to deactivate the Create circle command.
6. Create a line entity, by activating the line command found in the ribbon bar. Select start point of line at the origin and stretch a line vertically above the circle as shown below. Select the green check mark to close the line command.



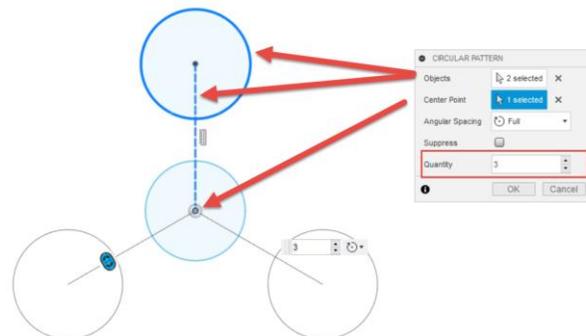
7. When creating sketches that will be used for creating 3d modeling features, sometimes it's convenient to create reference geometry known as construction geometry.
8. Right-click on the vertical line, and choose Normal/Construction, from the drop down menu. You will notice the solid line becomes a dashed line; meaning it can not be used towards feature creation.
9. Activate the circle command again, and create another circle with its CenterPoint at the end of the line, then select the escape key to deactivate the circle command.



10. With some basic geometry now created, you can replicate selected sketch curves in an arc or circular pattern.
11. Under the Create drop down menu, select and activate the Circular Pattern command.

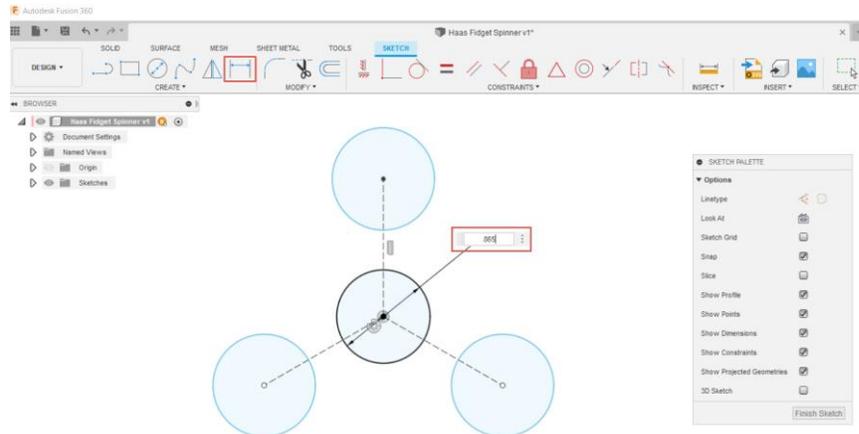


12. In the Circular Pattern command dialog box, select both the construction line and upper circle. Then to define the Center point of the circular pattern, choose the design origin point.
13. Be sure to adjust the circular pattern quantity to 3.
14. With these basic geometry shapes created we can further define this active sketch using both dimensions and constraint relationships.

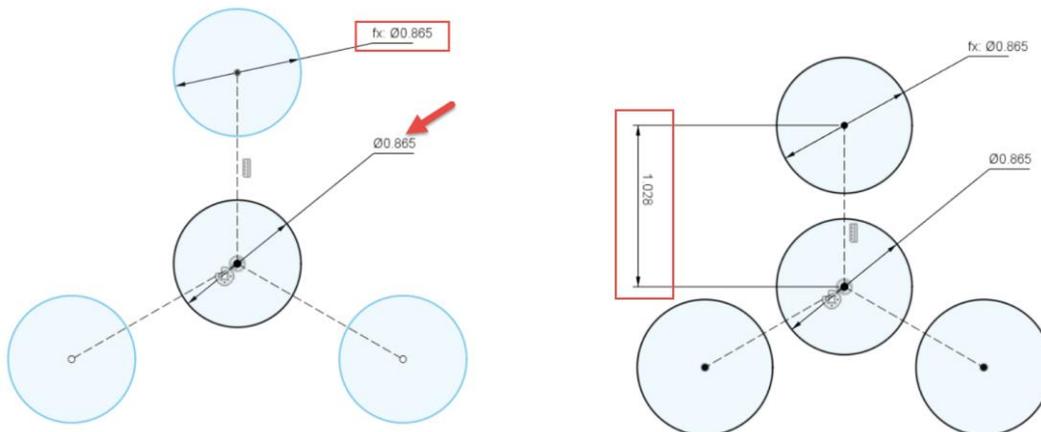


Step3: Sketch Dimensions and Constraint Relationships

1. You will notice that the construction lines are black and the circles are blue in colour. This means that the circle size and positions are not fully defined in this sketch. To fully define a sketch, you can use either dimensions or constraint relationships, or a combination of both.
2. First, define this sketch by activating the dimension command found in the ribbon bar. Alternatively pressing the D key on the keyboard as a hot key will activate the dimension command as well.

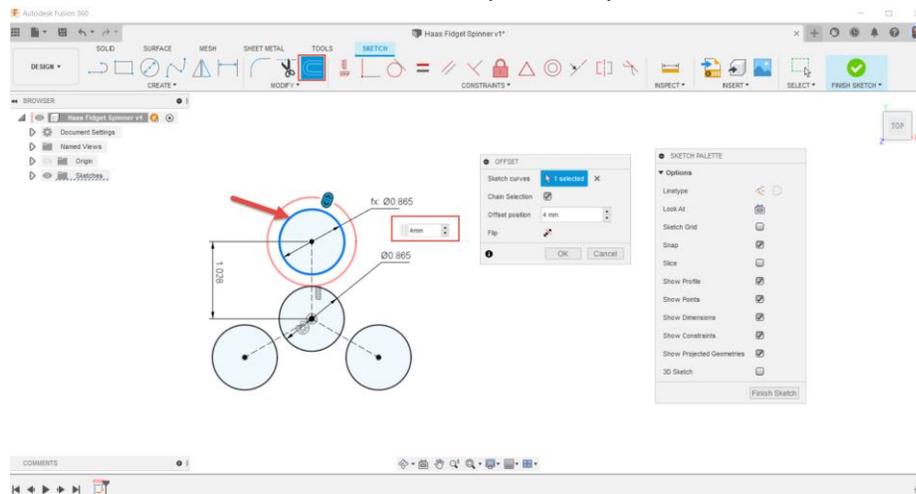


3. Select the center circle and drag out a dimension, and in the dimension box, enter 0.865”.
4. Instead of creating 3 more dimensions, create a new dimension on the top circle that is a function of the first dimension.

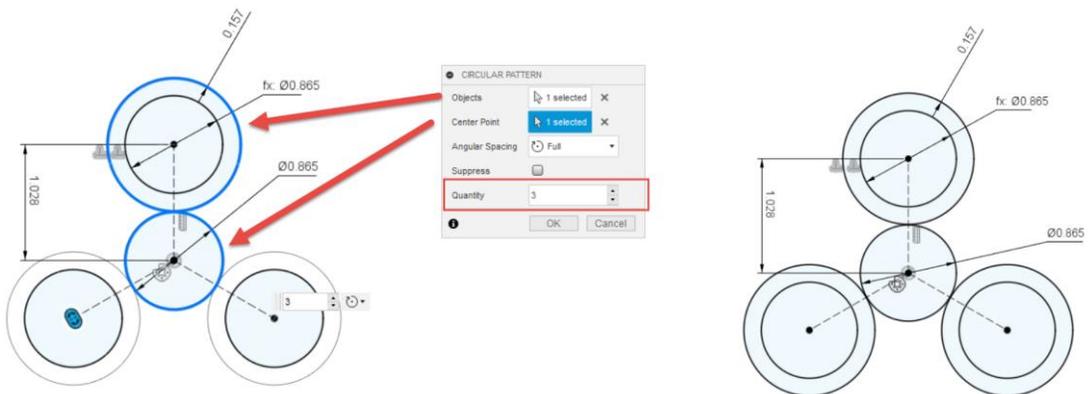


5. With the dimension command still active, select the top circle in this sketch, drag out a dimension. Instead of entering a number, select the 0.865” dimension. This will create a new dimension (d3) with fx beside it, meaning that it is equal to the 0.865” dimension. So, if we change the 0.865” dimension, the functioned dimension will update as well.
6. Define the length of the construction line as 1.028” shown above, right.
7. Notice that our sketch is now all black in colour, meaning it’s fully defined.

8. Next, use the Offset command to offset the top circle by 4mm.

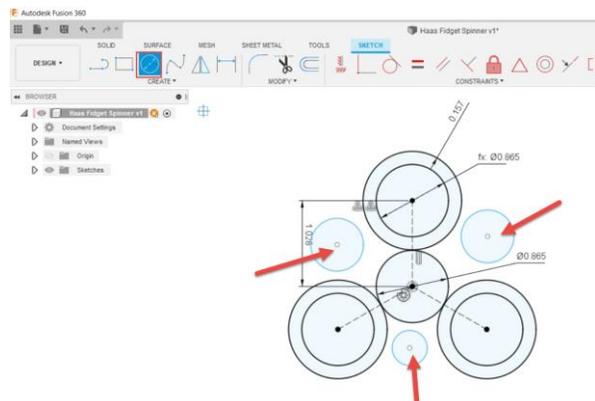


9. Repeat the Circular Pattern command to pattern the offset circle 3 times and complete the command with selecting ok.

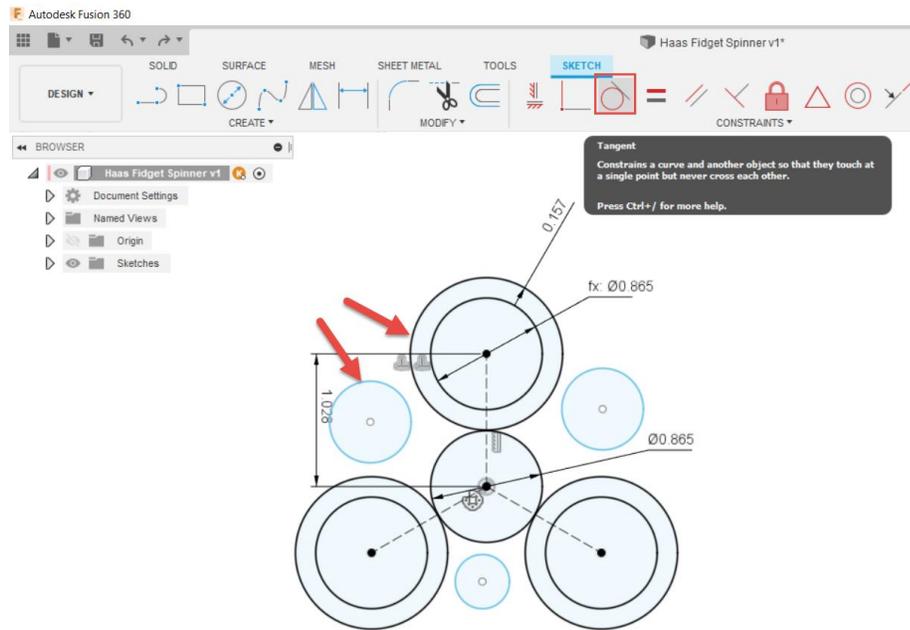


10. Your sketch should still remain fully defined having all geometries remaining black in colour as shown above right.

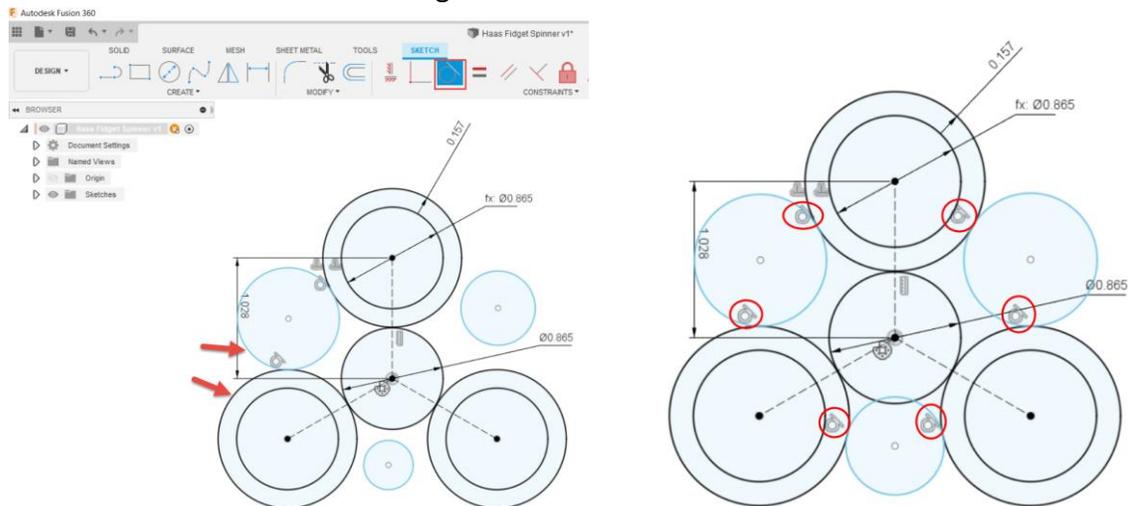
11. Re-activate the circle command, and sketch 3 new circles as shown below. For the moment, don't worry about the size or position of circles, but try to place them in approximate locations as shown below.



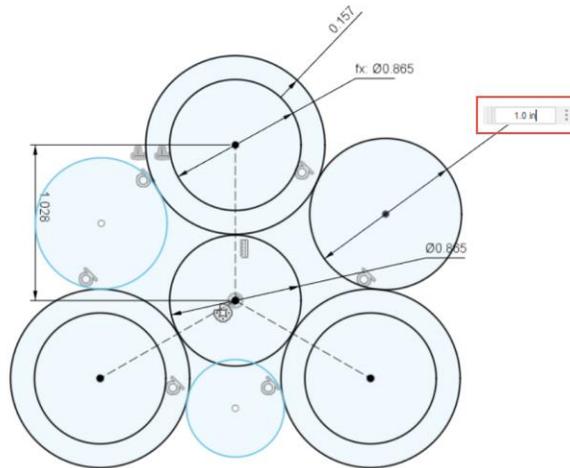
- As described earlier, to fully define a sketch, you can use either dimensions or constraint relationships, or a combination of both.
- Activate the Tangent constraint command and create Tangent relationships for each blue circle by selecting the blue circle and black circle as shown below so that the blue and black arc become tangent to each other.



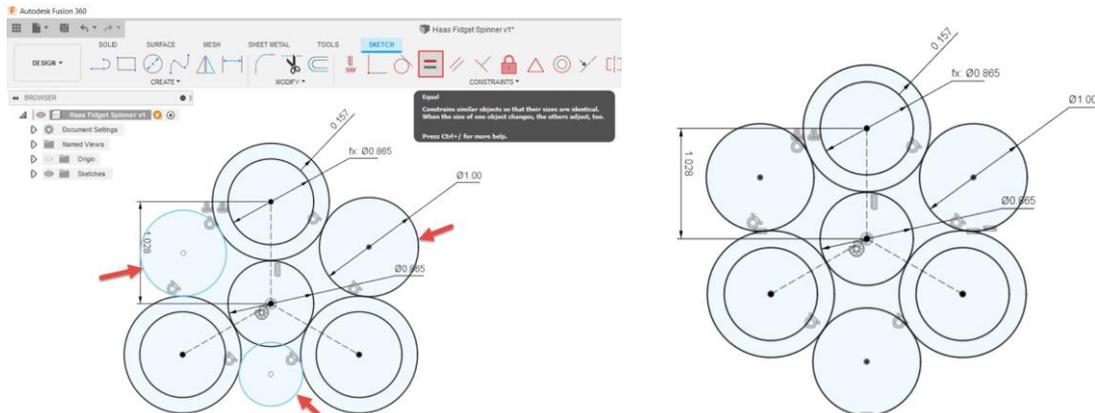
- Repeat this process once more for this same blue arc but select the lower black arc this time shown lower below left image.



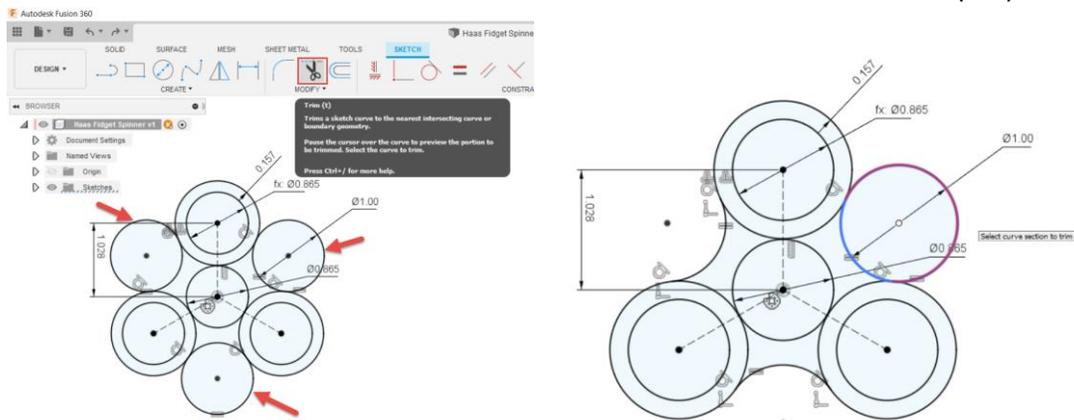
- Repeat this process again for the remaining two arcs. Once completed, you will see 6 tangent icon symbols on you sketch as shown above right.
- Notice that all 3 circles are still blue, they are tangent to the adjacent black arcs, and they are still different sizes.



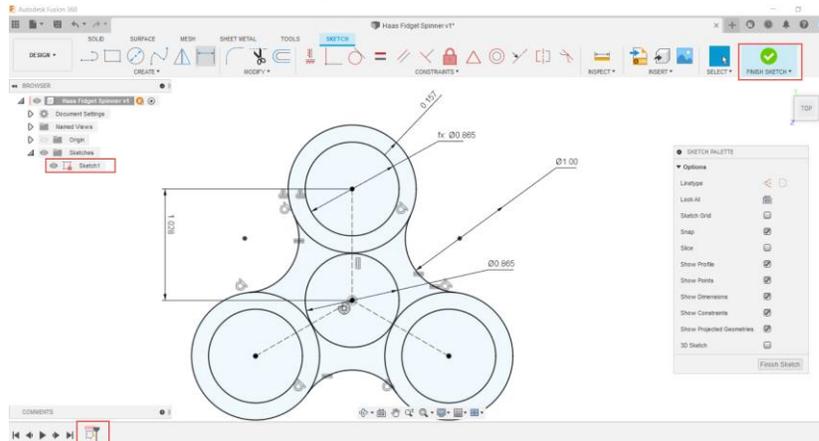
17. Create a new dimension to define one of the blue circles to have a 1.0" diameter as shown above.
18. Instead of creating two more dimensions, assign an Equal constraint relationship for the two remaining blue circles. Once complete, your sketch should become fully defined (black in colour).



19. Your sketch has some geometry that is not needed to complete 3D features.
20. Activate the Trim command and select areas of the arcs that we can trim (cut) away.



21. Your sketch should now be complete and fully defined. Choose Finish Sketch by clicking the green check mark in the upper right corner.

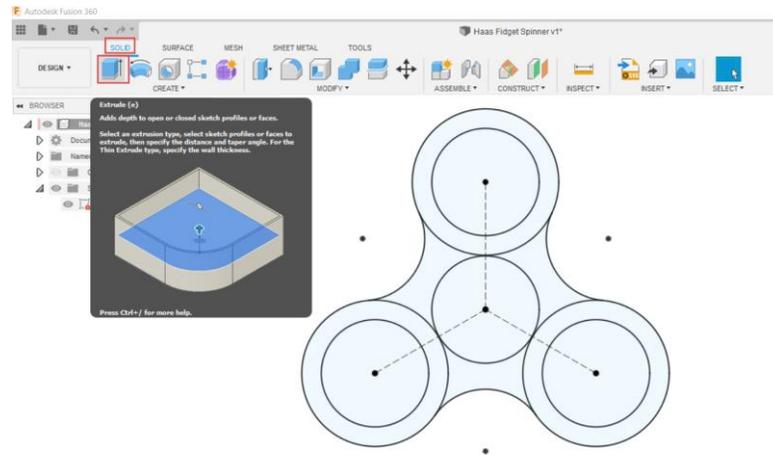


22. Click the **Save** icon  to save your design.

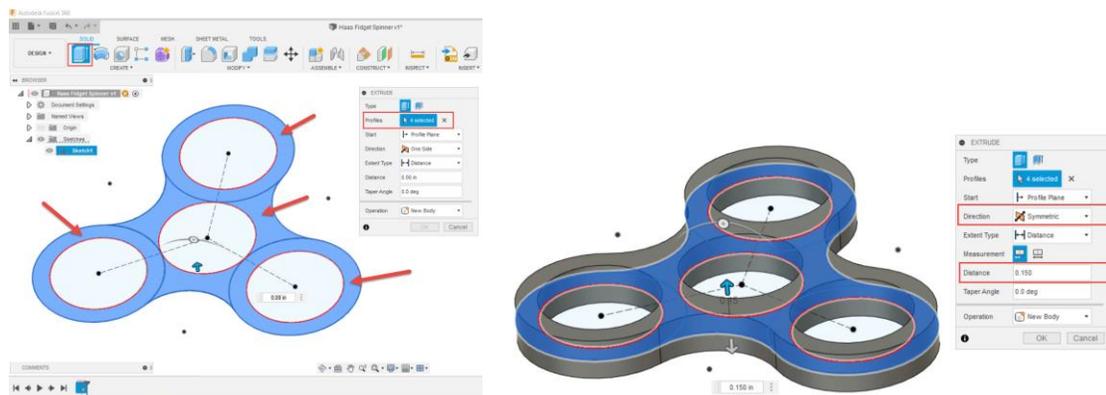
23. Notice in your browser there is sketch1 contained in the Sketches folder as well you can see the same sketch present in your design timeline along the bottom of your user interface.

Step 4: Modeling 3D Features

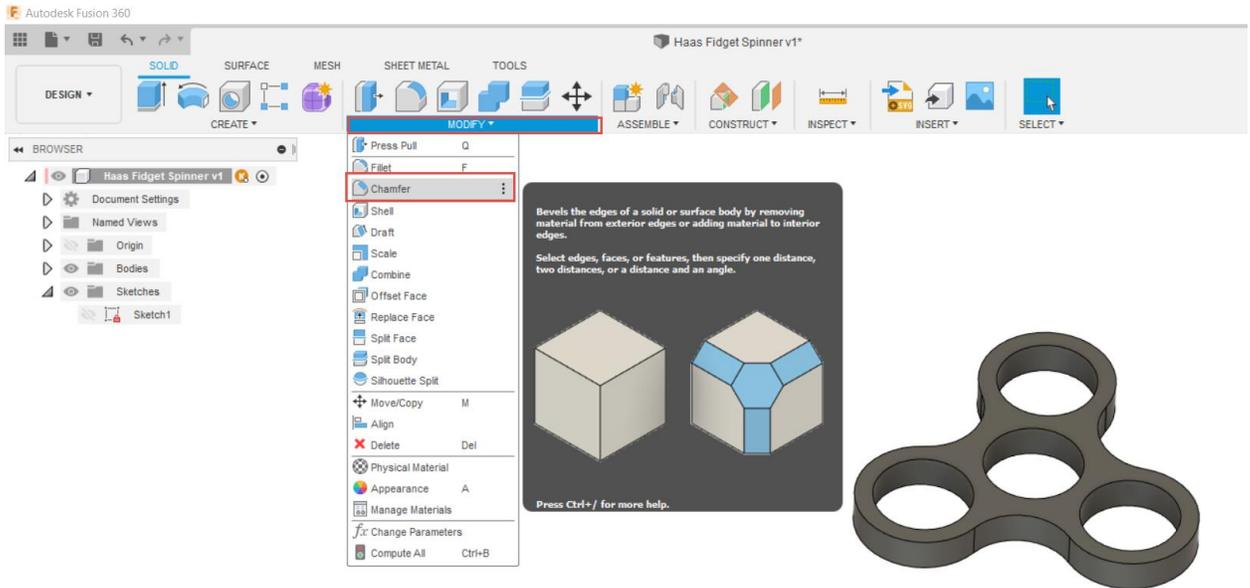
1. With having a 2D sketch fully defined, we can transition a 2d shape into a 3D CAD model containing multiple CAD features.
2. When working in the Fusion 360 Design workspace, you can toggle between Solid, Surface, Mesh and Sheet Metal modeling tools; each having a dedicated ribbon bar of icon feature commands.
3. In the Solid Modeling ribbon bar, try pausing you mouse over a few of them to read the tool tip flyouts that describe the different features in more details.



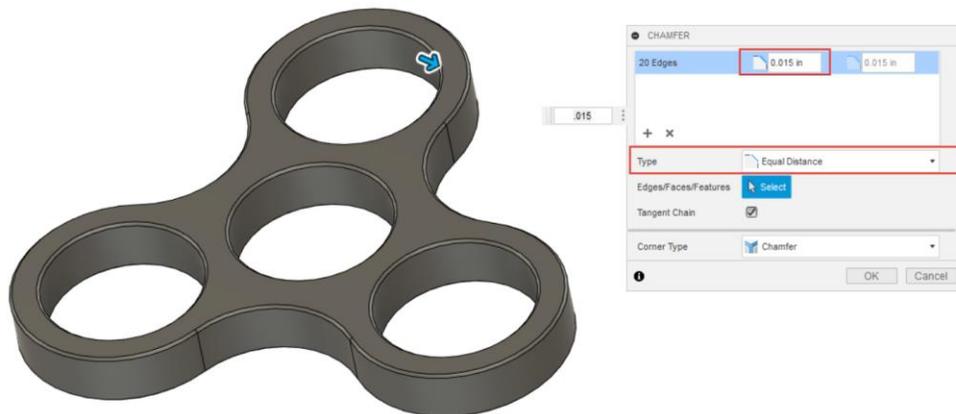
4. Activate the Extrude command and select the 4 closed profiles highlighted in blue, below left.



5. Grab the blue Extrude aero and drag upward to give these selected profiles some thickness. Define the extruded thickness as 0.150”.
6. Also, within the Extrude dialogue box, change the Direction to be Symmetric. This symmetric option extrudes the selected profiles 0.150” in each direction equally. Ensure that the operation type is set to New Body. Choose Ok to complete the Extrude feature.
7. Now modify the solid body geometry with creating a new Chamfer feature by activating the Chamfer command found in the Modify drop down menu.



8. Select all 4 circle edges and the outer edge profile, both on the top and bottom of the CAD model.



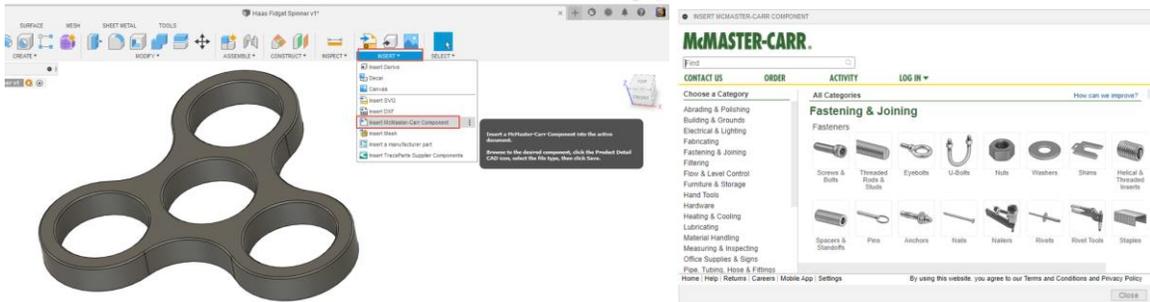
9. With all 10 edges selected, define the chamfer size to be 0.015". Ensure Chamfer Type is set to Equal Distance and choose OK.

10. Notice your Design Timeline at the bottom of screen contains one sketch and two model features.

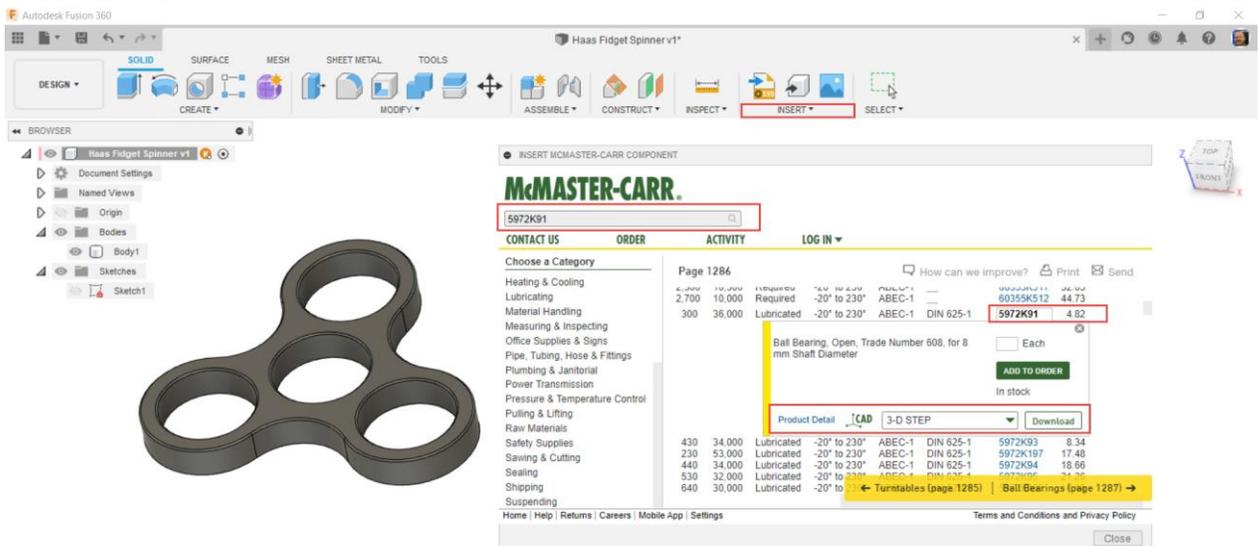
11. Click the **Save** icon  to save your design.

Step 5: Inserting Assembly Referenced Components.

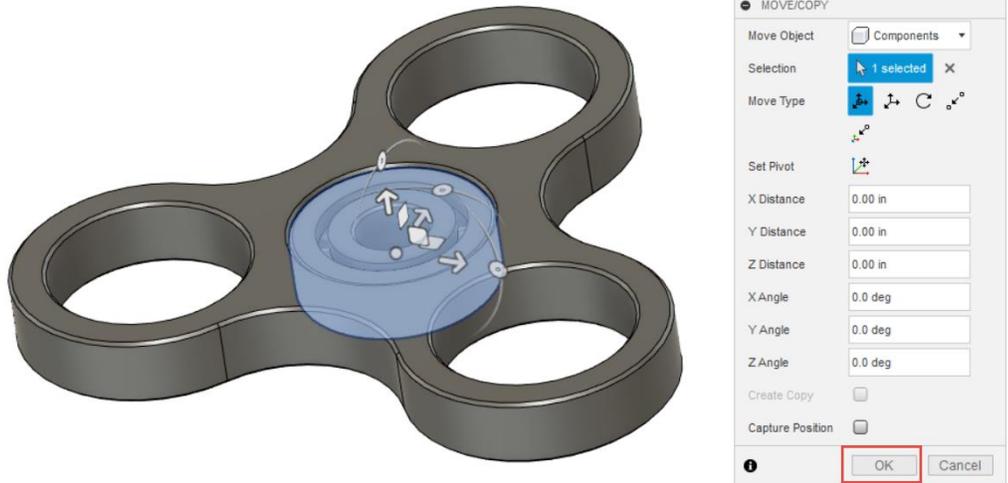
1. When creating your design in Fusion 360, you can design in many different ways from parametric or direct modeling techniques to importing geometry from other CAD systems. Being able to reference pre-modeled CAD data easily will help you model your finished design concept quickly.
2. Fusion 360 easily allows you to insert CAD model data from part suppliers such as McMaster -Carr. With direct linking access into massive CAD model libraries, you can easily browse to desired components, click on additional product details, select a specific file type, and save the CAD modeled components directly into your own design.
3. Under the Insert drop down menu, choose Insert McMaster-Carr Component.



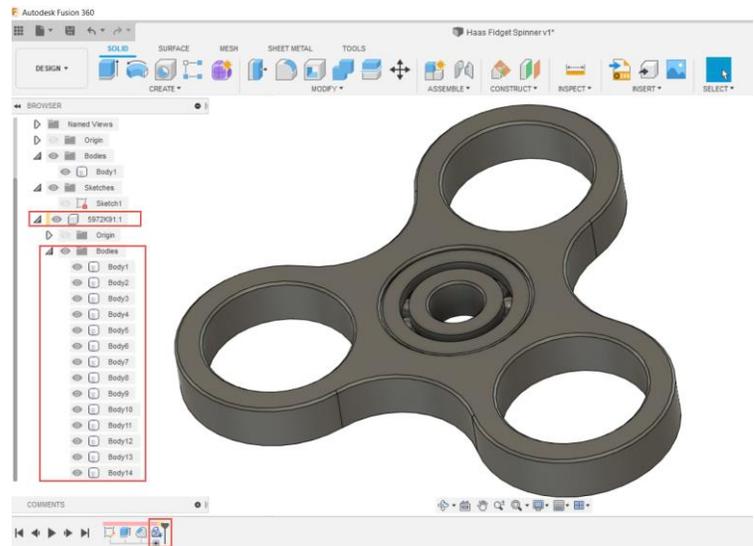
4. In the McMaster-Carr dialogue window, search for a specific sized metric bearing for your fidget spinner project.
5. Type bearing part number 5972K91 in the Find search field.



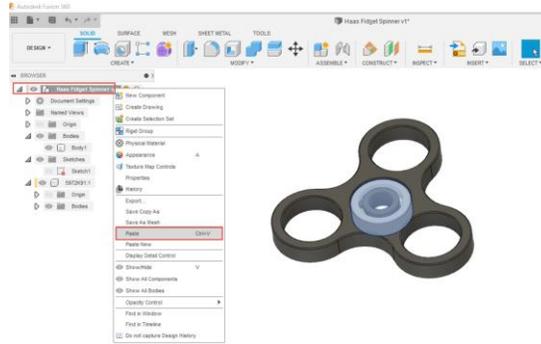
6. Once the specific bearing is located, near the Product Details, choose 3-D STEP file and click on Download.
7. The 3d bearing CAD model instantly downloads into your Fusion 360 Design document.
8. The design origin of the bearing 3d cad model locks onto your design documents origin. Click OK to insert bearing component into your fidget spinner design.



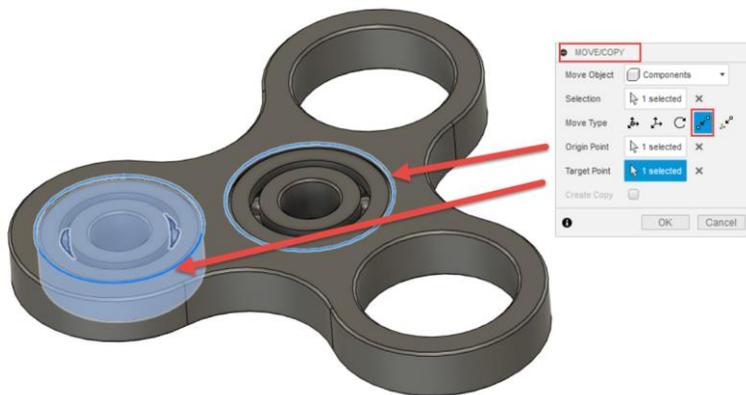
9. Notice your design timeline now shows an inserted base featured component.
10. This new bearing “CAD model” component shows in your browser as a separate component model containing multiple bodies of the various parts of the actual bearing assembly.



11. Minimize the bodies folder of the bearing component.
12. You can copy multiple instances of a design component.
13. Right-click on the bearing component in the browser, choose copy, then right-click on main component level of you design document, right click, then paste.



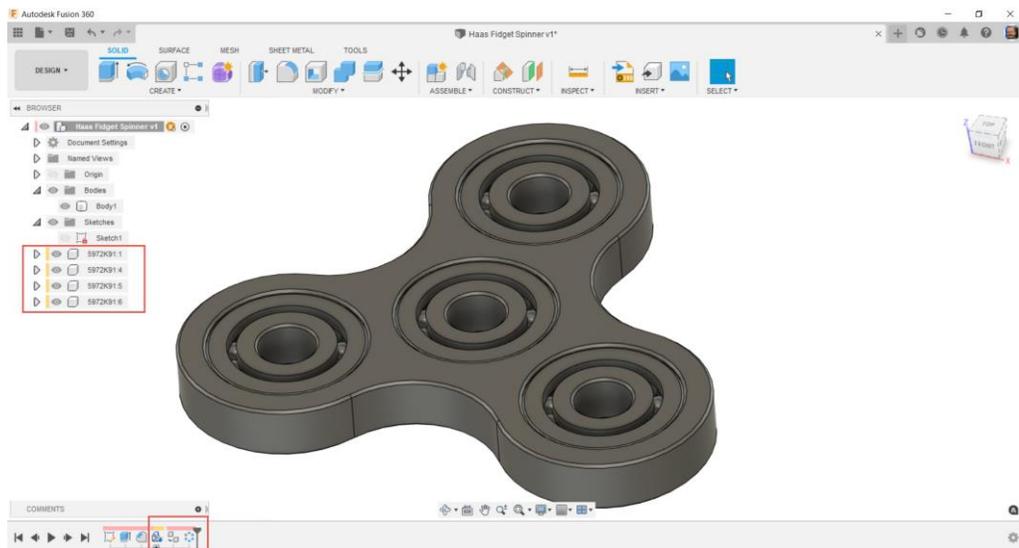
14. Notice in your browser that there is another instance of the bearing component.
15. With the Move/Copy dialogue box open, select the Point to Point move Type, then for the Origin point, select the center circle. For the Target Point, select the other circle as shown below. Note that for both circle selections, grab the top edge of chamfer circle edges on the top face of the fidget spinner.



16. Click OK to complete the Paste, Move/Copy command.
17. Your browser should now show two instances of the bearing components.
18. Instead of repeating the Copy/Paste command two more times; use the Circular Pattern modeling feature.
19. Under the Create drop down menu, choose Pattern then Circular Pattern.



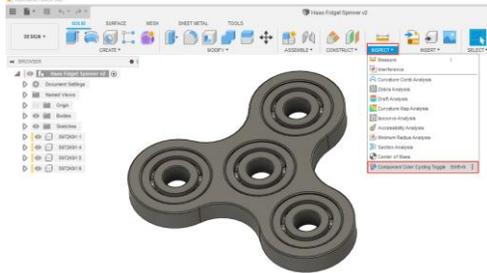
20. Set the Type to Components, and for pattern Object, select the secondary instance of the bearing component.
21. Select the ID face of the center bearing as Axis to rotate around.
22. Ensure Pattern Quantity is set to 3. Click OK to complete the Circular pattern.
23. Take notice of your design time line and design browser.
24. The browser shows that your Fidget Spinner design documents contains a total of 4 additional bearing components.
25. Your design timeline highlights 3 specific design features; Insert Base Feature, Copy/Paste, and a Circular Pattern.



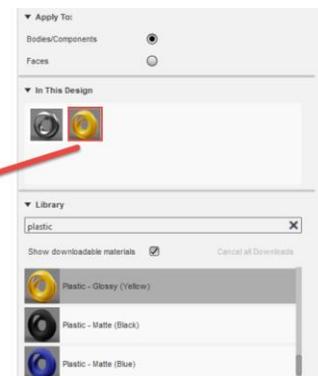
26. Click the **Save** icon  to save your design.

Step 6: Adjusting Component Appearances

1. To complete your design look, you can adjust the appearance of components of your Fidget Spinner Design.
2. To help see a colour difference between multiple components you can activate the Component Colour Cycle Toggle located in the Inspect drop down menu. This applies different colours to each component to help differentiate between the components.



3. Alternatively, if you deactivate Component Colour Cycling toggle in the same way; we can also assign specific desired colours to each component.
4. Let's assign a plastic appearance to the Fidget Spinner component.
5. In the Design workspace select the A key on keyboard to activate appearance selection.
6. In the appearance search field enter plastic.
7. At the top of the search filter list, left click and hold, then drag the colour appearance to the Design area of the appearance dialogue box.



8. Then left click and hold, drag design colour onto the component.
9. Click the **Save** icon  to save your design.

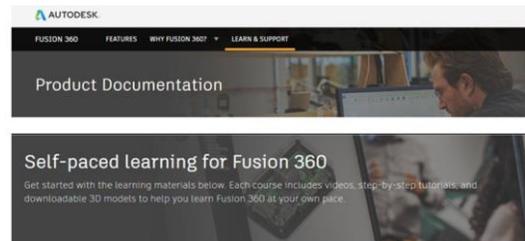
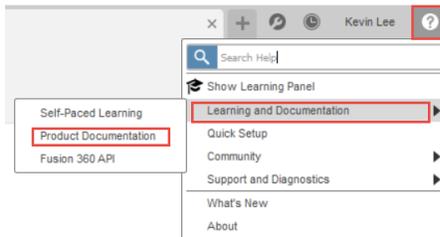
CONGRATULATIONS

You have successfully completed the guided lessons and designed your Haas Desktop Fidget Spinner project using Autodesk Fusion 360.



With Fusion 360 you can directly access additional learning resources. When inside of Fusion 360 near the top right corner of your screen, select the help icon.

Here you can access further self-paced learning tutorial guides and additional product documentation.



To learn more on how to get started with Fusion 360 or to learn how to use the basic solid modeling tools in Fusion 360, check out the self paced learning tutorial videos.

Getting started in Fusion 360

- Video series
- 17 videos
- Beginner

Solid modeling basics

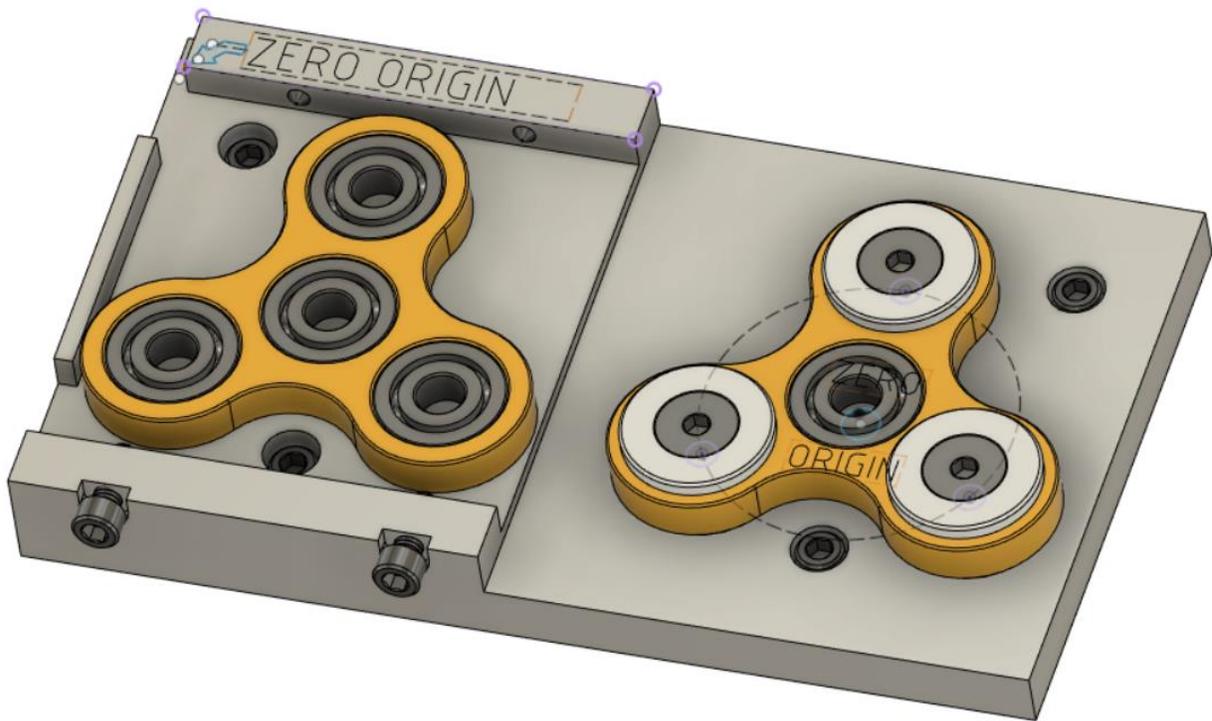
- Video series
- 12 videos
- Beginner

LESSON 4

CAD Modeling Manufacturing Assemblies

After completing this lesson, you will be able to:

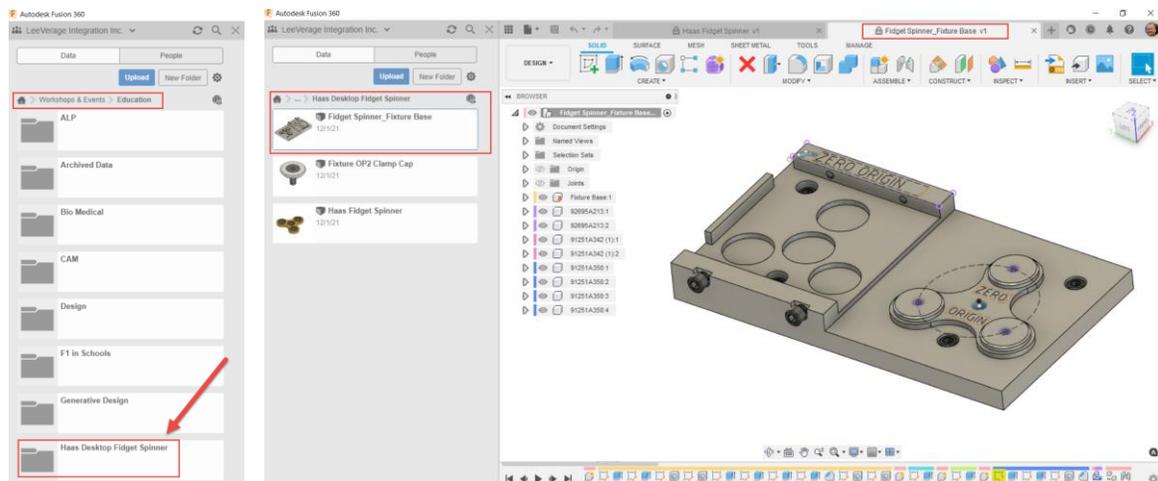
- Discover Fusion 360 Sample files
- Understand Fusion 360 Assemblies
- Assign Component Rigid Groups
- Create a Manufacturing Assembly
- Capture Design Intent with Joint Relationships



Step 1: Discover Fusion 360 Sample files

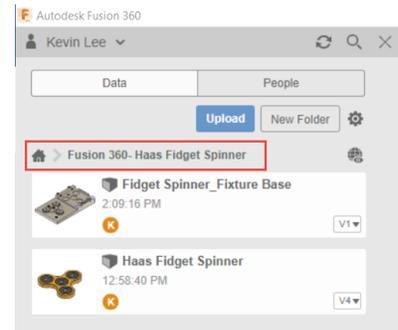
Fusion 360 contains many resources to help you with your design project. There are multiple sample directories containing CAM, design, electronic, generative design, and simulation sample files. These resources can be used to quickly build up your design intent to capture design and manufacturing workflows.

1. Open your data panel and scroll down towards the bottom where you will find many of these sample libraries.
2. At the bottom you will find a project folder named Workshop & Events.
3. Open the Workshops & Events project, and continue to open the Haas Desktop Fidget Spinner project folder.

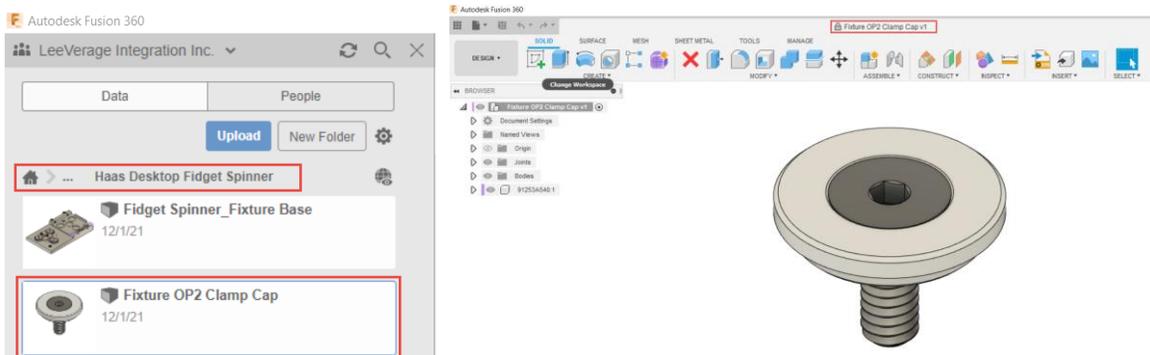


4. Open the Fusion 360 sample document named **Fidget Spinner_Fixture Base**.
5. Note that you now have two active documents currently open in Fusion 360 as shown with the tabs at the top.
6. However, the Fidget Spinner_Fixture Base document is currently protected showing with the padlock beside the document name.
7. Proceed to save this document into your own **Fusion 360 -Haas Fidget Spinner** project by selecting the Save As option and ensure that saves into your own local project folder.

- Once you have completed the save operation, the document is no longer protected as a Sample file.



- Repeat this same process to save the Fixture OP2 Clamp Cap sample document into your own local project folder.
- Open the Workshops & Events project, and continue to open the Haas Desktop Fidget Spinner project folder.
- Open the Fusion 360 sample document named Fixture OP2 Clamp Cap.



- Similarly, as before, the Fixture OP2 Clamp Cap document is currently protected showing with the padlock beside the document name.
- Proceed to save this document into your own **Fusion 360 -Haas Fidget Spinner** project by selecting the Save As option and ensure that it saves into your own local project folder.

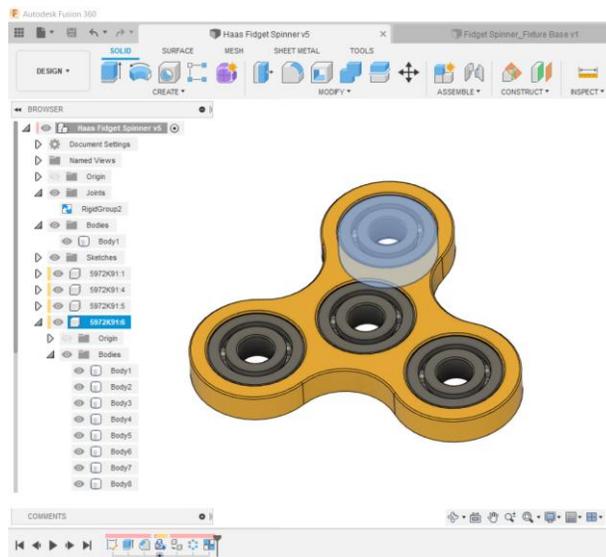
Step 2: Understanding Fusion 360 Assemblies

In Fusion 360, an Assembly is a collection of components that function as a single design allowing you to capture your design intent. Any design that contains two or more components is considered an assembly.

You can create assemblies that leverage a variety of different strategies, based on the needs of your design. You may:

- Create assemblies that consist of all internal components, all external components, or a mixture of both component types.
- Define the structure and relationships in the assembly at the beginning of a project or as you evolve the design.
- Take a top-down or bottom-up approach to the entire design, or vary your approach for specific components and subassemblies.
- Work alone on a design or collaborate with other project members.
- Collaborate concurrently or asynchronously with other project members.

A distributed design is a design with one or more external components referenced into an assembly.



Distributed designs enable multiple project members to edit different components in the assembly at the same time. As each project member edits components in context, the entire assembly updates to reflect their changes.

Step 3: Assigning Component Rigid Groups

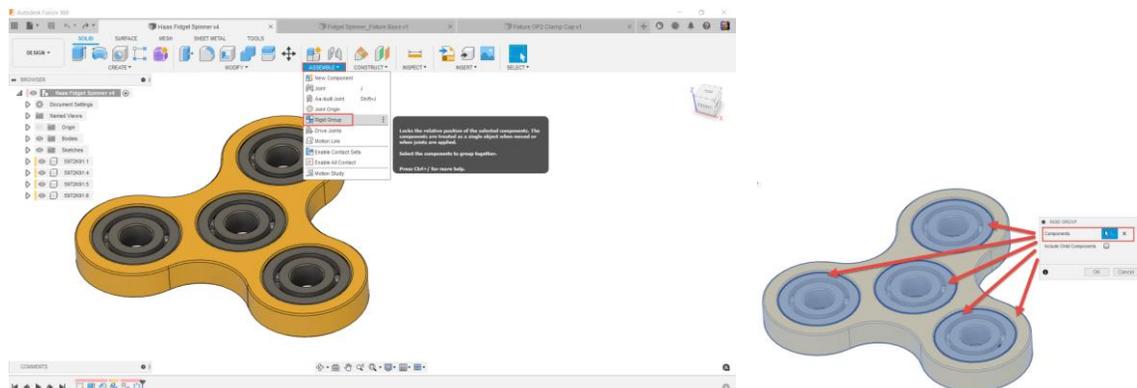
When designing in Fusion 360, you can create bodies, components and assemblies. An assembly is a collection of components that function as a single design in Fusion 360. There is no special file type for assemblies in Fusion 360, so each design can contain:

- A single component comprised of bodies
- An assembly of components and nested subcomponents.

When you create a new component in a Fusion 360 design, it becomes an assembly. You can use the position, joint, and motion features in Fusion 360 to define relationships between components in a design that allows you to capture design intent.

To simplify the process of positioning components within a complex assembly, you can create several smaller assemblies; apply joint, rigid groups, and add contact set relationships as needed. This approach lets you position the subassembly as a single unit in upstream assemblies.

With defining a Rigid Group, this allows you to lock the relative position of the selected components. The components are treated as a single object when moved or when joints are applied.

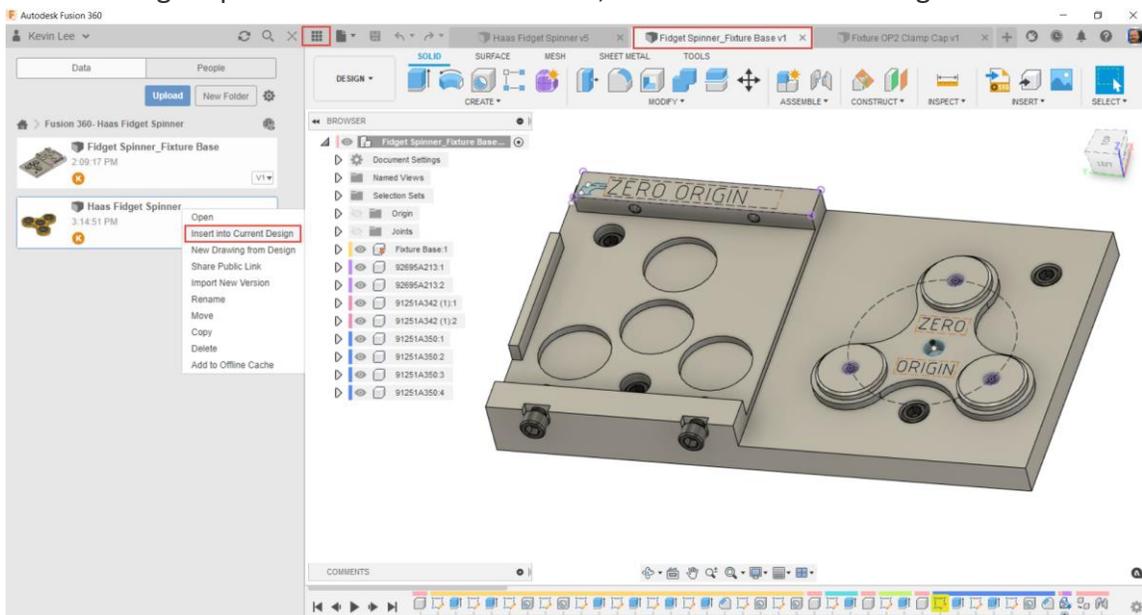


1. Activate your current open document named **Haas Fidget Spinner** by selecting its tab at top of screen.
2. Under the Assemble drop-down menu, select Rigid Group.
3. Select the Fidget Spinner component and all four sub-assembly bearing components as shown above.
4. Notice in your design time-line that you have a Rigid Group feature that is also present within the Joints folder in the browser.
5. Click the **Save** icon  to save your design.

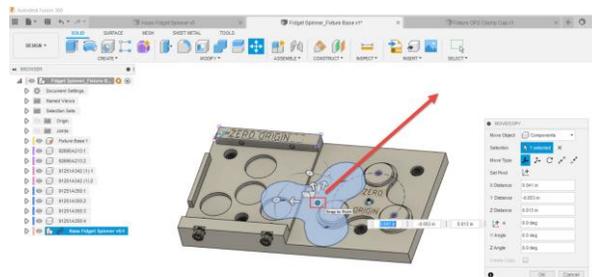
Step 4: Create a Manufacturing Assembly

Creating manufacturing assemblies in Fusion 360 is helpful so that you can visualize your CNC machining setup. There are options in a manufacturing CAM setup where you can highlight bodies or components to specify recognised fixtures, vices, chucks, clamps, etc. that toolpaths need to avoid when activating toolpath shaft and holder detection.

1. Activate your current open document named **Fidget Spinner_Fixture Base** by selecting its tab at the top of screen.
2. Expand open your Data Panel.
3. Within your Data panel project folder, you should see two document files. Right-click on the Haas Fidget Spinner document and choose, Insert into Current Design.

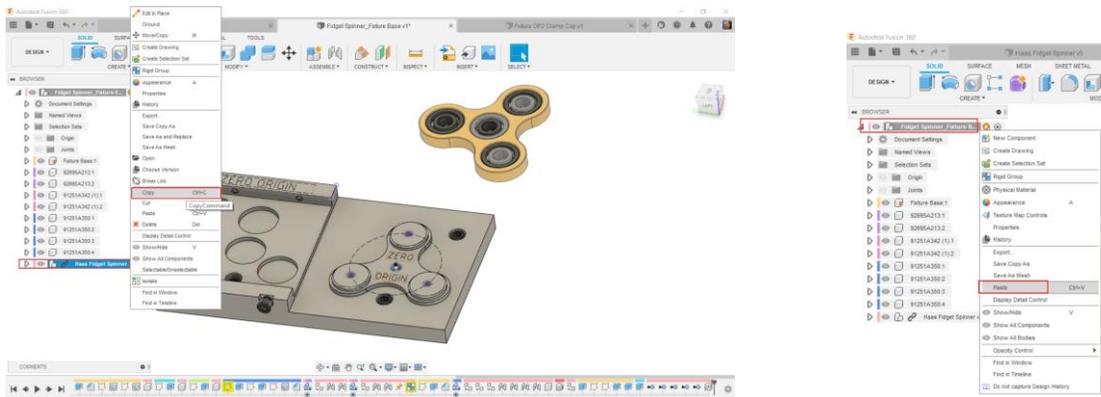


4. Close the data panel to make your screen larger.
5. With inserting a referenced design into your current design; the Move/Copy command becomes active.
6. For now, you can Free Move the Fidget Spinner component away from the Fixture Base component; left-click and hold component reference origin, and drag toward the red arrow direction shown below.

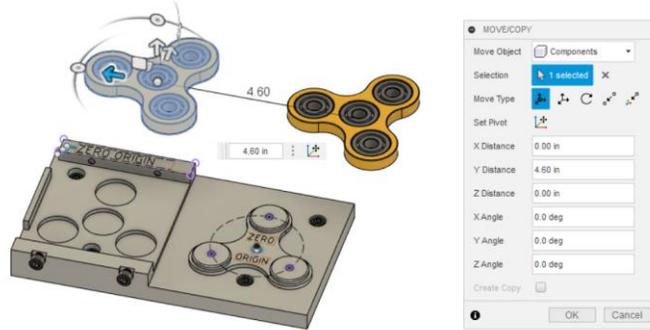


7. Click OK to complete the Move command.

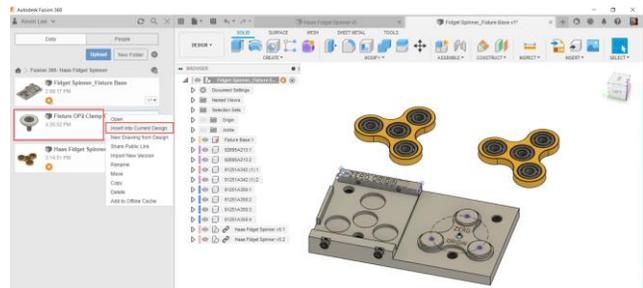
8. Create another instance of the Haas Fidget Spinner into your manufacturing assembly.
9. In the browser, right-click on the Haas Fidget Spinner component and choose Copy shown below left.



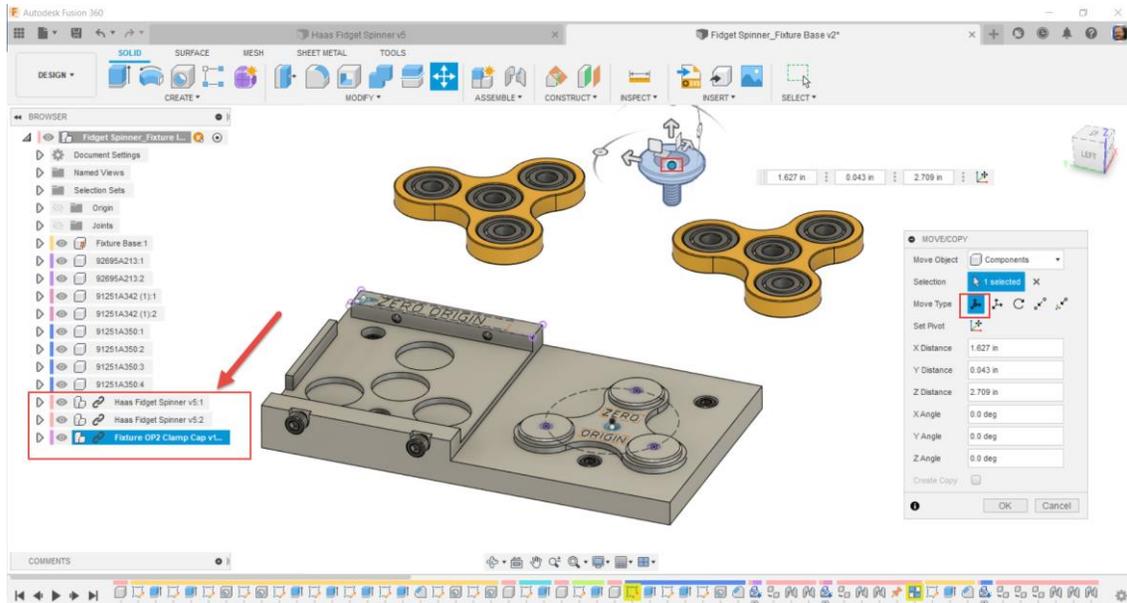
10. Then right-click on document component level and select paste as shown above right.
11. The secondary instance of the Fidget Spinner component is on top of the first instance.
12. Before you close the Move/Copy command, left-click and hold the blue arrow to free drag the secondary instance to the left.



13. With your Haas Fidget Spinner design as a referenced (linked) document inside of your Fidget Spiner_Fixture Base document; design changes will be automatically referenced to keep all design documents up to date and current.
14. In this case, we have created two instances of the Haas Fidget Spinner into our manufacturing assembly so that we can visualize the manufacturing workflow moving from side 1 machining operations through to side 2 machining operations all with using one work holding fixture.
15. Re-open you Data Panel and right-click on the Fixture OP2 Clamp Cap component to insert into your current design.



16. Close your data panel to make your user interface larger.
17. Before you close the Move/Copy command, left-click and hold the blue arrow to free Move drag the secondary instance to the left.



18. Take note of your browser in that you now have three linked sub-assembly document components in your manufacturing assembly document.
19. Click the **Save** icon  to save your design.

Step 5: Capture Design intent with Joint Relationships

In the Design workspace in Fusion 360, you can create joints between components in an assembly.

A Joint is a mechanical relationship that defines the relative position and motion between 2 components in an assembly. As you design an assembly with moving components, simulating motion is an important part of the prototyping process.

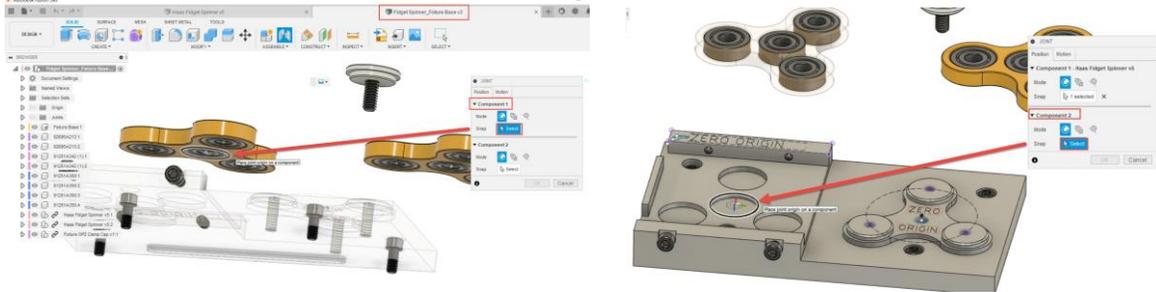
There are 7 different types of joints in Fusion 360. You can create joints with both the Joint and As-Built Joint commands. Each joint type uses different degrees of freedom to define motion.

Type	Description	Motion Allowed
	Rigid Locks components together. Removes all degrees of freedom.	None
	Revolute Component rotates around Joint Origin.	1 rotation
	Slider Component moves along a single axis.	1 translation
	Cylindrical Component rotates around and moves along a single axis.	1 translation 1 rotation
	Pin-slot Component rotates around one axis and moves along a different axis.	1 translation 1 rotation
	Planar Component moves along two axes and rotates around a single axis.	2 translation 1 rotation
	Ball Component rotates around all 3 axes using a gimbal system (3 nested rotations).	3 rotation

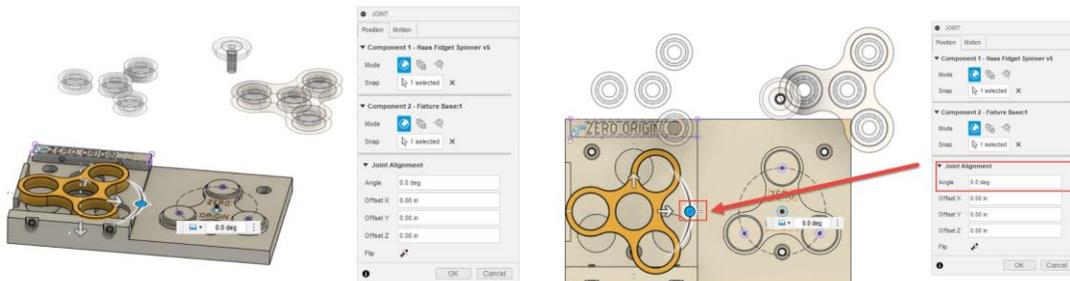
In Fusion 360, joints reduce the complexity of component relationships by helping you focus on what components can do, instead of what they can not do. This differs from other approaches like constraints or mates, which lock down available degrees of freedom (DOF) to limit motion. This requires you to understand the basic physics and math involved to ensure the components in an assembly move the way you intend.

In contrast, joints define motion between components, and only leave the required degrees of freedom open to create that motion. This reduces the need to understand the physics, and creates an understanding between the motion and the result.

1. Activate your current open document named **Fidget Spinner_Fixture Base** by selecting its tab at the top of screen.
2. In the design workspace under the Assemble drop down menu, choose the Joint command.
3. The Joint command dialogue box allows you to first select Component 1 origin snap point of the component that you want to move. See image below left.



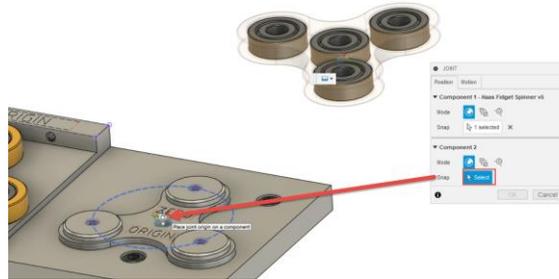
4. Select the circle edge on the bottom face of the left fidget spinner. Once selected, the component changes its transparency.
5. Then define Component2 snap position to be the left center hole edge of the fixture base component shown above right.
6. Once both component references have been selected, component 1 then moves to align itself with the reference point selected for component 2.



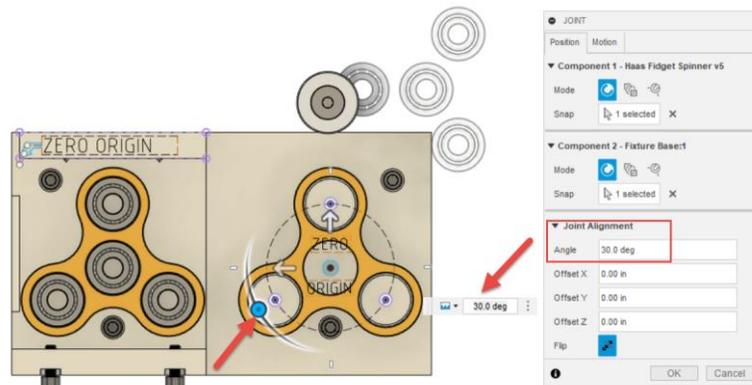
7. Before we accept this new position, ensure that it's rotated into the correct orientation.
8. To visualize the orientation perspective, choose the Top view on the view cube.
9. Adjust the rotation slider handle or enter -30 for the joint alignment angle parameter.
10. Click OK to accept this active joint relationship.



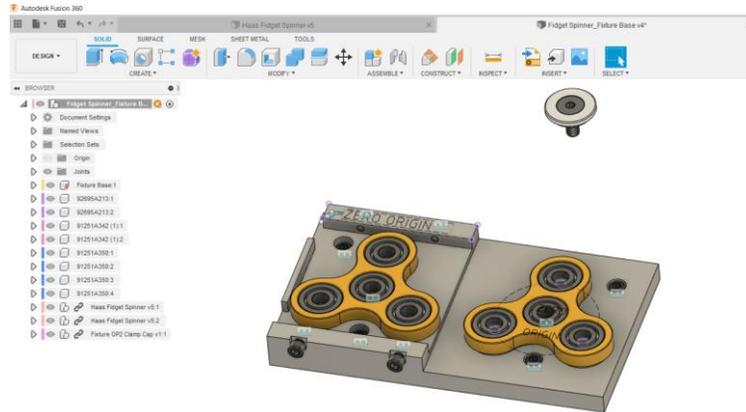
11. Once the joint relationship is complete, you will see that the four bearing components then move to be in the proper position as they belong to the rigid group relationship defined earlier within the Haas Fidget Spinner component document.
12. Repeat this process for the second instance of the **Haas Fidget Spinner** by creating a new joint command. Try using the keyboard J key as a keyboard shortcut to activate the joint command.
13. Select the center circle edge on the bottom face of the right fidget spinner and select the circle edge near the zero-origin text on the fixture base shown below.



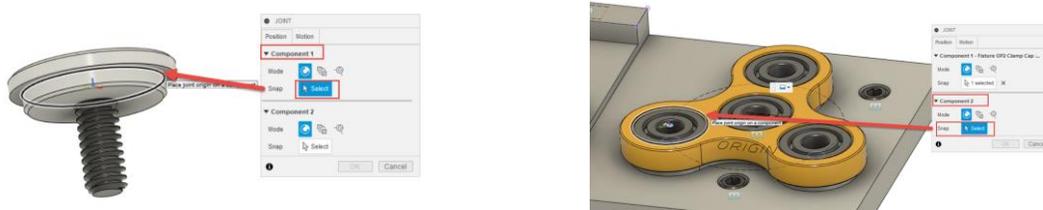
14. Note, the component 2 selection point will be on the same level as the Zero Origin text sketch.
15. Before we accept this new position, ensure that it's rotated into the correct orientation.
16. To visualize the orientation perspective, choose the Top view on the view cube.
17. Adjust the rotation slider handle or enter 30deg. for the joint alignment angle parameter.



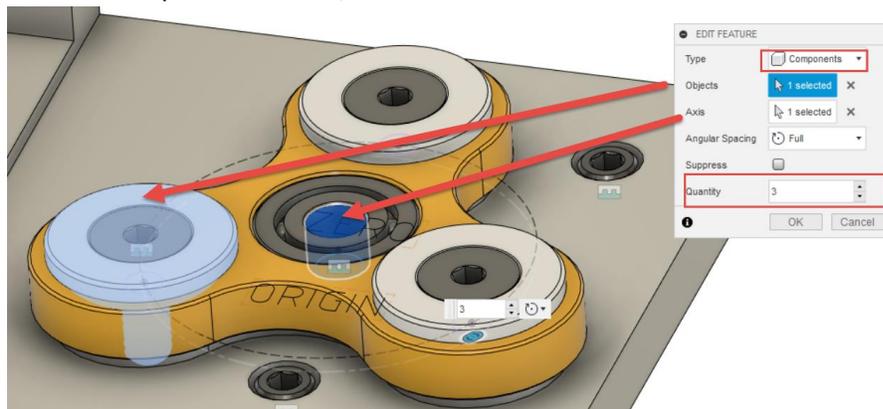
18. Click OK to accept this active joint relationship.
19. You should now have both Fidget Spinner components positioned accurately onto the fixture base component as shown below.



20. To complete the visualization of our manufacturing setup, proceed to create a new joint relationship that will position the **Fixture OP2 Clamp cap** that will securely hold down the fidget spinner component during the secondary machining operations.
21. Activate the Joint command and select the circular edge on the bottom side of the clamp cap as component 1 selection; then for component 2 selection, pick an upper circular edge of left bearing bore.

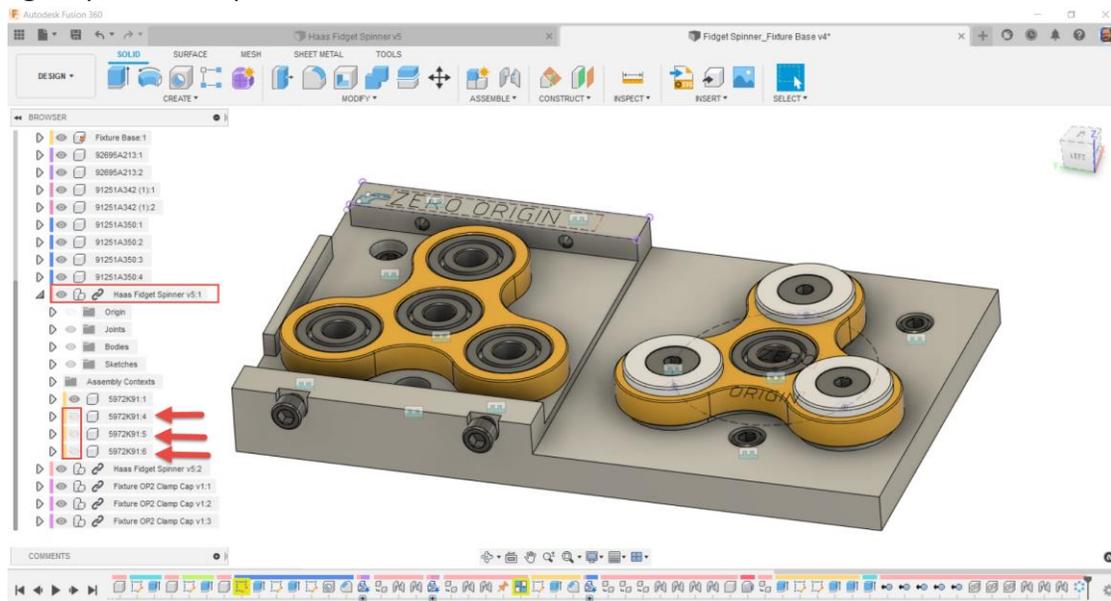


22. Choose OK to accept this rigid joint relationship created by default.
23. Under the Create drop down menu, choose Pattern then Circular Pattern.



24. Set the Type to Components, and for pattern Object, select the Fixture OP2 Clamp Cap component.
25. Select the ID face of the center bearing as the Axis to rotate around.
26. Ensure Pattern Quantity is set to 3. Click OK to complete the Circular pattern.
27. Take notice of your design timeline and design browser.
28. The browser shows that there is now a total of three Fixture OP2 Clamp Cap components in this manufacturing assembly.

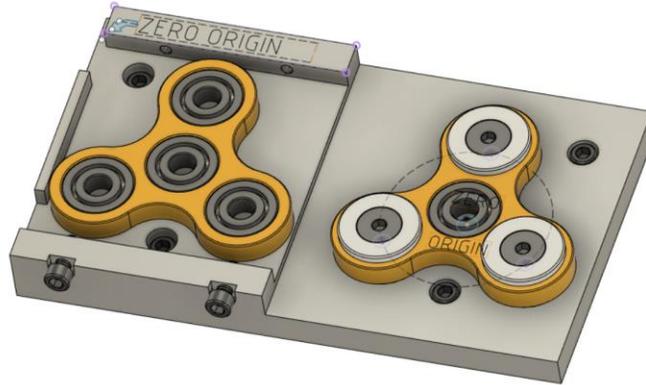
29. Considering the Clamp Cap components would actually be interfering with the three bearings, you can hide the visibility of the bearings inside of the first instance of the Haas Fidget Spinner component.



30. Click the **Save** icon  to save your design.

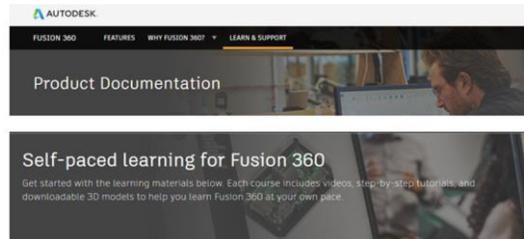
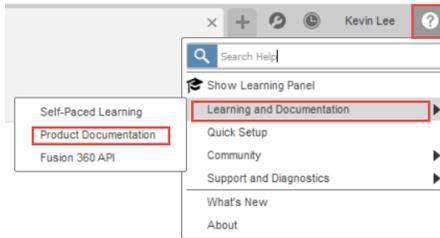
CONGRATULATIONS

You have successfully completed the guided lessons to create a CAD Model Manufacturing Assembly project using Autodesk Fusion 360.

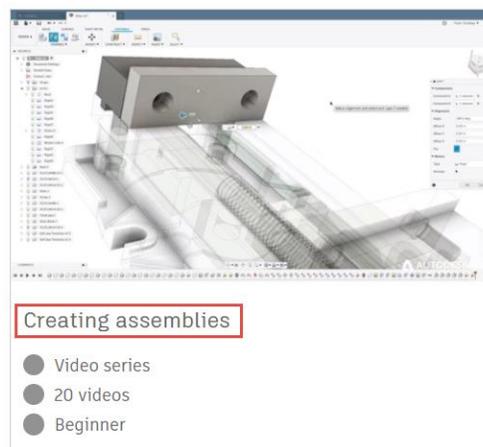
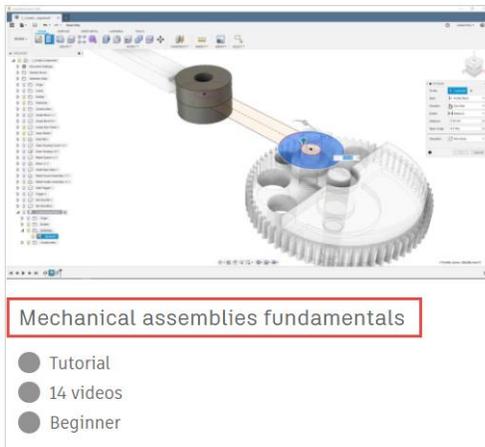


With Fusion 360 you can directly access additional learning resources. When inside of Fusion 360 near the top right corner of your screen, select the help icon.

Here you can access further self-paced learning tutorial guides and additional product documentation.



To learn more on the fundamentals for building mechanical assemblies in Fusion 360, check out the self paced learning tutorial videos.

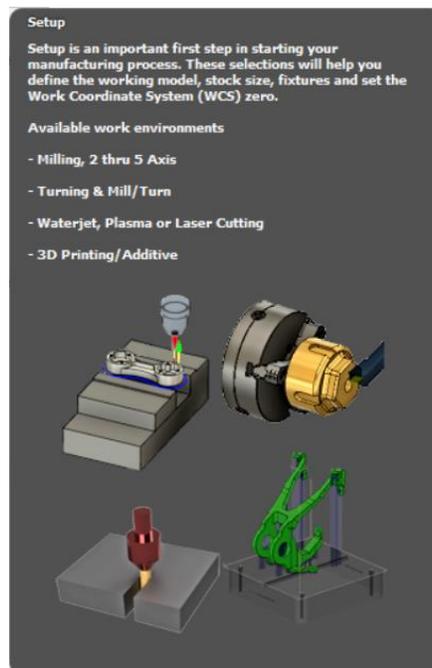


LESSON 5

Defining a Manufacturing Setup

After completing this lesson, you will be able to:

- Create a New Manufacturing Setup
- Define Work Coordinate System (WCS)
- Identify Program Origin
- Determine Stock Size and Orientation
- Establish Post Processing Output
- Create a Secondary CAM Setup



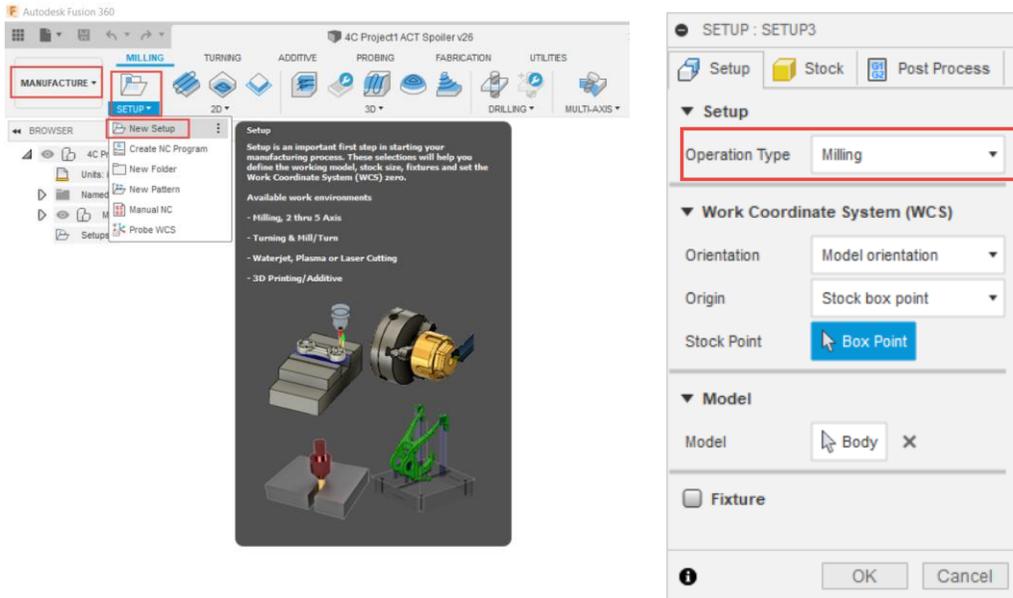
Setup is an important first step in starting your manufacturing process; where you'll tell Fusion 360 what type of toolpaths to create (ie. Milling, Turning), which models are parts, which models are fixtures and how big the raw stock material will be.

Understanding the importance of defining a manufacturing setup will define the purpose of having the Manufacture workspace in Fusion 360.

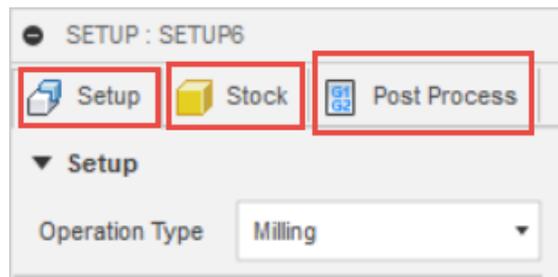
CAM stands for Computer Aided Manufacturing. CAM is the step where you convert the part model into a language that can be used for the manufacturing process (usually G code).

Step1: Create a New Manufacturing Setup

1. Ensure that you are currently in the Manufacturing workspace.



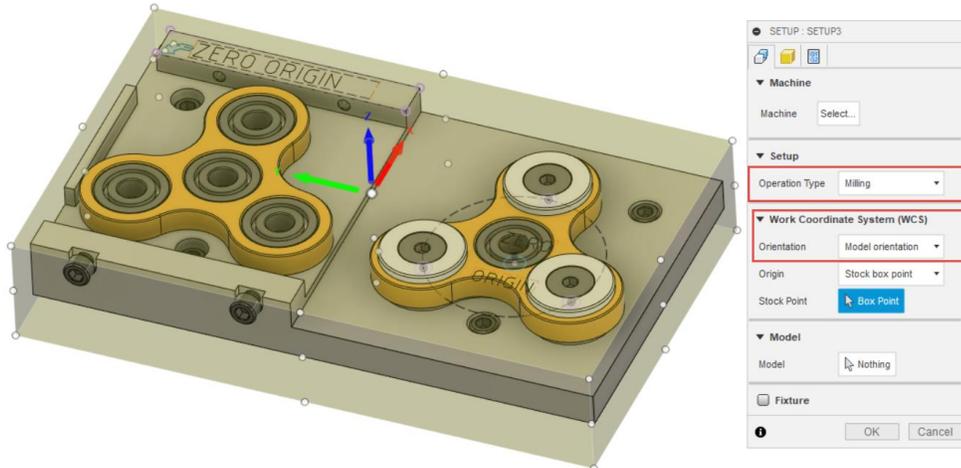
2. Create a new setup by selecting the Setup icon drop-down arrow, choose New Setup as shown above.
3. When defining a new setup, you will see three specific tabs; Setup, Stock, and Post Process.



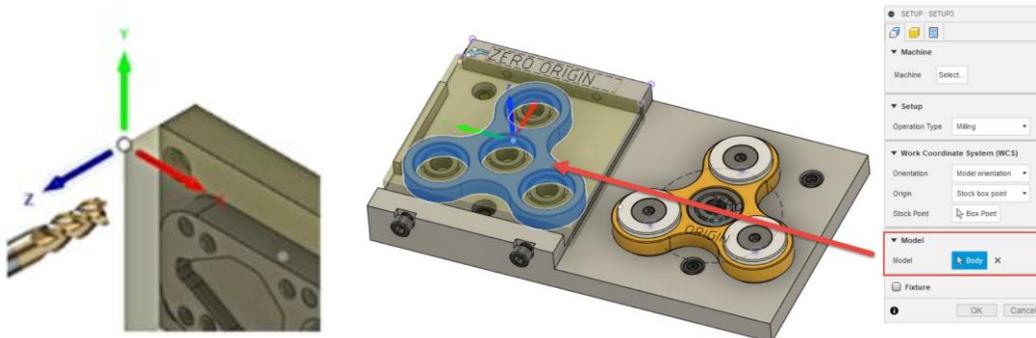
4. In the Setup tab, set Operation Type to Milling as shown above.
5. With Milling Mode selected, our setup specifies the manufacturing process to align with all types of milling machine configurations. Vertical and Horizontal milling machines along with multi axis machines configured with 4th and 5th axis capabilities can all be programmed in Fusion 360 with the setup operation type set to Milling.

Step2: Define Work Coordinate System (WCS)

1. By default, when creating a new setup, your Work Coordinate System (WCS) orientation is set to Model Orientation.
2. Model orientation uses the orientation of the current part for the WCS orientation. This means your setup WCS aligns to the standard plane orientation that was used when creating the cad model in the Design Workspace.

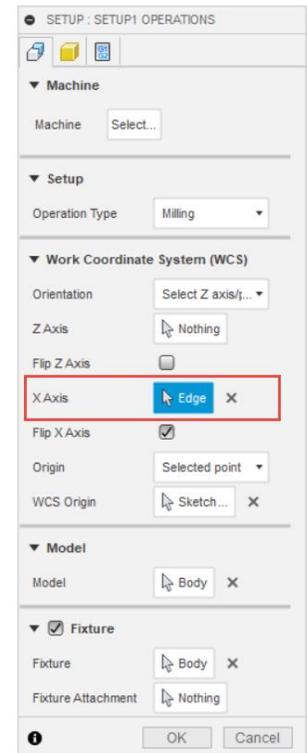
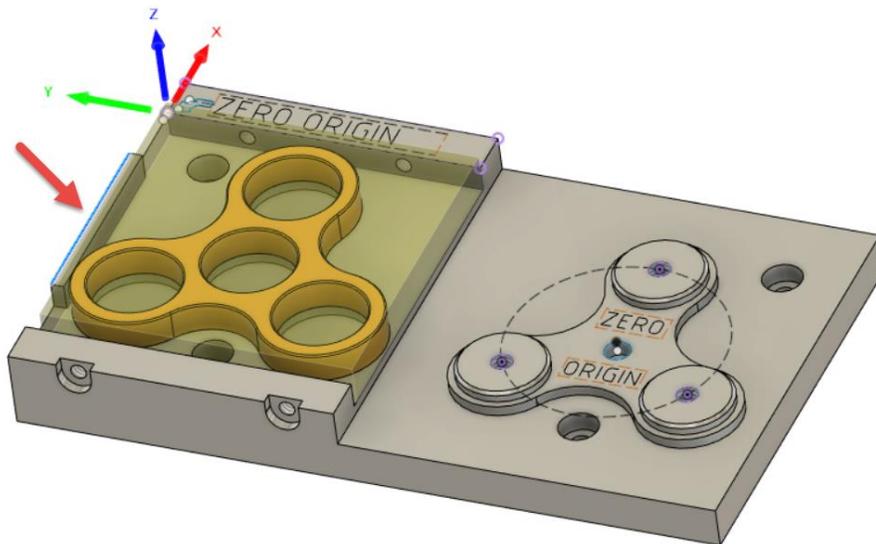


3. Notice in the browser under the model origin folder, the X,Y,and Z axis of the CAD model align with the WCS of your new Manufacturing Setup.
4. The setup WCS orientation is extremely important as this reflects the machining Z-axis tool plane for the setup. The Z axis should point away from the machining plane.
5. Note the orientation of the Z axis and the tool spindle axis orientation shown below.



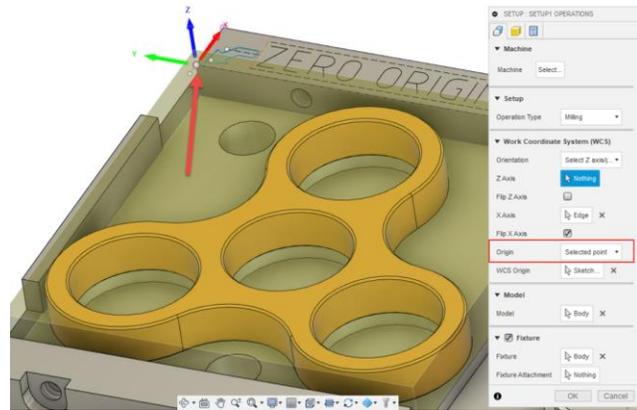
6. Notice that when you created this new CAM milling setup, Fusion 360 has encompassed all the model components with a transparent stock representation. In this case, we want to start with defining a CAM setup that will focus on side 1 machining operations.
7. You must identify the model component that you want to machine for side 1. Remember, the model bodies defined in the setup are used to drive the toolpaths as the CAM Setups create a relationship between the model and stock condition.
8. Select the model component to use for stock setup and ultimately toolpath calculations as shown above.

9. There are WCS orientation options available such as Select Z axis/plane & X axis, Select Z axis/plane and Y axis, and Select X&Y axis.
10. These options allow you to select various faces and or edges to define the Z axis and another face or edge to define either the X or Y axis.
11. However, the more visual way to align the WCS is to use the Blue, Red, Green arrows at the WCS intersection point.
12. Note, the arrows themselves are broken into two sections, the head of the arrow and base of the arrow.
13. Select the base of the red arrow to align the X axis of the WCS, then select a linear edge on the model that aligns correctly in the X direction.

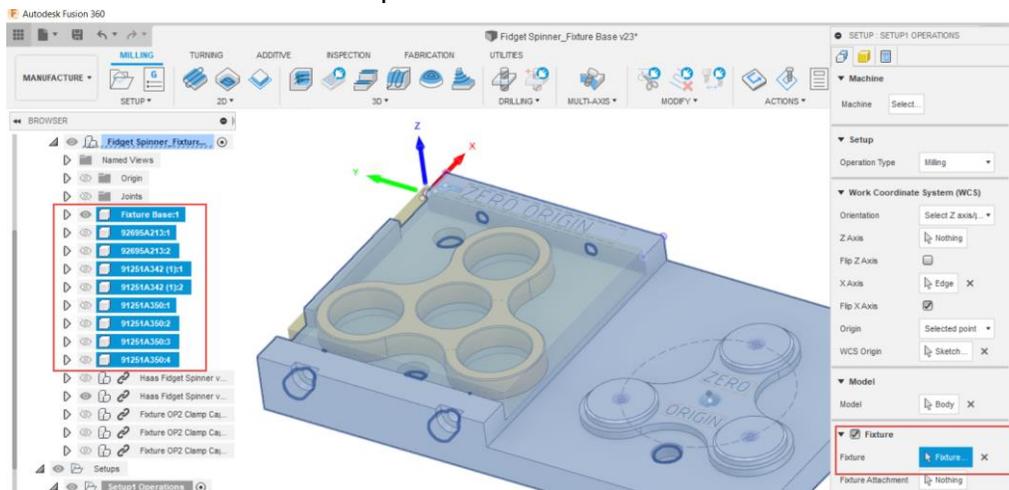


Step 3: Specify Program Zero Position

1. Once your setup Work Coordinate System (WCS) is defined correctly it becomes important to then specify where you want your program zero position to be. This position establishes the (G90 absolute) zero position for the NC g-code program.
2. The Origin drop-down menu provides options to allow selection of the Model Origin, Selected Point, Stock Box Point, and Model Box Point.
3. The WCS position is also shown with the white dot where the Blue, Green, and Red arrows intersect. This WCS position will represent the X zero, Y zero, and Z zero position of your NC G code program.
4. We want to match the WCS position to where the zero position is identified on the Fixture Base component.
5. Scroll the Origin drop down list and choose the Selected point option.
6. Then select the point highlighted on the fixture base as the zero origin.



7. Activate the fixture check box. The Fixture designation allows you to select other bodies or components that can be used during toolpath simulation to verify tool/holder collision detection.
8. Select the Fixture Base model component to be used as the Fixture in this CAM setup.



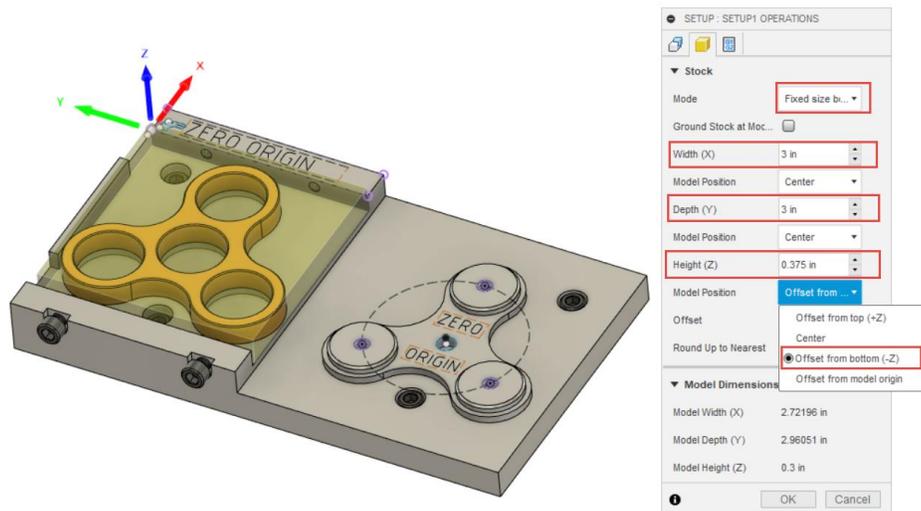
Step 4: Defining Stock Material size

When defining a new setup, you will see three specific tabs; Setup, Stock, and Post Process.

In the Stock Tab you will define stock shape and size.

There are several options such as: Fixed/Relative Size Box, Fixed/Relative Size Cylinder, Fixed/Relative Size Tube, and from Solid.

1. With this Haas Fidget spinner project, we can select a Fixed Size Box.



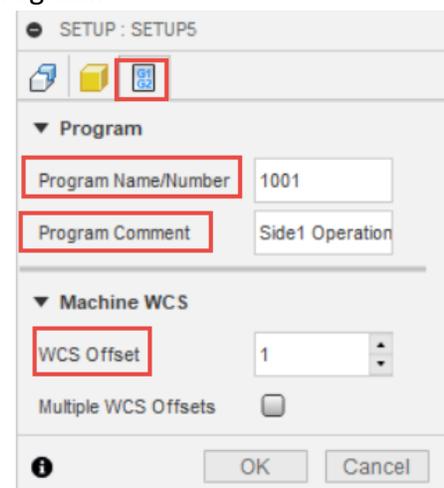
2. Enter the stock width (X) as 3in, depth (Y) as 3in., and height (Z) as 0.375”.
3. Adjust the Z Height Model position to be Offset from bottom (-Z); and offset value set to zero. This adjustment will position the bottom of the stock onto the model bottom face which is aligned with fixture face.

Step 5: Establish Post Processing Outputs

When defining a new setup, you will see three specific tabs; Setup, Stock, and Post Process.

In the Post Process Tab you will establish some standard post processing output credentials such as program number, program comment, and work coordinate (WCS) offset value to be used when post processing a single or group of toolpaths.

1. With program number parameter, enter 1001.
2. With program comment we can reference our machining sequence of events and label them as Side1 Operations.
3. It becomes important when post processing g-code that the zero position of our program can easily be identified with in the machining envelope on the machine. Most CNC machines allow for pre-setting specific machine coordinates and saving those positions of where our parts are located in the controller known as Work Offsets.
4. Common Work Offset commands used in standard G-code NC programs can range from G54 through G59.
5. The WCS Offset parameter allows you to specify a number 1-6.
6. Enter 1 into this parameter field as this will force the post processor to output a G54 command to reference the zero position of your NC program.



7. Click Ok to complete the process of creating a new manufacturing setup.
8. You will notice that a new setup has been created. Left-click twice on the setup to rename it to Setup 1 Operations.
9. Click the **Save** icon  to save your current state of design and manufacturing

Step 5: Create Secondary CAM Setup

Considering we are going to machine both side 1 operations and side 2 operations all on one fixture base component; it can be helpful to create a secondary CAM Setup to assign a new program zero-point position with current stock representation.

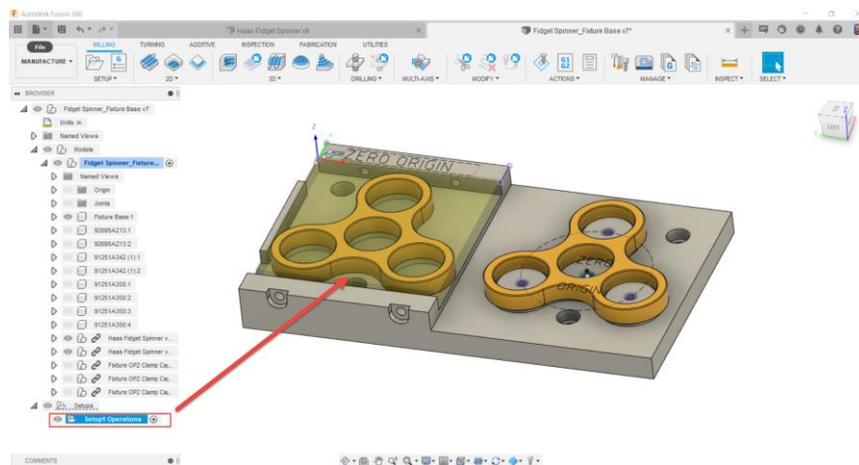
Fusion 360 supports Automatic In-Process Stock Generation that calculates a visual representation of the stock, known as in-process stock, which enables you to preview the effect that each toolpath has on the stock. When you simulate a toolpath that has in-process stock calculated, the simulation begins with an accurate representation of the stock.



To turn the display of in-process stock on and off, use the Stock Visibility button in the Navigation bar at the bottom-center of the canvas.

You can also define stock conditions within the CAM Setup by selecting another solid body model which could be another body or component that is present within your manufacturing assembly document.

1. Activate your current open document named **Haas Fidget Spinner** by selecting its tab at the top of screen.
2. Note that when we created CAM Setup1 you defined stock with a fixed size box that locked onto the current model position of the Fidget Spinner sitting on the fixture for side 1 machining operations.
3. Considering the second instance of the Fidget Spinner, for side 2 machining operations, is located on the fixture in a different location; you can create a new body that represents the current state of the stock so that it appears aligned correctly for side 2 operations.

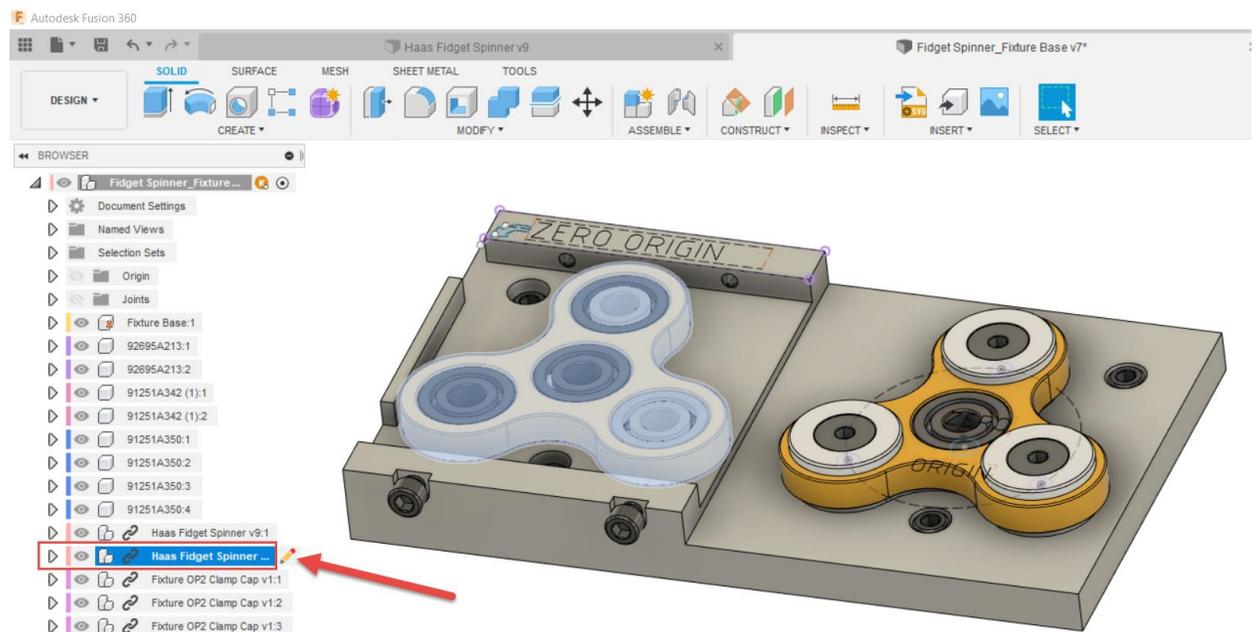


4. To achieve this, you can make use of the Edit in Place feature.

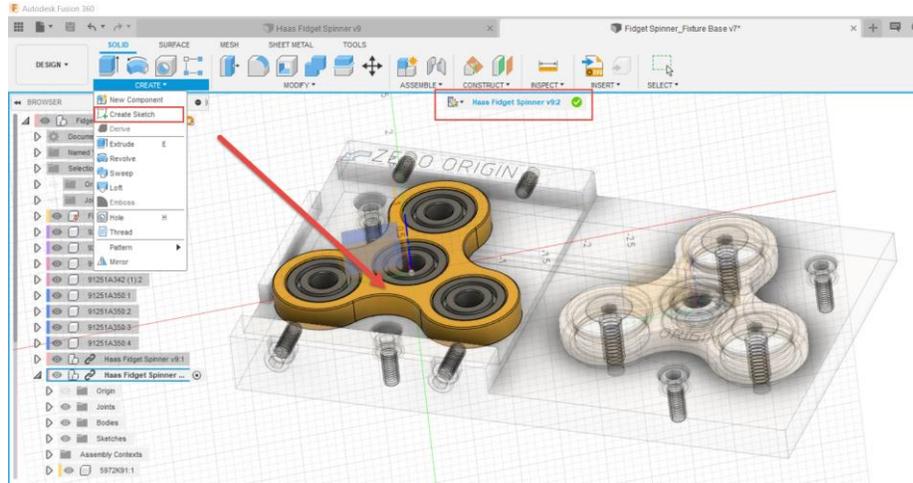
In Fusion 360, Edit In Place lets you activate and edit an external component in the context of the assembly without leaving the parent design. When you activate Edit In Place, you can see the component in its real location within the assembly as you edit it.

An Assembly Context is the connection between a parent design and an external component at a specific point in time. It holds positional information related to the assembly. It is where geometry is derived to maintain the association to the parent design.

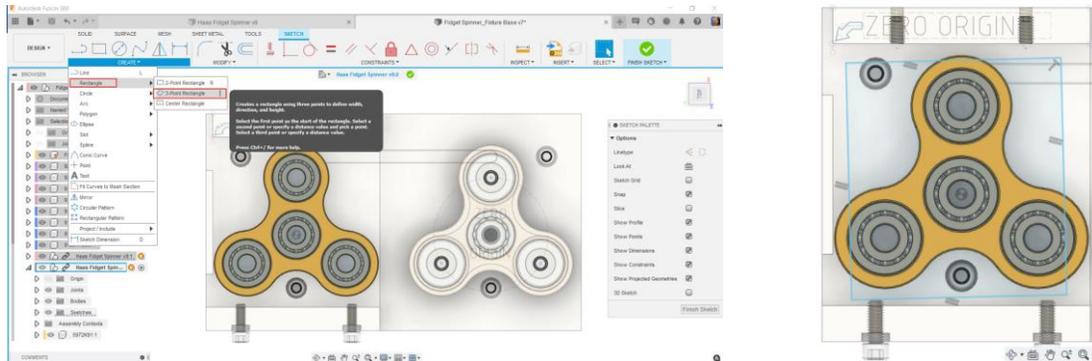
5. In your browser, highlight and select the Haas Fidget Spinner component that appears to be on the left side of the fixture and activate the Edit in Place command.



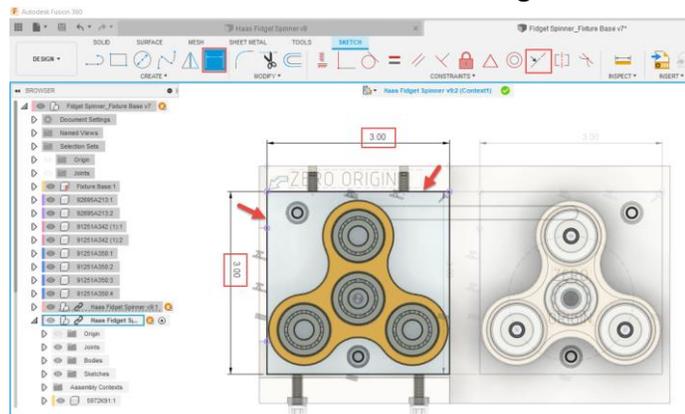
6. Once activated, note that other assembly components become transparent and only the Fidget Spinner component is active.
7. Create a new Sketch and select the top face of the Fidget Spinner component.



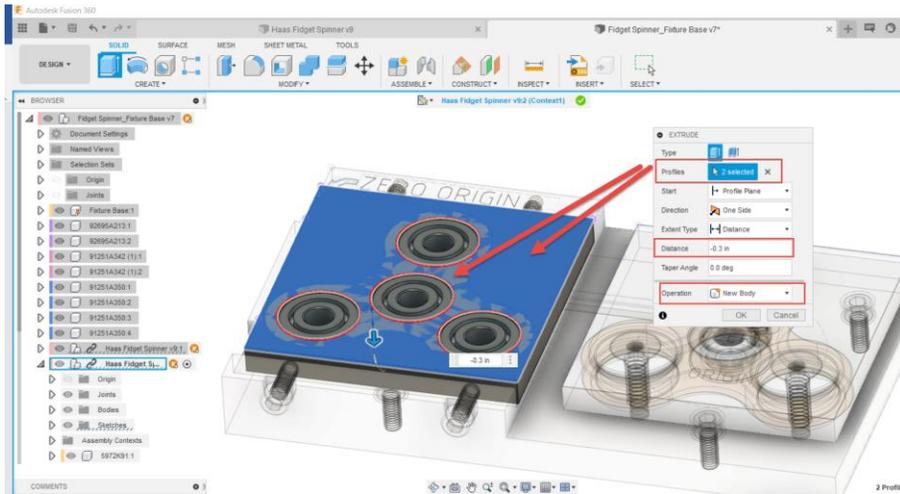
8. Your perspective view may appear to be at an angle, so use the view cube to reorientate your view perspective to be normal to the top view.
9. With in this new sketch, create a 3-point rectangle.



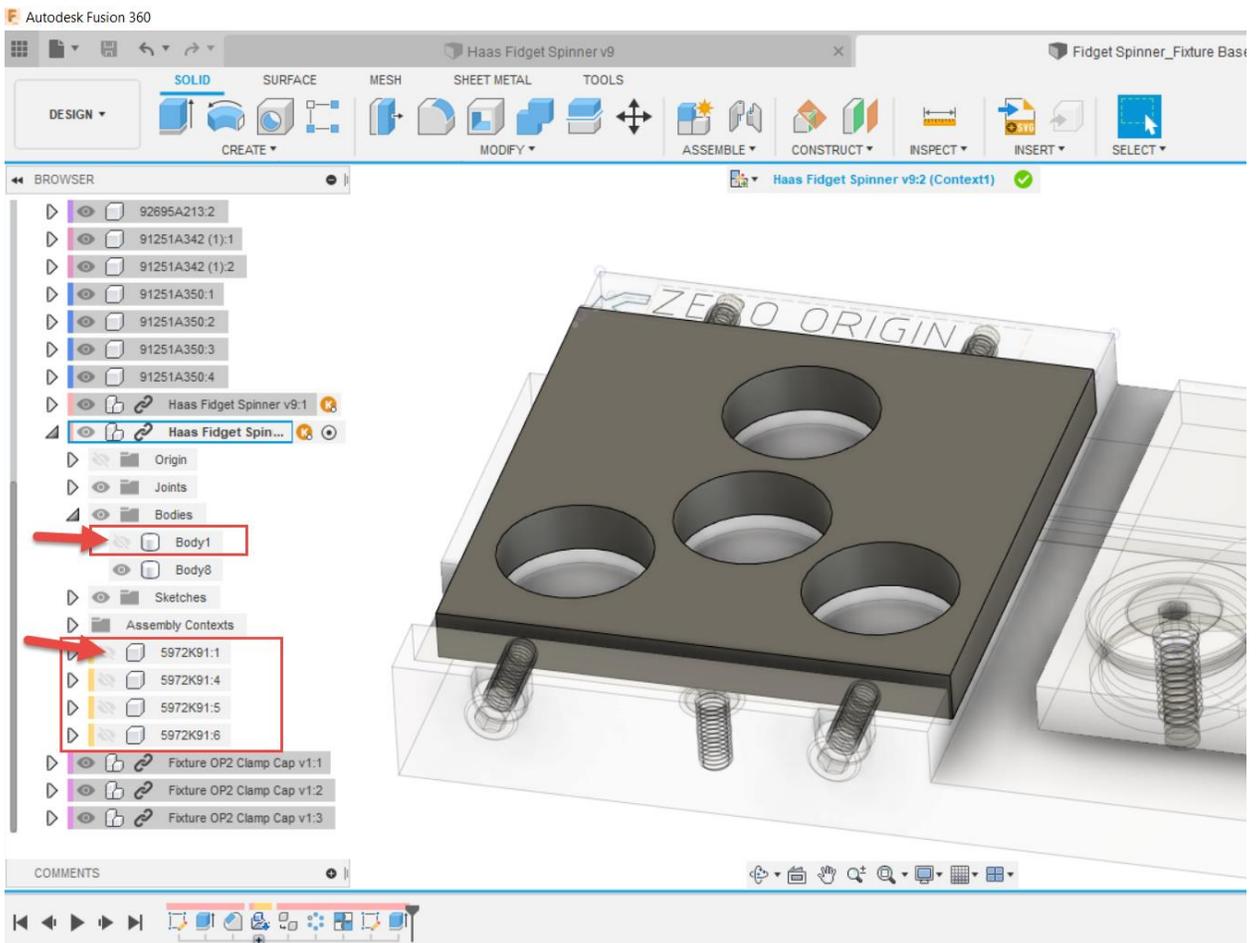
10. Stretch out a 3-point rectangle, don't worry if the rectangle shows incorrect alignment, as you will define constraints to position accurately.
11. Activate the Collinear constraint and align the upper Horizontal blue line with the edge of the fixture and the vertical rectangle line with the edge of the fixture.
12. Activate the Dimension command and dimension the length of both lines to be 3.00".



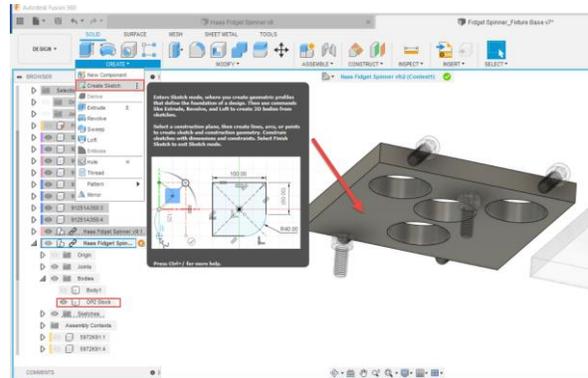
13. Activate the Extrude command, select both profiles shown below, set distance to -0.3", and ensure the Operation type is set to New body.



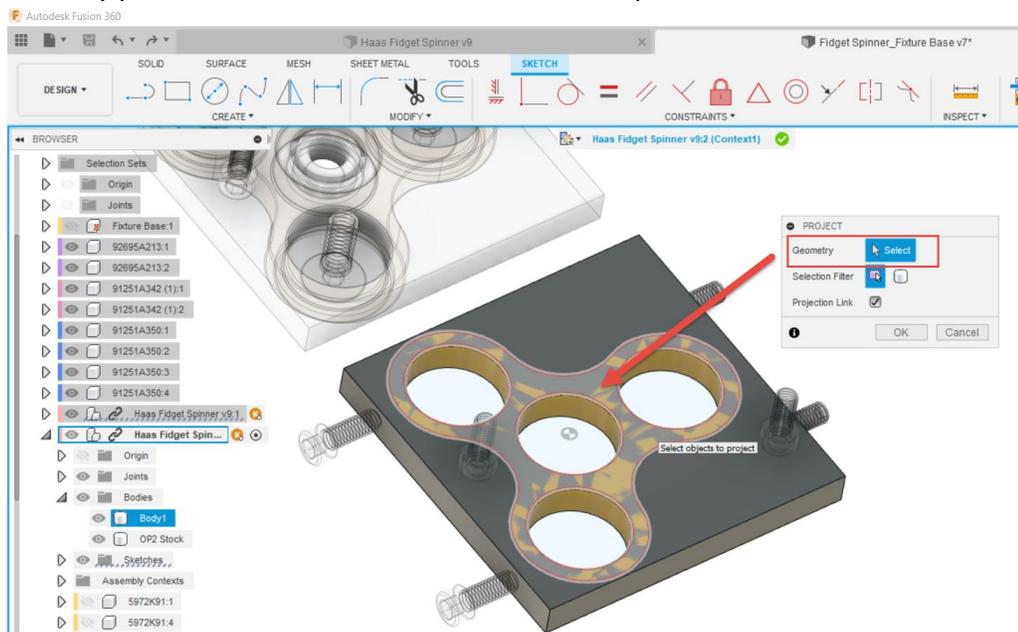
14. Choose Ok to complete the extrude feature command.
15. To see more clearly, turn off the visibility of Body 1 of the Fidget Spinner component, and turn off the visibility of all 4 bearing components inside the Assembly Contexts folder.



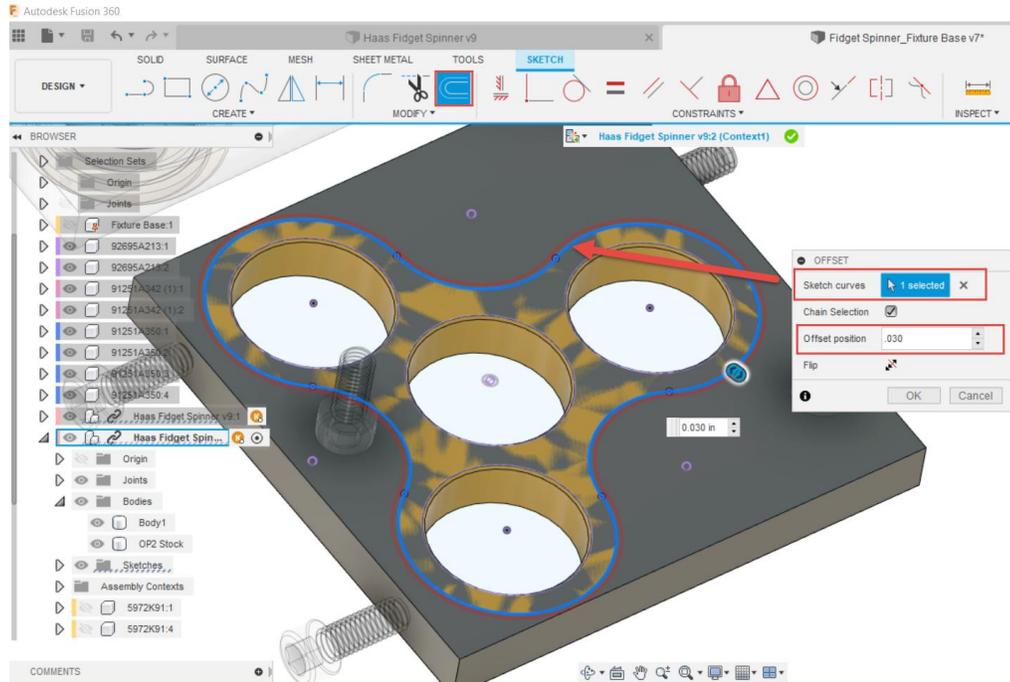
16. Rename the new body to OP2 Stock.
17. Turn off the visibility of the Fixture Base component.
18. Create a new sketch on the bottom face of the OP2 Stock body.



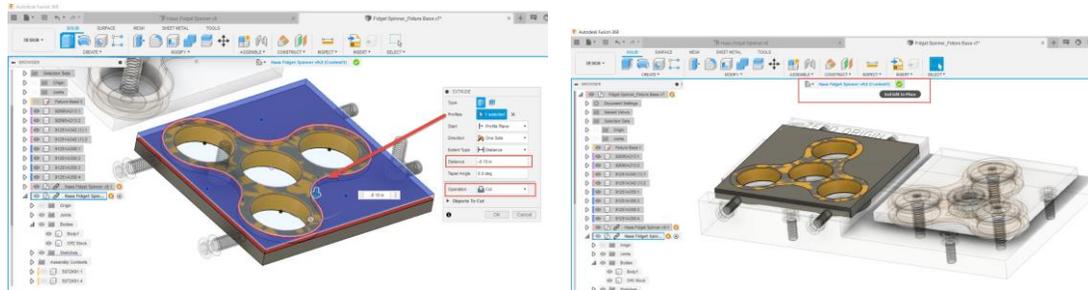
19. Turn on the visibility of Body 1 inside of the Haas Fidget Spinner component.
20. Activate the Project command under the Create drop down menu and select the fidget spinner body profile as shown below. Click OK to complete the command.



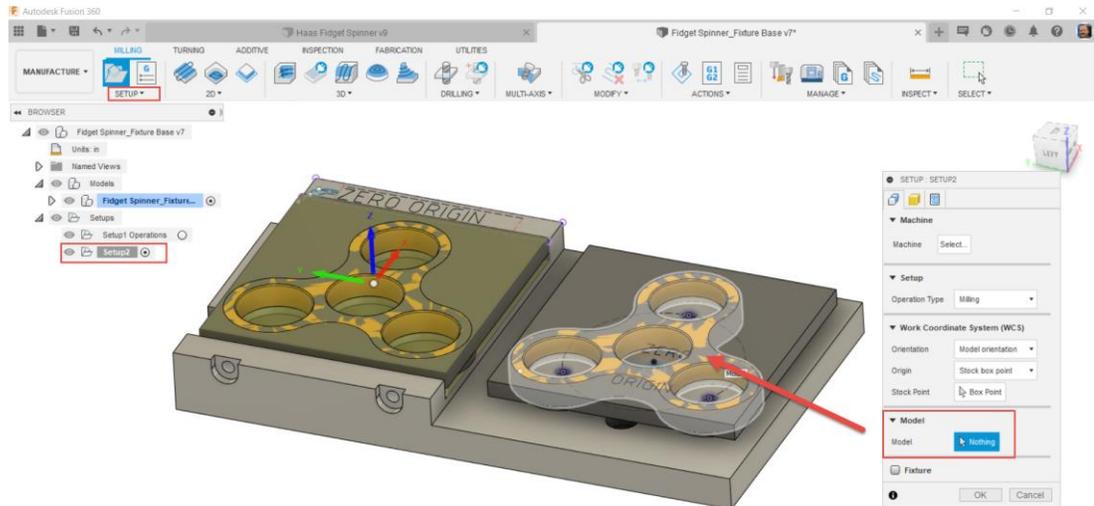
21. Activate the Offset command, select the outer edge profile for geometry and enter 0.030" for Offset position amount. Click OK to complete the command.



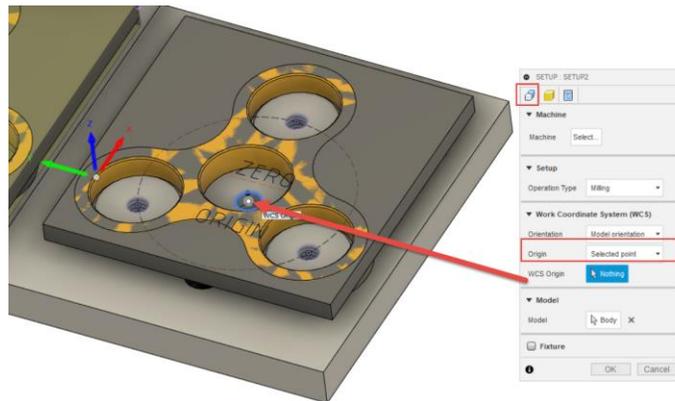
22. Activate the Extrude command and select the closed offset profile geometry as shown below. Set an extrude distance of $-0.1''$ and ensure Operation type is set to Cut. Click OK to complete the command.



23. Turn on the visibility of the Fixture Base component.
24. To end the active Edit in Place context, select the green check mark at top center area of the canvas.
25. Notice that this additional body named OP2 Stock appears in both instances of the Haas Fidget Spinner component.
26. Move from the Design workspace into the Manufacture workspace where you can assign this new OP2 Stock body effectively.
27. Create a new Manufacturing Setup, set Operation Type to Milling, then choose the second instance of the Fidget Spinner body to establish the model to be machined.

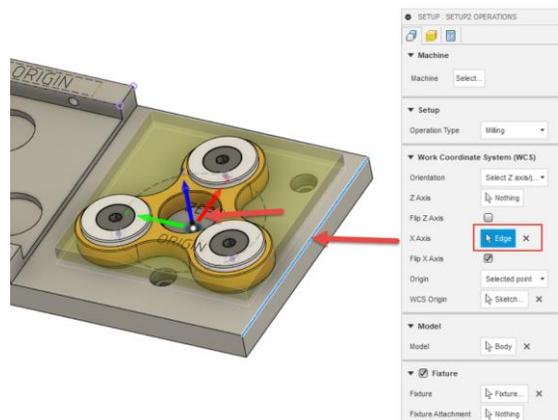


28. Change the Origin definition drop down menu to Selected Point and click Zero Origin point as shown below.



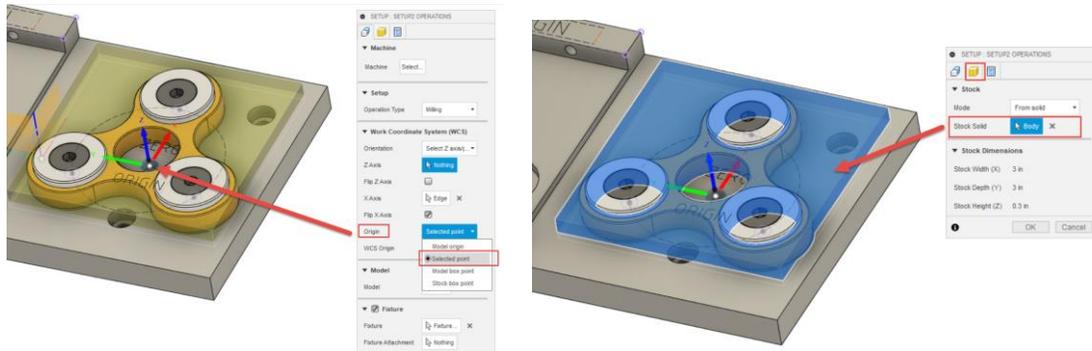
29. Notice the WCS blue, red green arrows are not aligned correctly.

30. Highlight the base of the red arrow, then select a linear edge on the fixture body that points in the direction of the red arrow.

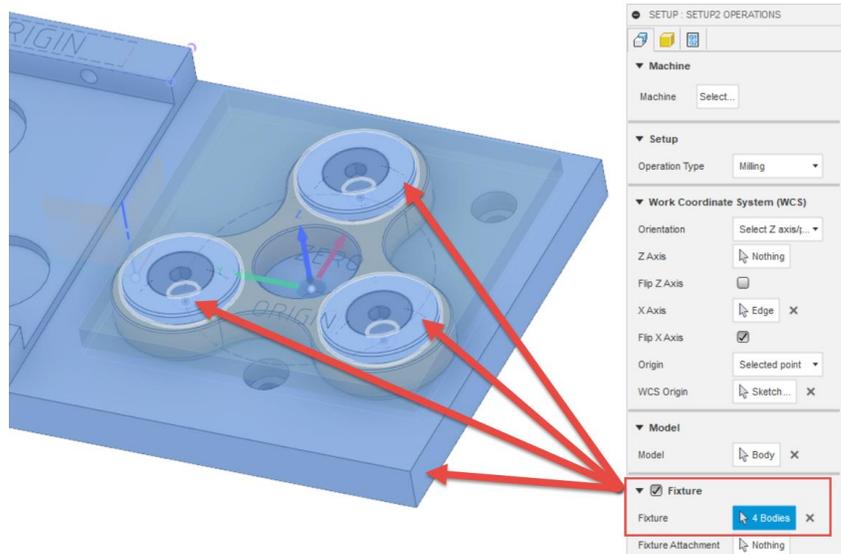


31. The red arrow head may point in the wrong direction, if so, select the head of the red arrow to reverse the direction.

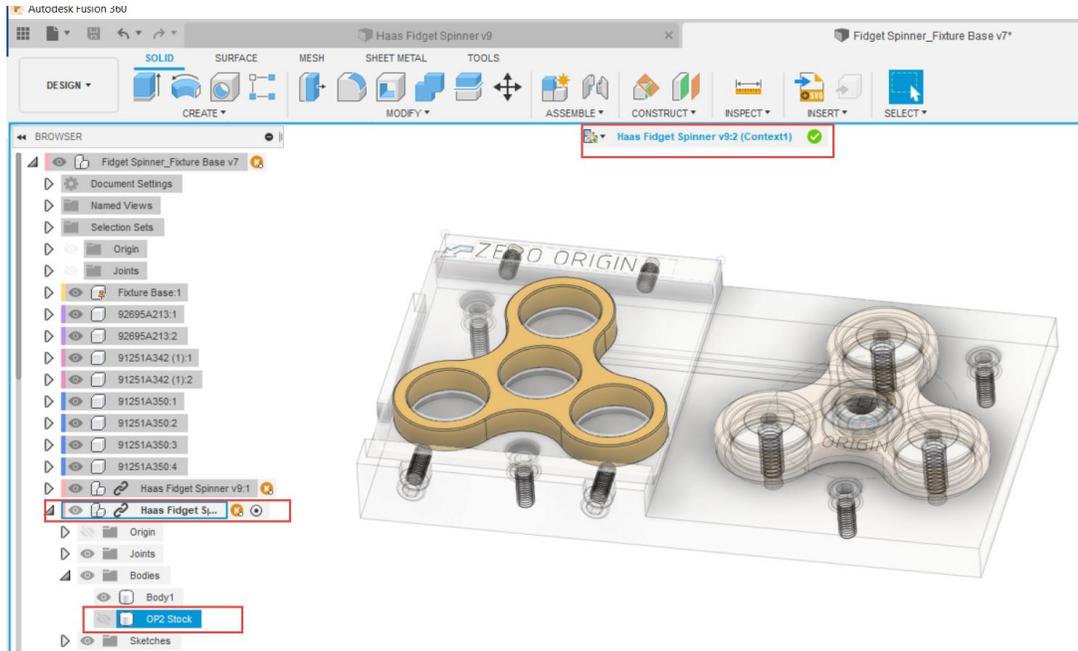
32. Ensure that your WCS is aligned correctly and positioned accurately as shown below.



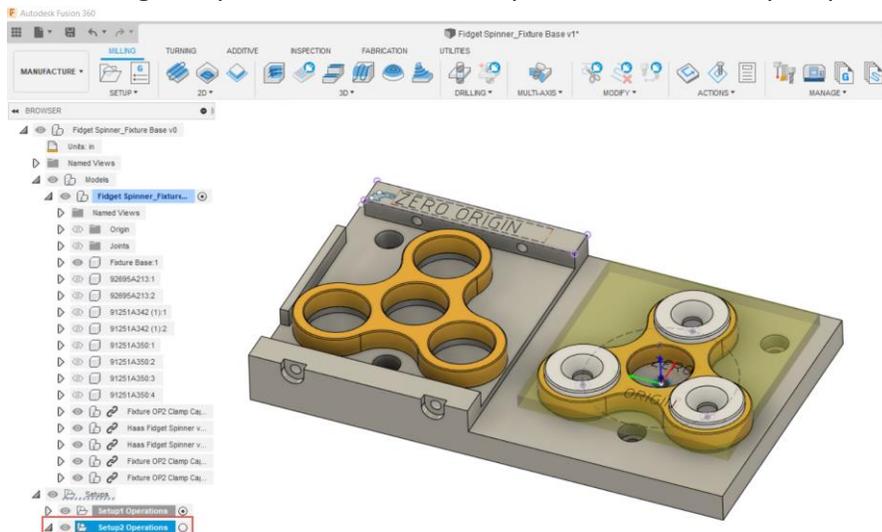
33. In the stock definition, second tab, select the From Solid option, then select the OP2 body shown above.
34. Also, activate the Fixture check box and select the Fixture Base component, hold down control key, and multi select the three Fixture OP2 Clamp Cap components.



35. In the Post Process Tab, you will establish some standard post processing output credentials for this new secondary Cam setup.
36. With program number parameter, enter 1002.
37. For the program comment, we can reference our machining sequence of events and label them as Side2 Operations.
38. Set the WCS Offset value to 2 so that your post processor will output G55 as work offset position to use on the machine for side 2 toolpath operations.
39. Click OK to complete the CAM setup.
40. Move back into the Design workspace, activate the Edit in place command on the Haas Fidget Spinner component, and turn off the visibility of the OP2 Stock body as shown below.



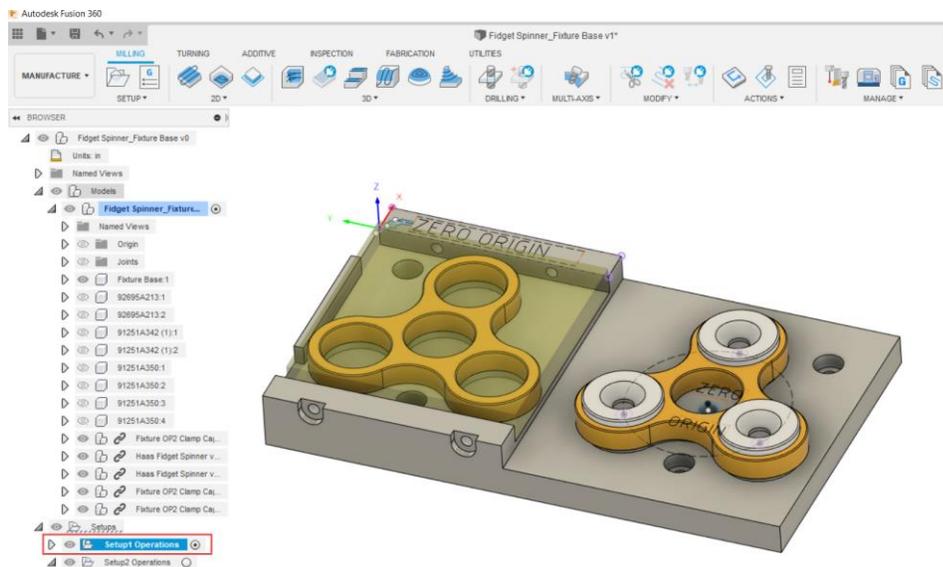
41. To end the active Edit in Place context, select the green check mark at the top center area of the canvas.
42. Move back into the Manufacturing workspace, and turn on the visibility of the Fixture Base, both instances of the Fidget Spinner, and all three Fixture OP2 Clamp Cap components.
43. In the Manufacturing Setups folder, rename Setup2 to be called Setup2 Operations.



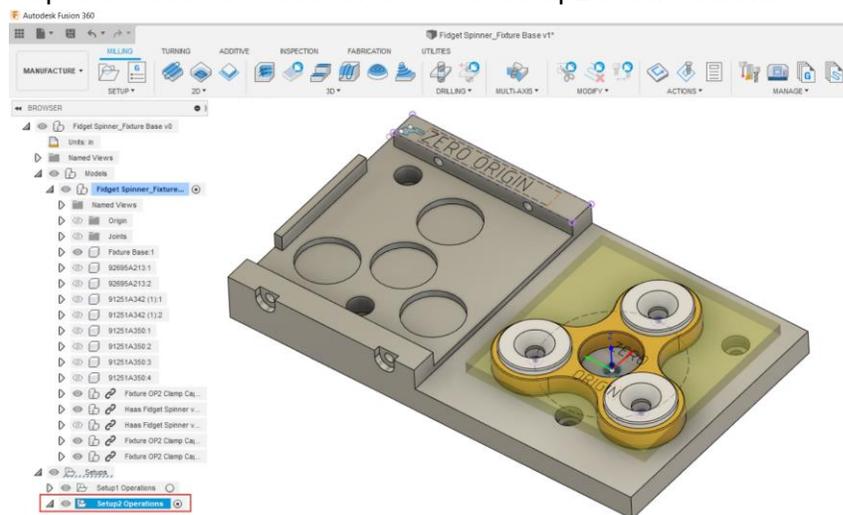
44. To synchronize the active component visibility with the active CAM setup visibility, ensure that Sync view with active setup and Sync visibility with active setup are both activated with check box located at bottom area of canvas.



45. Now, when you select the Setup1 Operations folder you will, visually, see the yellow transparent representation of stock condition for setup1 shown below.



46. Likewise, when you select the Setup2 Operations folder, you will see visually the yellow transparent representation of stock condition for setup2 shown below.



47. Click the **Save** icon  to save your design.

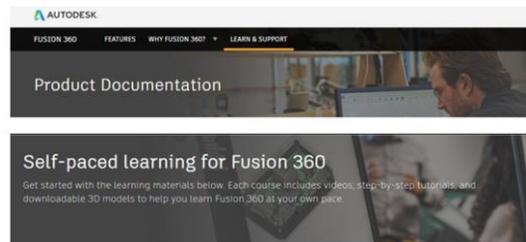
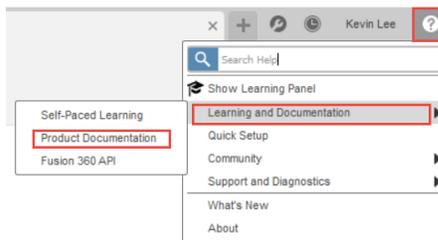
CONGRATULATIONS

You have successfully completed the guided lessons to define multiple Manufacturing CAM Setups using Autodesk Fusion 360.

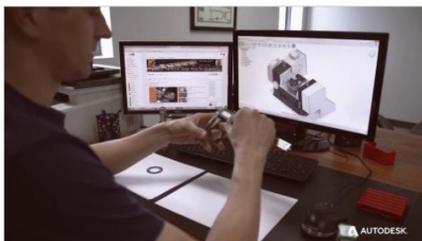


Within Fusion 360 you can directly access additional learning resources. When inside of Fusion 360 near the top right corner of your screen, select the help icon.

Here you can access further self-paced learning tutorial guides and additional product documentation.

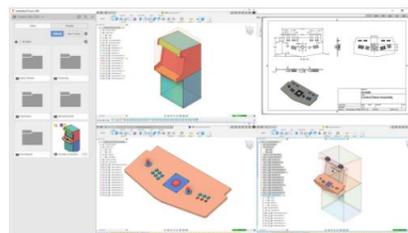


To learn more on the fundamentals for 3D modeling and collaboration in distributed designs using Fusion 360, check out the self paced learning tutorial videos.



Introduction to 3D modeling

- Video series
- 18 videos
- Beginner



Collaboration in distributed designs

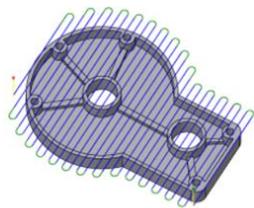
- Video series
- 3 videos
- Intermediate

LESSON 6

Manufacturing CAM Setup#1

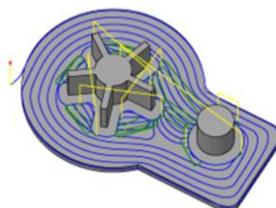
After completing this lesson, you will be able to:

- Create a Facing operation
- Understand High Speed Adaptive roughing
- Finish profile features with 2D contour
- Duplicating Operations
- Apply deburring Chamfer operations
- Verify Toolpaths with Stock Simulation



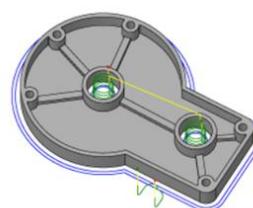
Face

Produces quick part facing to prepare raw stock for machining.



2D Adaptive

Creates an optimized roughing toolpath avoiding abrupt directional changes.



2D Contour

A finishing strategy that follows open or closed profiles that can be on different Z levels.

2D Milling is a planer machining process. It's sometimes called 2.5 axis machining because all of the cuts are limited to a 2-axis plane (normally XY) and the depth cuts are taken in the 3rd axis (normally Z). When you select the 2D pull down menu, there are a variety of toolpaths that represent this type of 2.5 axis machining.

Note that in Fusion 360 all 2D toolpath strategies require geometry (face or edge) selections to drive the toolpath.

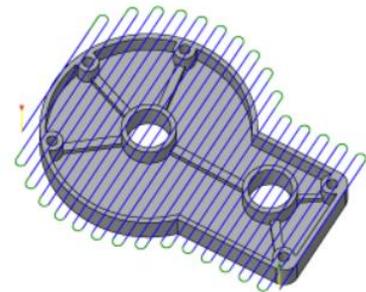
3D Milling is a multi axis simultaneous tool motion that follows the model geometry. When you select the 3D pull down menu there are several toolpaths including both roughing and finishing strategies. In Fusion 360, 3D toolpaths get applied to model body geometry allowing for tool boundary control, slope containment and check surfaces to avoid.

You will find some of the parameters will be similar between toolpath 2d and 3d strategies. Selecting a tool from the library, editing tool parameters, feed rates, and setting the machining heights will be common to all toolpaths.

Combining both 2D and 3D toolpath strategies to machine your part allows for seamless CAD revision updates, easy geometry selections, and automatic toolpath calculation workflows.

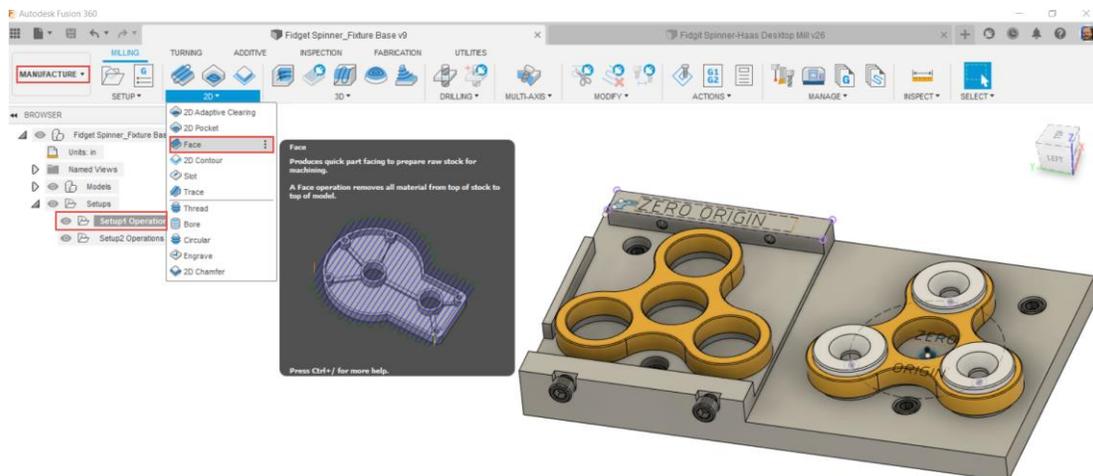
Step 1: Create a Facing Operation

The Face toolpath is often the first operation performed on raw stock material to prepare it for further machining. Typically, the depth of material to be removed is the distance between the top of the stock and the top of the model.

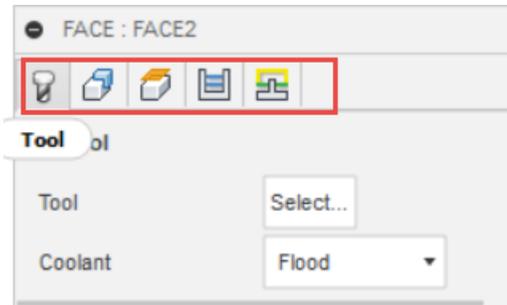


The Face toolpath helps ensure flatness on a surface so it can sit flat on a vise or fixture for accurate machining. It can also be used for clearing flat areas in general.

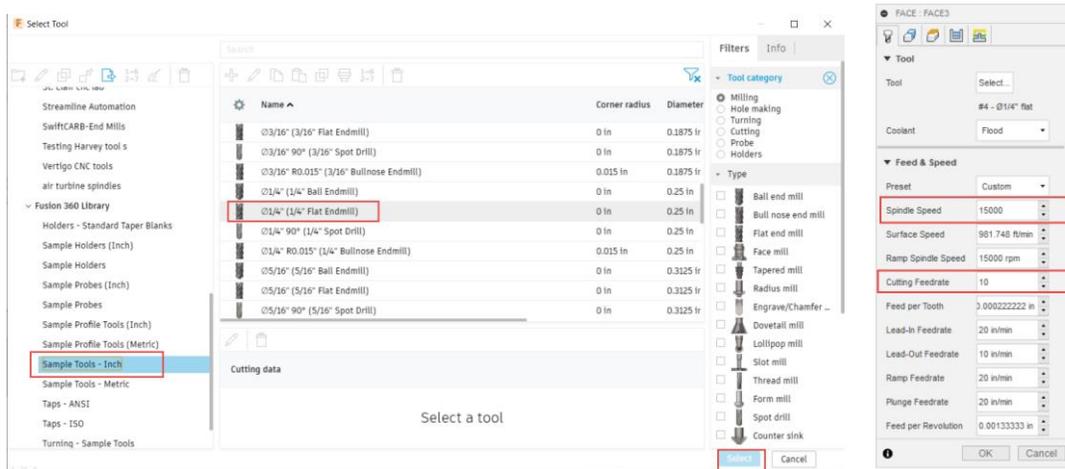
1. With your currently open active document, **Fidget Spinner_Fixture Base**, ensure you are in the Manufacture workspace.
2. Be sure that **CAM Setup1 Operations** folder is active and select the Face operation in the 2D drop-down menu.



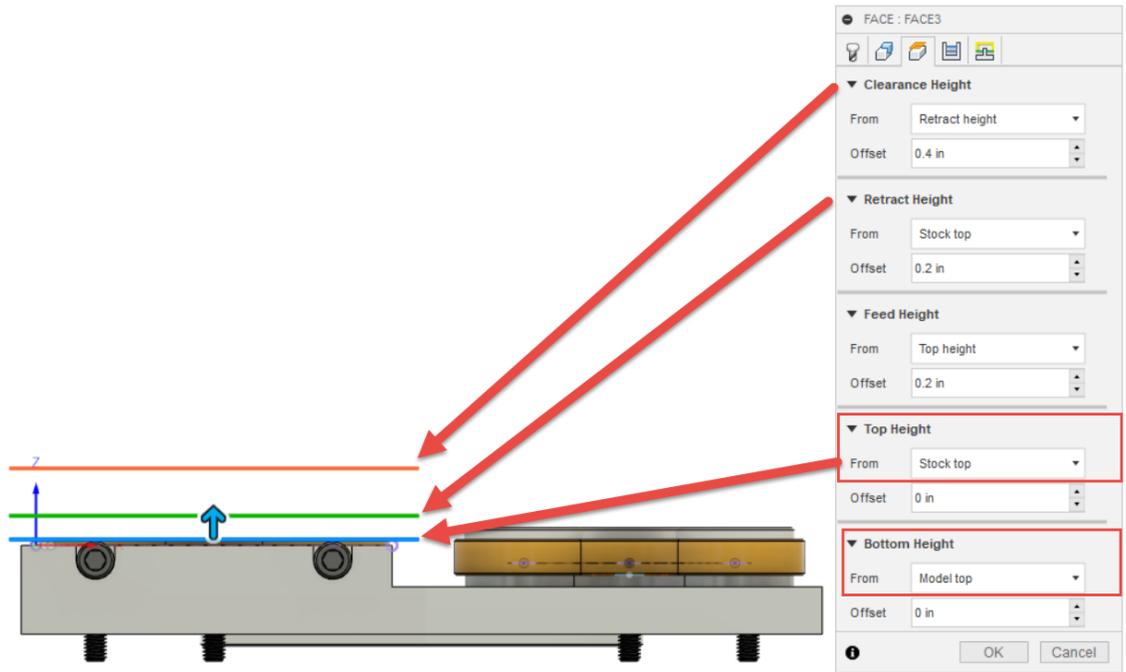
3. For the most part, every toolpath strategy will open with having access to 5 tabs of information; Tool, Geometry, Heights, Passes, and Linking.



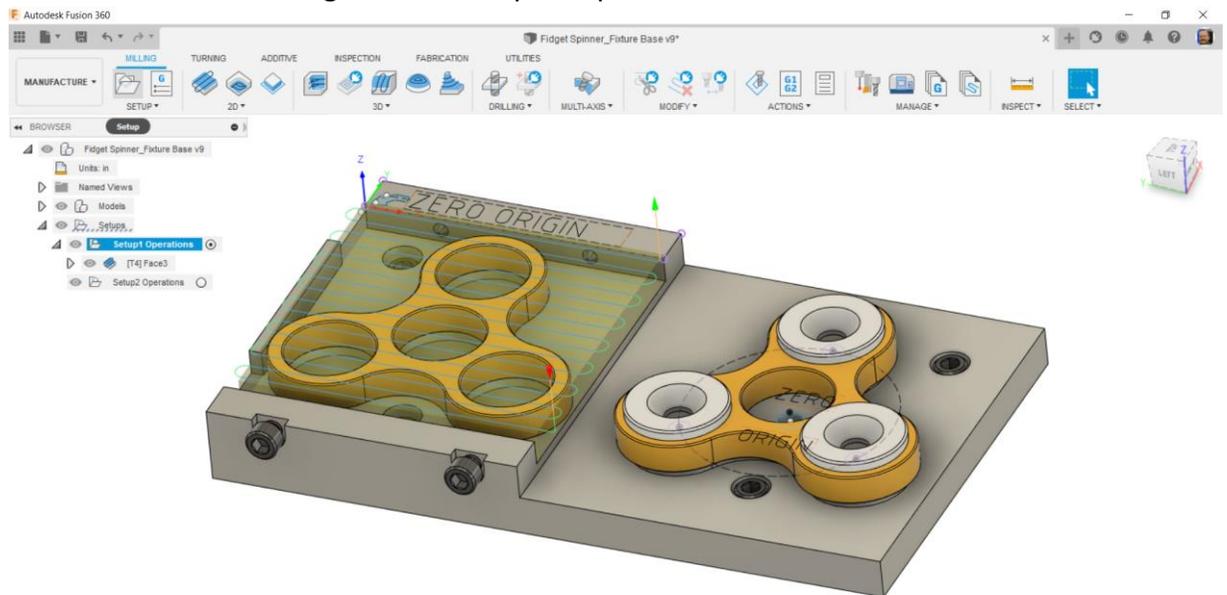
4. In the Tooling Tab, click Select, to find a tool.
5. The tool selection dialogue box opens allowing you to select a tool from sample tutorial, local, or cloud libraries.
6. On the left, scroll down the list of tool libraries and in the Sample Tools-Inch library, select tool , the 1/4" Flat Endmill as shown below.



7. Adjust the Spindle Speed RPM to 15000 and change the Cutting Speed to 10 inches/min. as shown above for this toolpath operation.
8. In the Heights Tab, you will notice that the Top Height of the toolpath will begin at the Stock top reference, and the Bottom Height of the toolpath will machine down to the Model Top surface. Remember, this relationship between the model and stock definition was established when creating the CAM setup.



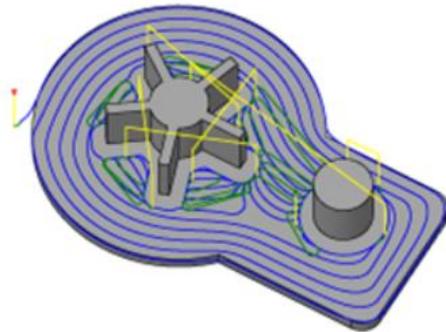
9. Click OK to finish creating the Face toolpath operation.



10. Click the **Save** icon  to save your design.

Step 2: Understanding High Speed Adaptive Roughing

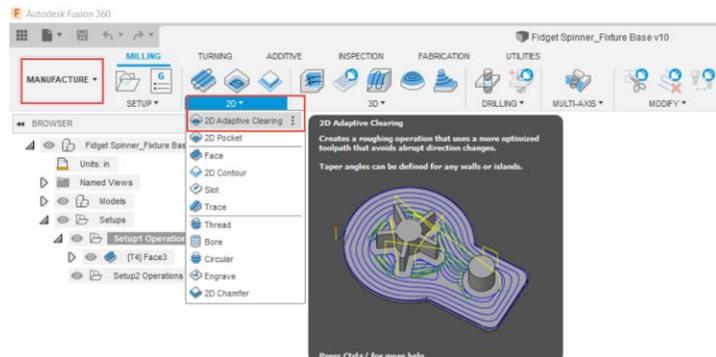
The 2D Adaptive Clearing is a roughing toolpath strategy that avoids abrupt directional changes. You can clear a cavity, open pocket, or the area around a boss by selecting the edges, sketches, or solid faces as the machining area.



Sudden changes in how much the tool is engaged while cutting material around it can cause tool breakage. To avoid sudden changes in tool engagement, 2D Adaptive Clearing maintains a constant engagement between the tool and its surrounding material. The Optimal Load parameter sets the level of engagement and can be adjusted depending on the tool manufacturer's recommendations.

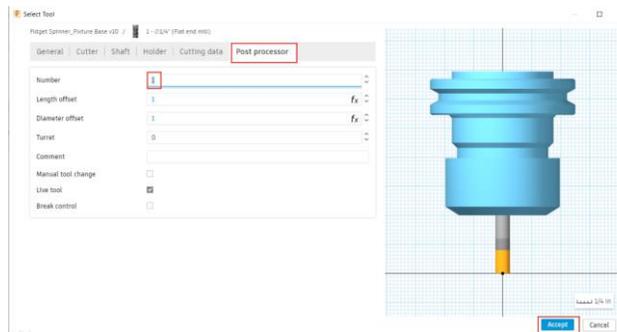
The 2D Adaptive Clearing toolpath is preferred for roughing simple pockets and other simple geometry whereas 3D Adaptive Clearing offers more control and can be used for more complex geometry.

1. On the Manufacture workspace toolbar, click Milling > 2D > 2D Adaptive Clearing to create a new toolpath.

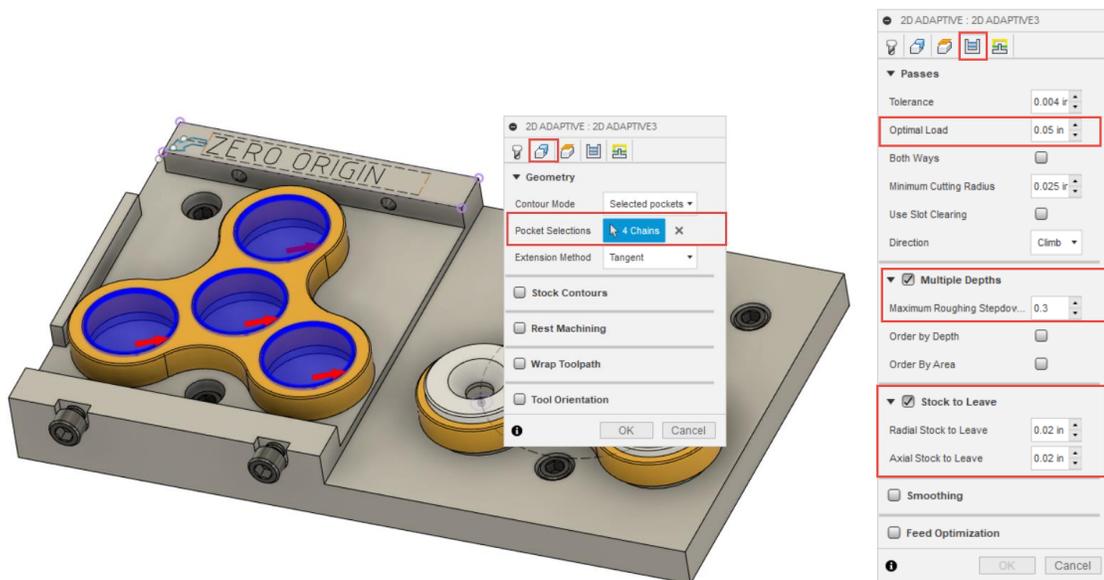


2. Notice this new toolpath automatically uses tool 4, the tool from the previous operation.
3. In the tooling tab, choose the tool Select button.
4. Right click on tool 4, and select Edit tool.

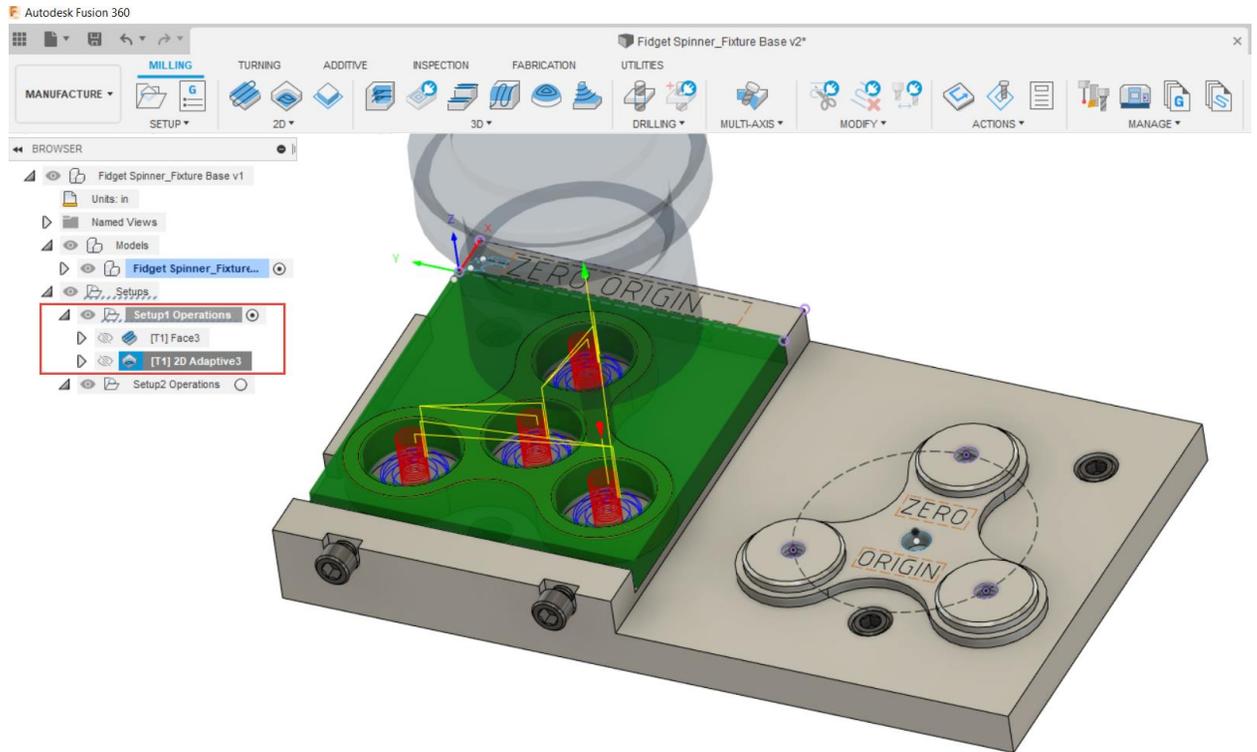
- In the Post Processor tab, of the Tool Edit dialogue box, change the tool number from 4 to 1, and Enter. Notice that the tool Number, Length Offset, and Diameter Offset are now matching.



- In the Cutting data tab, adjust the Spindle Speed RPM to 15000 and change the Cutting Speed to 10 inches/min. This time you are editing the feeds and speeds of the tool so that every time you select this tool from your document library, the tool will use these feeds and speeds.
- Choose Accept to complete the tool edits.
- You will get a notification asking if you want to update the listed operations to use the new feeds and speeds, select Yes.
- In the Geometry Tab, select the 4 pocket inner bore edges, the lower rails of the bore chamfers.



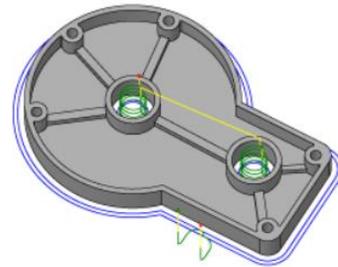
- In the Heights Tab, set Bottom Height from Model Bottom and apply a -0.010" offset.
- Adjust the Optimal Load to 0.1" in the Passes tab, activate Multiple Depths and set Maximum Roughing Stepdown to 0.15". Ensure Stock to Leave parameter is turned on, leaving 0.020" Radial Stock to Leave, and leave 0 on Axial Stock to Leave.
- Click OK to finish creating the 2D Adaptive toolpath operation.



13. Click the **Save** icon  to save your design.

Step 3: Finish Features with 2D Contour

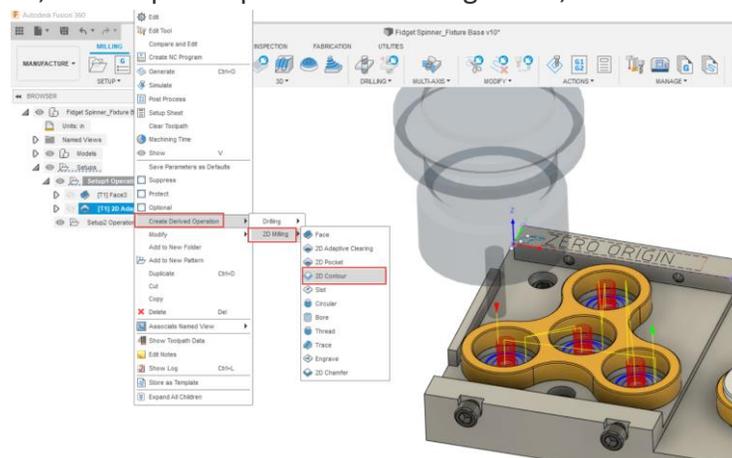
The 2D Contour creates a toolpath based on a contour selection, which is an outline of a part or feature profile. You can select a sketch or an edge on the part for the contour selection. The contour selections can be open chains and can be on the inside or outside a part. In 2D contour, the cutting motion always takes place on a 2D plane.



2D Contour is commonly used as a finishing toolpath allowing for tool radius compensation. You can use multiple finishing passes to avoid cutting the full width of the cutter. You can also machine tabs to help hold a part in place, useful when cutting on a sheet of material.

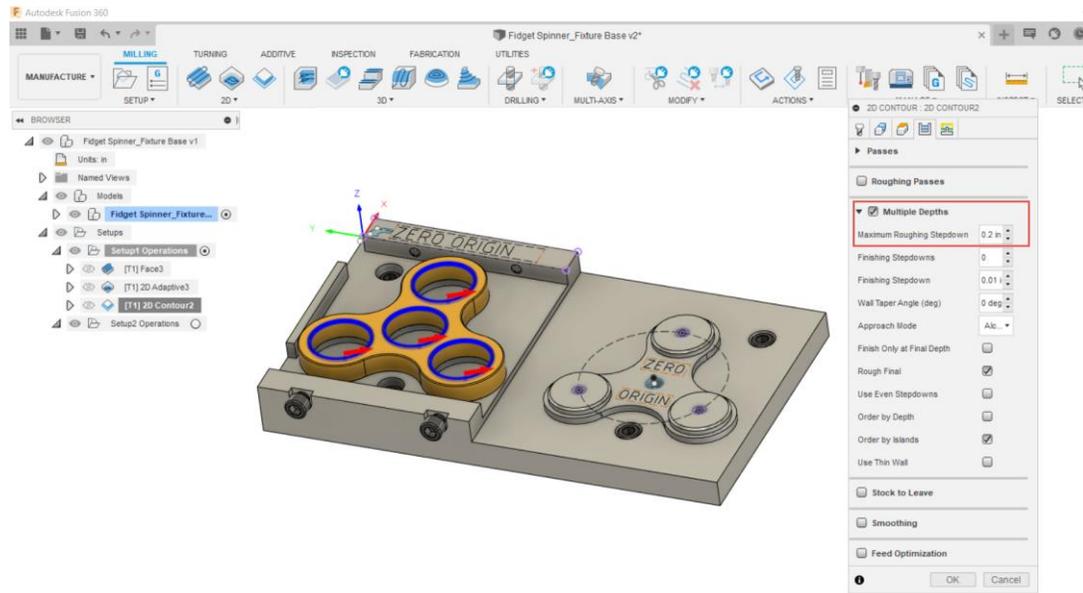
The 2D contour toolpath is ideal for straight walls whereas 3D Contour is useful for tapered walls and organic surfaces.

1. Under the 2D toolpath drop down menu, you can choose 2D Contour.
2. However, in this instance, considering we want to finish the same geometry profile selections that were used for 2D Adaptive, you can Derive a new operation from an existing toolpath.
3. Create a derived operation by right clicking on your 2D Adaptive toolpath, select Create Derived Operation, then expand open the 2D Milling menu, choose 2D Contour.



Note that Derived toolpaths lets you copy the parameters of an existing toolpath operation and use in it a different type of toolpath.

4. A new 2D contour strategy is created that contains as much information as possible from the previous 2d Adaptive operation such as tooling information, geometry selections, and height settings.
5. In the Passes tab, adjust Maximum Roughing Stepdown to 0.200", and ensure Stock to Leave is disabled.

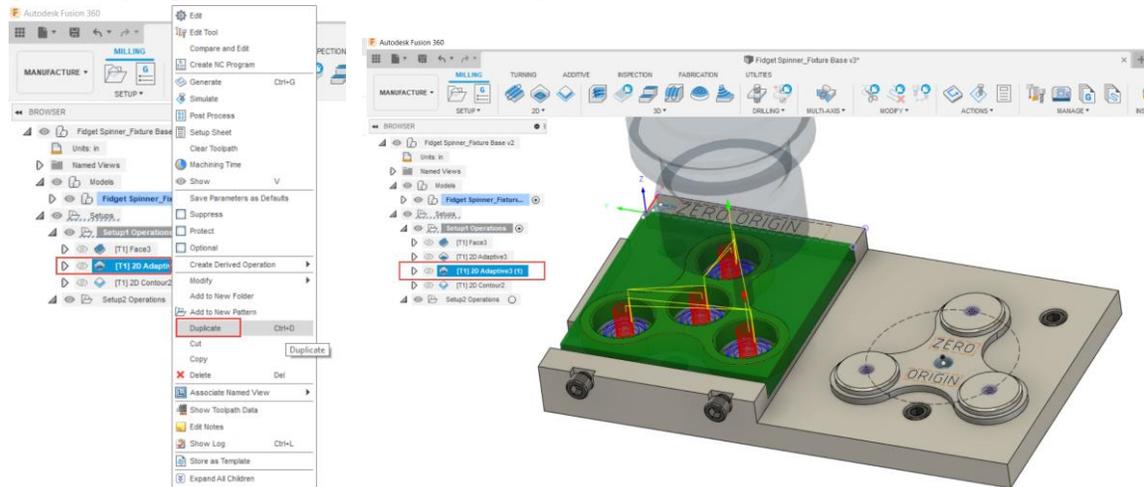


6. It is always safe to ensure the minimum distance between the tool and the part surfaces during retract moves by setting the Safe Distance to 0.1" in the Linking tab; then click OK to finish the 2d contour toolpath settings.
7. Click the **Save** icon  to save your design.

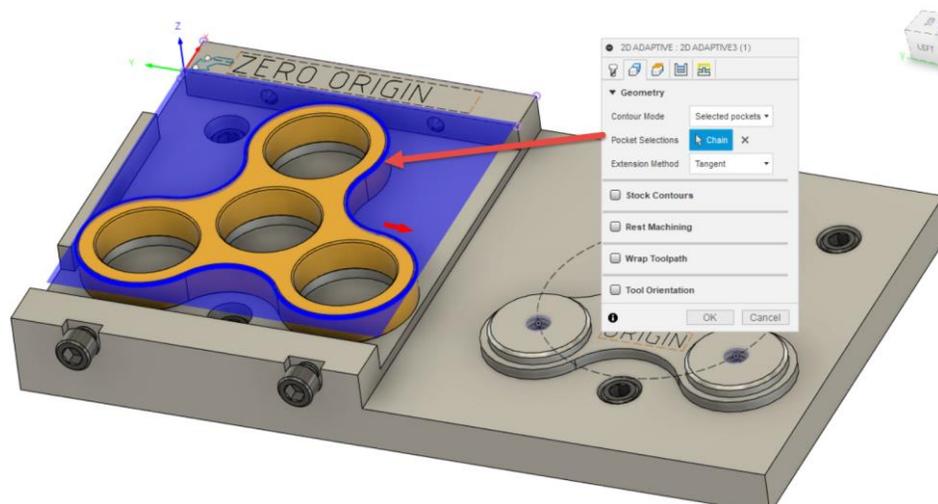
Step 4: Duplicating Operations

To save time when programming multiple toolpaths, you can reuse most of the settings of a toolpath by using Copy, Duplicate, or Created Derived Operations as completed in step 2.

1. Your previously created 2D Adaptive toolpath can be reused but reapplied to new geometry, right click on it and choose Duplicate.

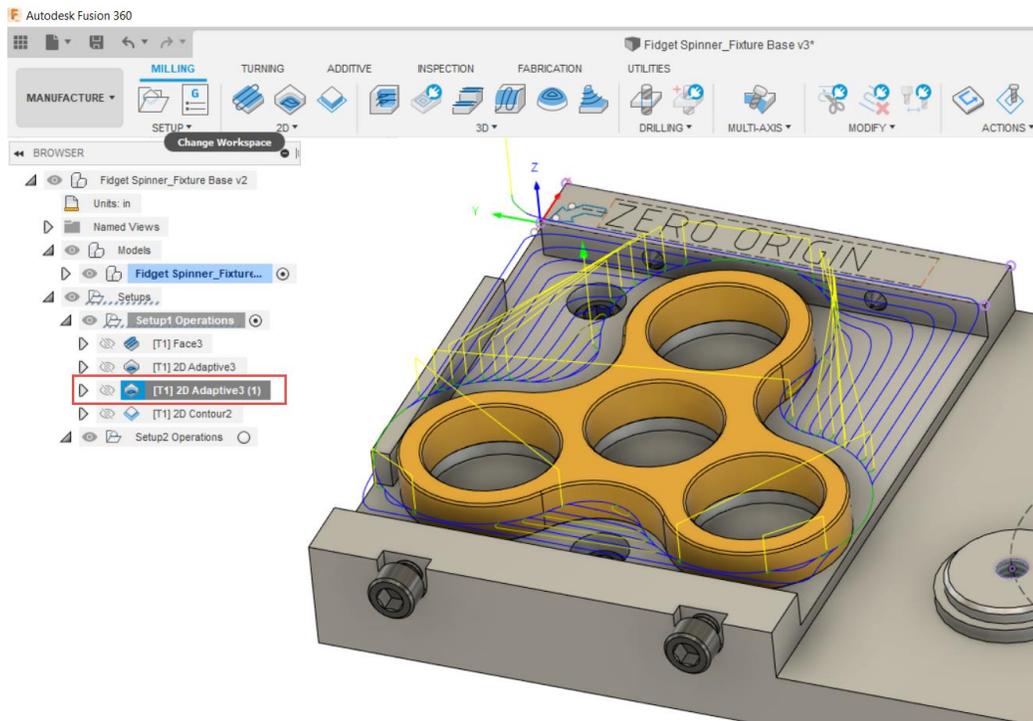


2. The newly created duplicated operation appears as a direct copy under your original 2D Adaptive toolpath.
3. Select the duplicate operation, right-click and edit, and reselect new geometry to drive the toolpath.
4. Select the upper outer edge profile of the Fidget Spinner to confirm a new pocket selection.



5. In the Heights tab, set Top Height of toolpath to Model Top, and set Bottom Height of toolpath to Model top as well but with a -0.040" offset.

- Deactivate the Multiple Depths check box in the Passes tab and ensure Stock to leave has 0.020" set for Radial stock to leave.
- Choose OK to confirm adjustments.



- This 2D adaptive operation is mainly used to remove the extra stock material from around the outside profile so that you can apply a chamfer toolpath operation to debur all machined edges before it gets flipped upside down to complete side 2 machining operations.

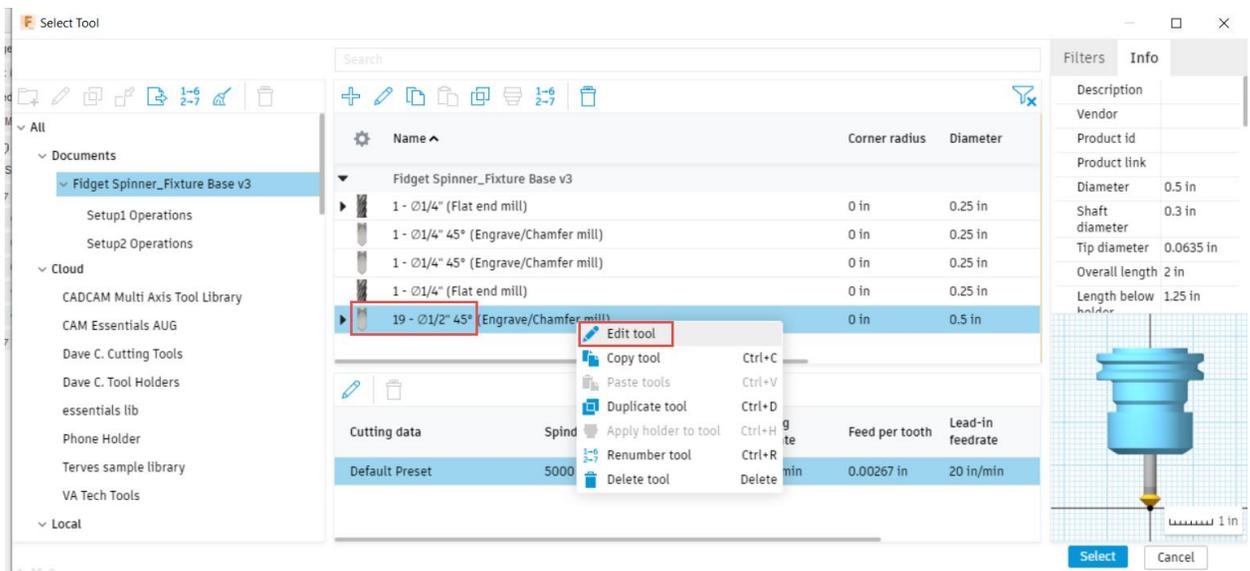
- Click the **Save** icon  to save your design.

Step 5: Deburring with Chamfer Toolpaths

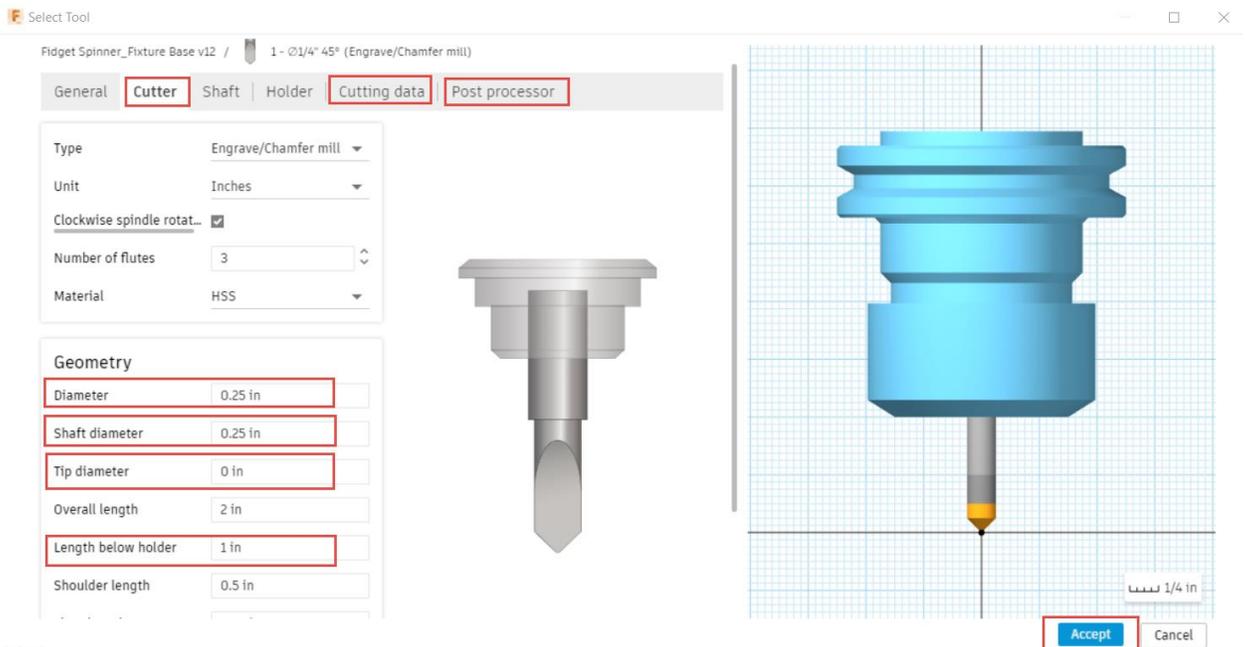
2D Chamfering lets you mill beveled edges on a part. There are two ways to program a chamfer, depending on whether the CAD model already has chamfered features or not. In both cases, you can use the 2D Contour strategy to chamfer the model.

Alternatively, you can also use a 2D Chamfer toolpath for chamfering edges that do not have CAD modelled chamfer features.

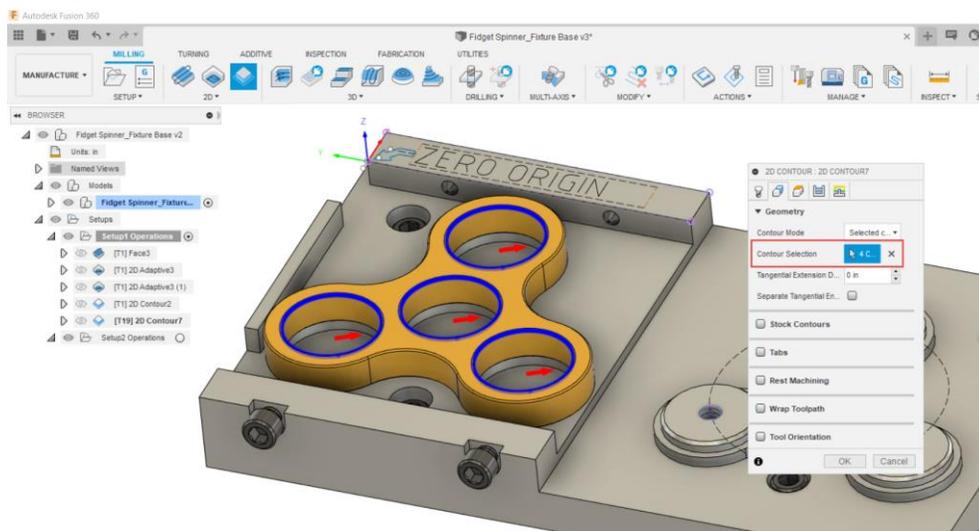
1. Under the 2D toolpath drop down menu, create a new 2D Contour toolpath strategy.
2. In the Tooling tab, select a new library tool.
3. In the tool library filter selection, enable the Engrave/Chamfer tool filter and ensure the selection filter is searching all of the Fusion 360 libraries.
4. In the Tutorial – Inch library, scroll list to tool 19, the ½” 45 degree engrave/chamfer tool and choose Select. Now that a library tool has been selected for this toolpath operation, in the Tooling Tab of this operation, choose select again, then right click on tool 19, and choose Edit tool.



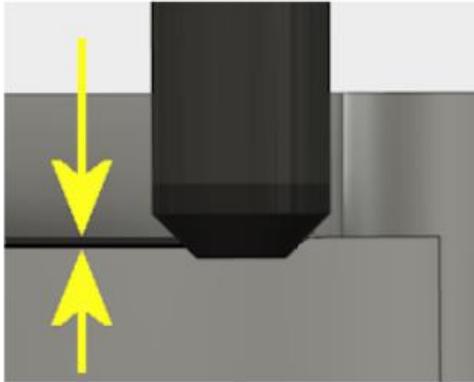
5. Edit the tool in the Cutter tab, adjust the diameter to 0.25”, shaft diameter to 0.25”, tip diameter to 0, and 1.0” for the tool Length below holder.
6. In the Cutting Data tab, set both spindle speed and ramp spindle speed to 15000 RPM, and 10 in/min feedrate.



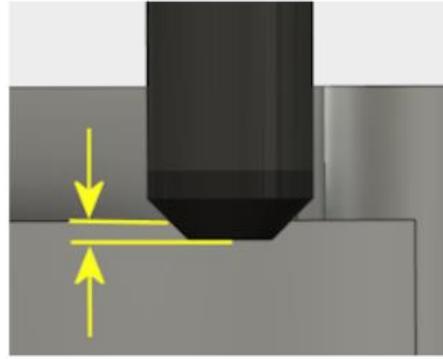
7. In the Post processor tab, set the tool number to 1. In most cases with CNC machines that have tool changers, you would have a different tool number for each different tool used in toolpaths.
8. Choose Accept to finish tool editing parameters.
9. In the Geometry tab, select the lower chamfer edges of the 4 bore features on the model as shown below.



10. Ensure that the Bottom Height parameter in the Heights tab is set to Selected contour.
11. Notice in the passes tab, since you have a chamfer tool selected, there now is a Chamfer settings parameter that lets you define Chamfer Width and Chamfer Tip Offset amount.



Chamfer Width

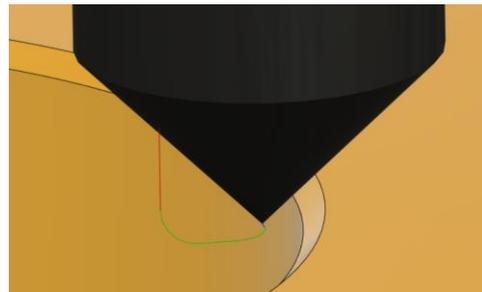


Chamfer Tip Offset
Tool Tip Clearance Extension

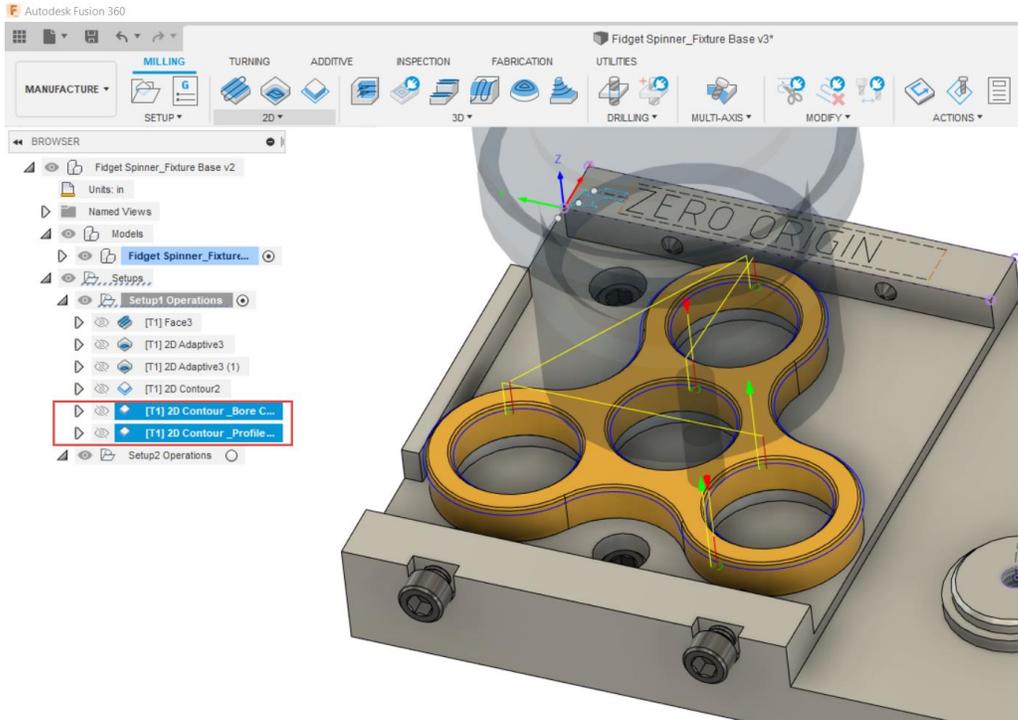
Chamfer Width sets the width of the chamfer for edges that are not already chamfered on the model. For Chamfered edges, this is an additional offset; similar to using negative stock to leave.

Chamfer Tip Offset adds to the toolpath depth, while keeping the tool in contact with the selected edge by adjusting the toolpath radial offset, essentially cutting the chamfer with the top of the cutting flutes on the chamfer tool.

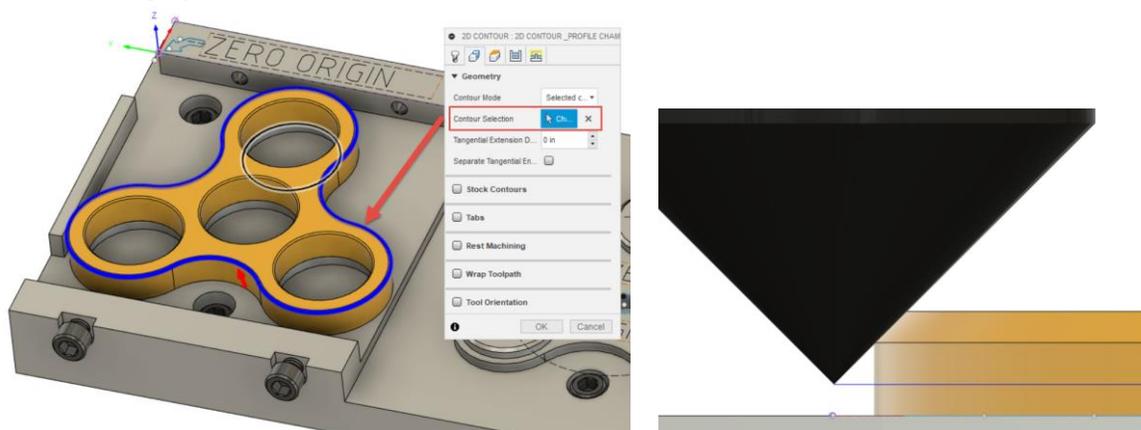
12. Considering the model already has modeled chamfers and that we selected the lower edges of the chamfer feature, leave Chamfer Width set to 0.
13. Adjust the Chamfer Tip Offset to 0.020" to machine the chamfer further up the cutting flutes to avoid machining with the tip of the tool.



14. Select Ok to complete the contour toolpath.
15. Rename this operation to be called Bore Chamfers.
16. Duplicate this Contour operation and rename the duplicate toolpath to be called Profile Chamfer.



17. Re-select the driving geometry for the 2D Contour_Profile Chamfer operation by going into the geometry tab, clear the previous (4) selections, and reselect the outer lower chamfer edge profile.



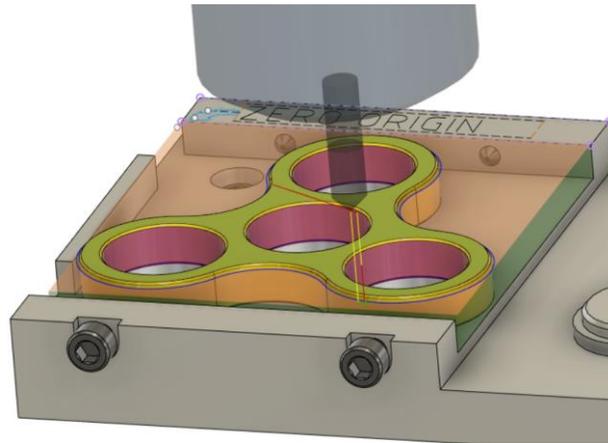
18. Select Ok to complete the contour toolpath adjustments.

19. Click the **Save** icon  to save your design.

Step 6: Verify Toolpaths with Stock Simulation

After completing this lesson, you will be able to:

- Verify toolpath simulation seeing material be removed from stock
- Adjust stock transparency
- Visualize machining action and tool motion



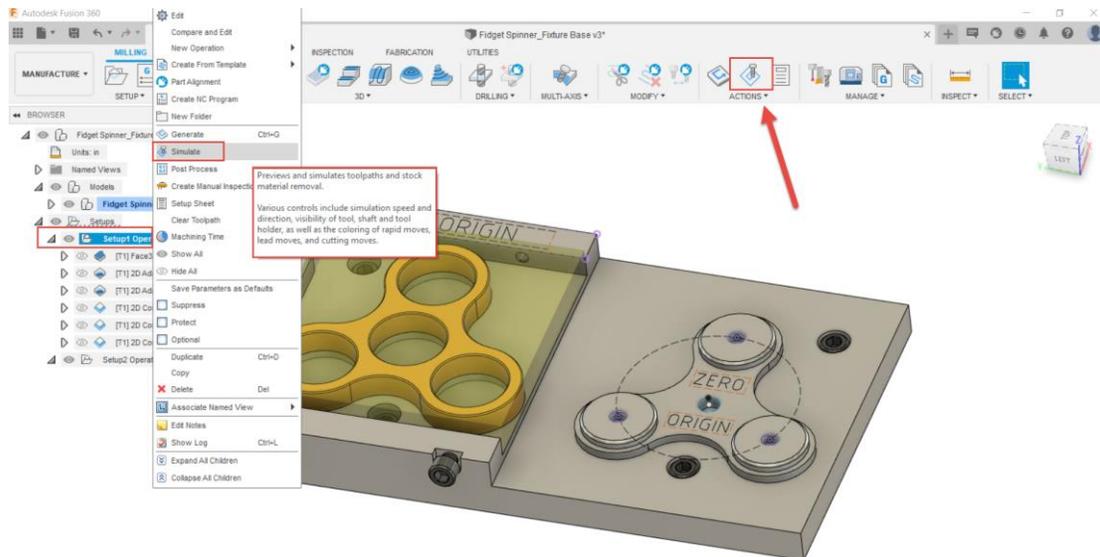
Toolpath simulation in the Manufacturing workspace visualizes the movement of a tool as it travels along a toolpath, showing how much stock material is cut away with every operation. Simulating gives you the ability to validate toolpaths, and check for potential collisions, before you post process operations.

You can simulate a single toolpath, a combination of toolpaths, or an entire setup. Toolpaths are simulated in the order that they appear in the Browser.

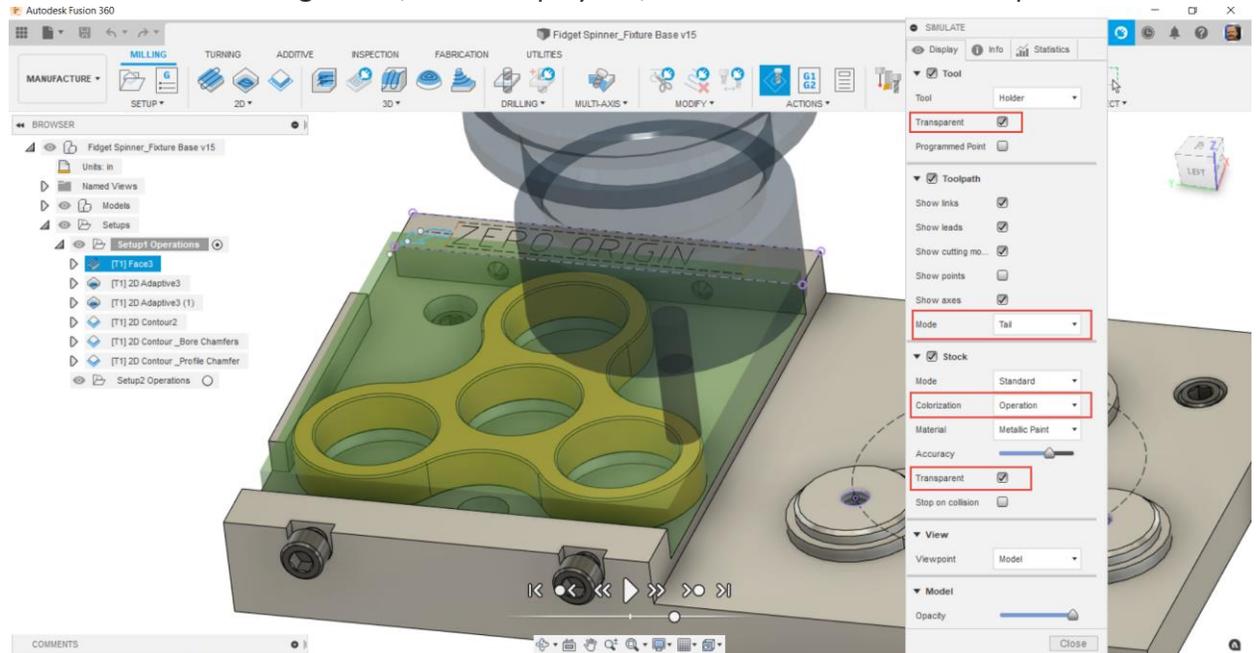
When including the visualization of stock during simulation, the colorization option lets you decide how the stock looks. You can show the stock in a chosen material, apply different colors depending on the current operation or the type of tool, or use comparison colors to indicate when too much, or not enough material has been cut away.

You can control the opacity of the model during simulation, so that the model does not obscure the stock, for better stock visualization.

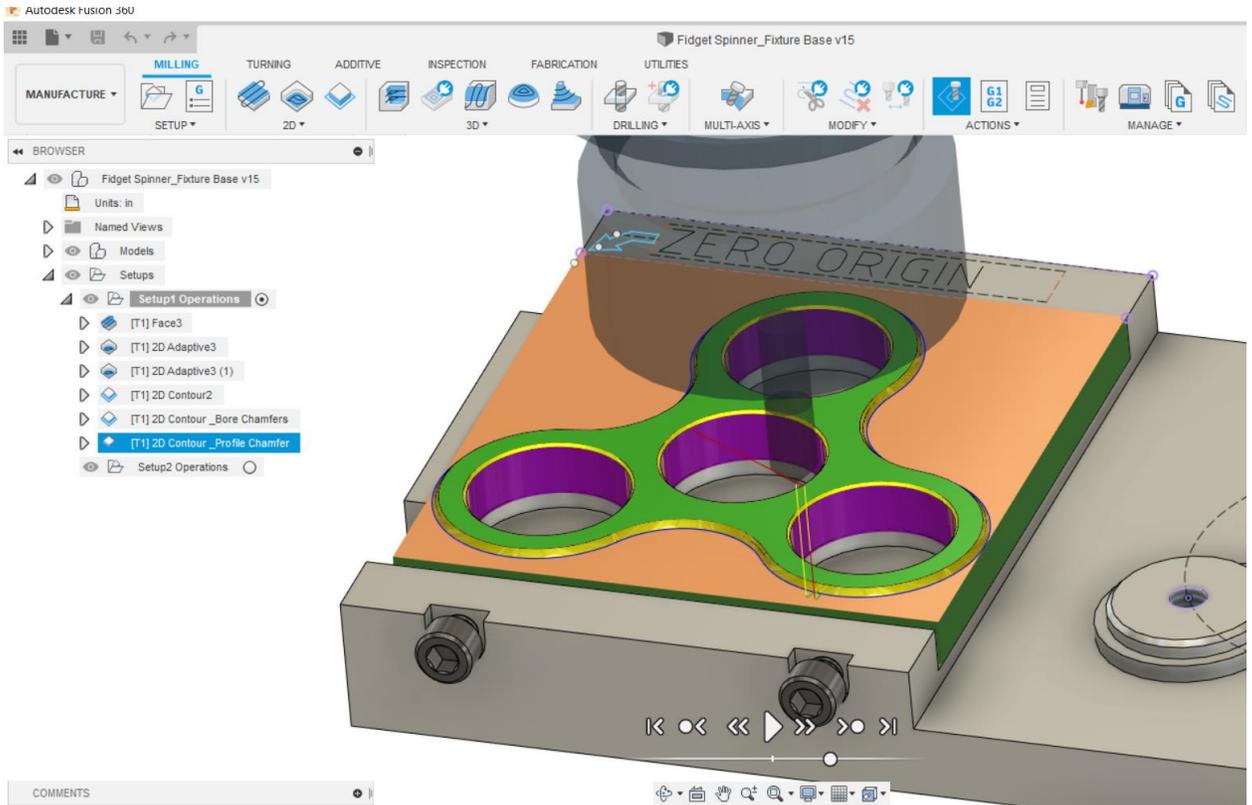
1. To verify toolpaths with stock simulation, right-click on Setup1 Operations folder and choose Simulate or alternatively you can activate simulation under the Actions drop down menu.



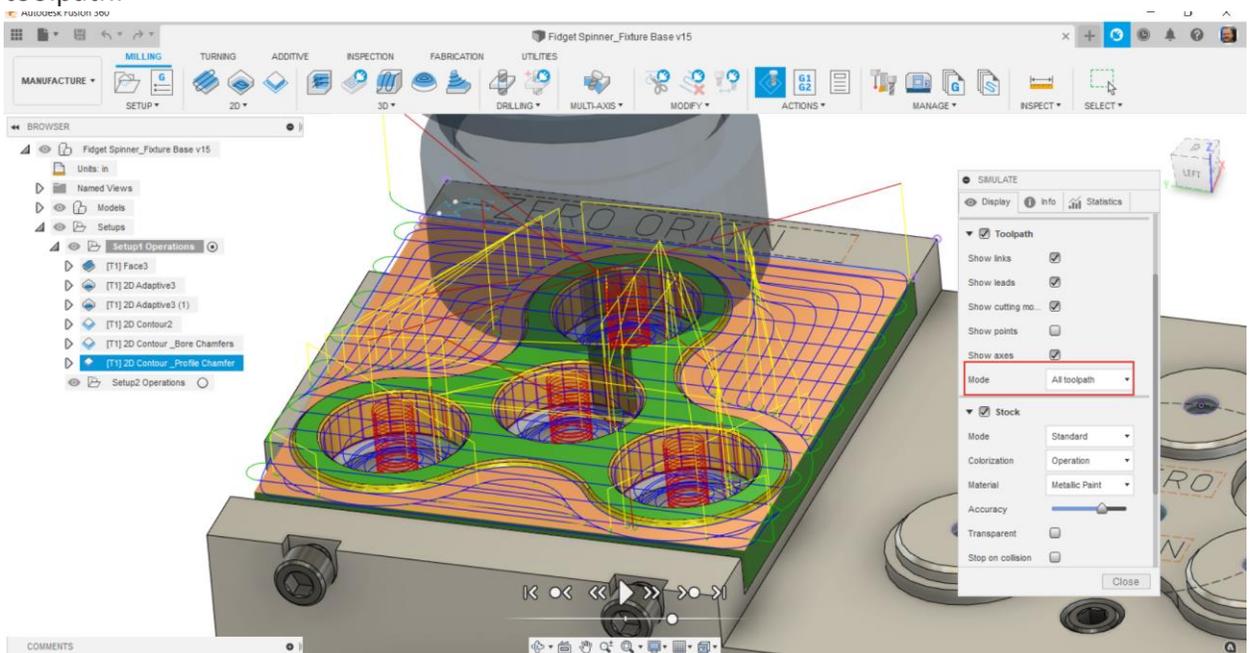
2. In the simulation dialog box, in the Display tab, activate the tool to be transparent.



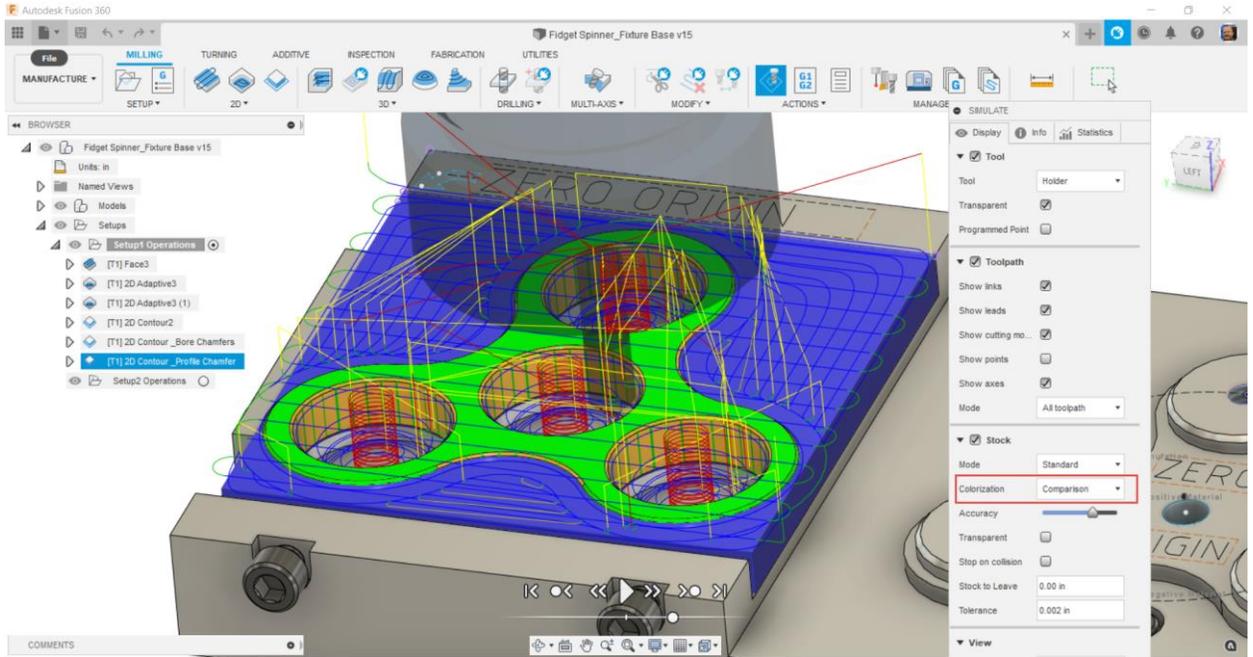
3. Ensure toolpath simulation mode is set to Tail and Colorization is set by Operation.
4. Turn on Stock material to be Transparent.
5. Proceed to select the play button to visualize the tool removing material from the stock body.



6. Once toolpath simulation is complete you will see a visual representation of the current machined condition of the stock
7. Try adjusting some of the simulation display options such as setting toolpath mode to All toolpath.



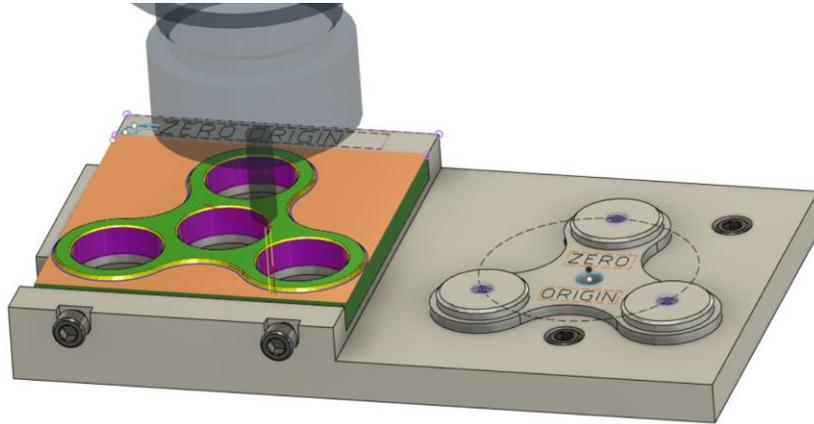
8. Additionally, you can try changing the Stock colorization to Comparison mode.
9. This mode shows exactly where left over stock material is remaining in blue color.



10. You can also explore both the Info and Statistics tabs for additional information such as estimated machining cycle time.
11. With having toolpath simulation fully verified you can proceed to close simulation mode.
12. Click the **Save** icon  to save your design.

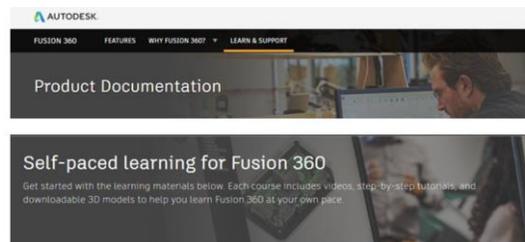
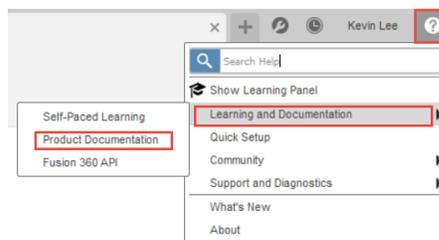
CONGRATULATIONS

You have successfully completed the guided lessons to create toolpath strategies for Manufacturing CAM Setup1 using Autodesk Fusion 360.

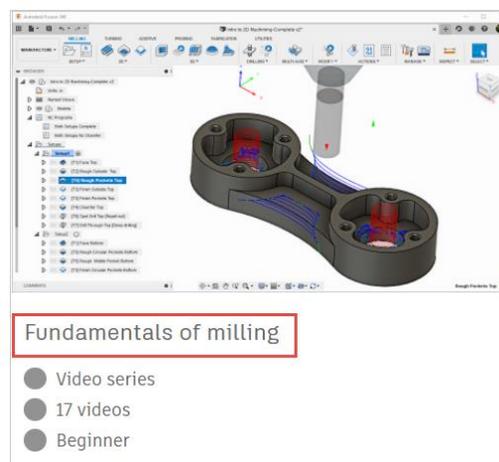


With in Fusion 360 you can directly access additional learning resources. When inside of Fusion 360 near the top right corner of your screen, select the help icon.

Here you can access further self-paced learning tutorial guides and additional product documentation.



To learn more on the fundamentals of Milling using Fusion 360, check out the self paced learning tutorial videos.



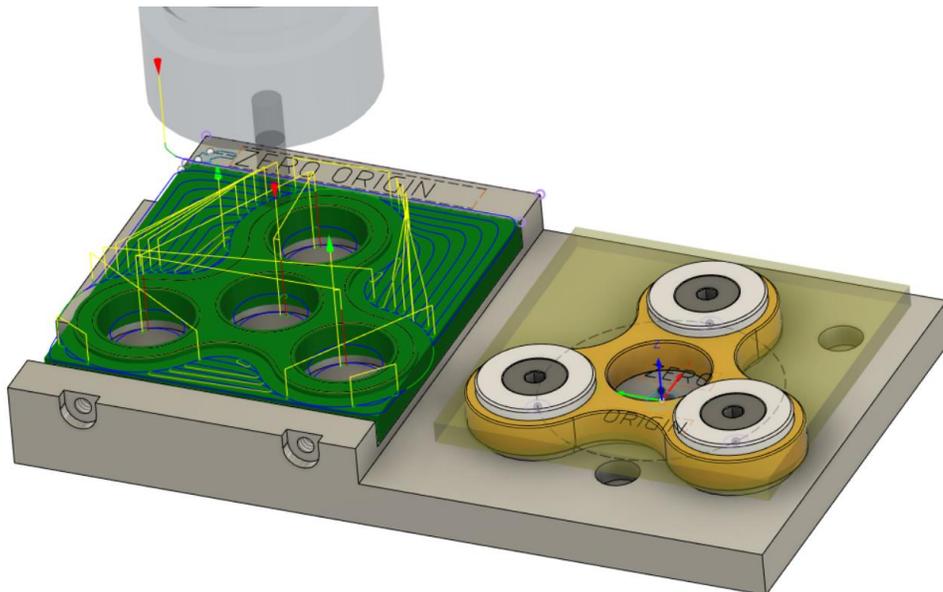
LESSON 7

Manufacturing CAM Setup#2

After completing this lesson, you will be able to:

- Copy toolpaths to another Setup
- Reapply toolpaths
- Set Chamfer Tip Offset
- Verify with Toolpath Simulation
- Create Setup Sheet Documentation

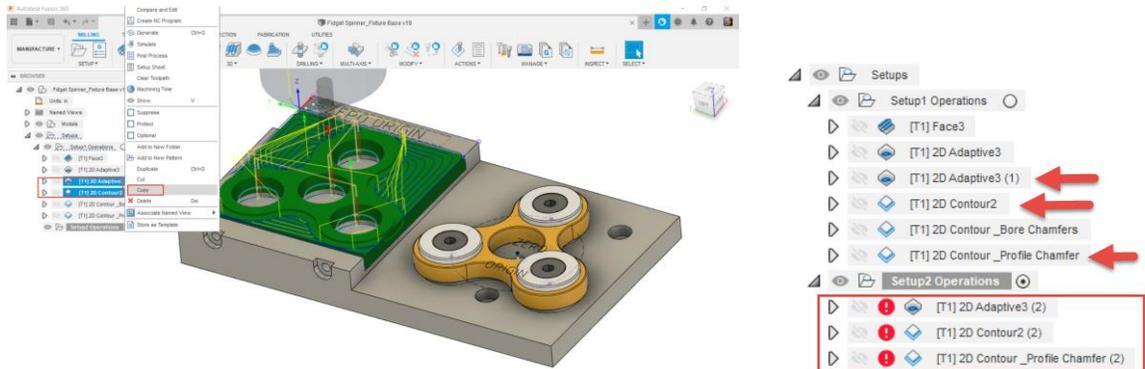
Fusion 360 allows you to build on your work with effective ways of quickly creating new toolpaths operations from existing operations. Rather than creating new toolpaths for the 2nd side of the part, you can use proven toolpaths from CAM Setup#1.



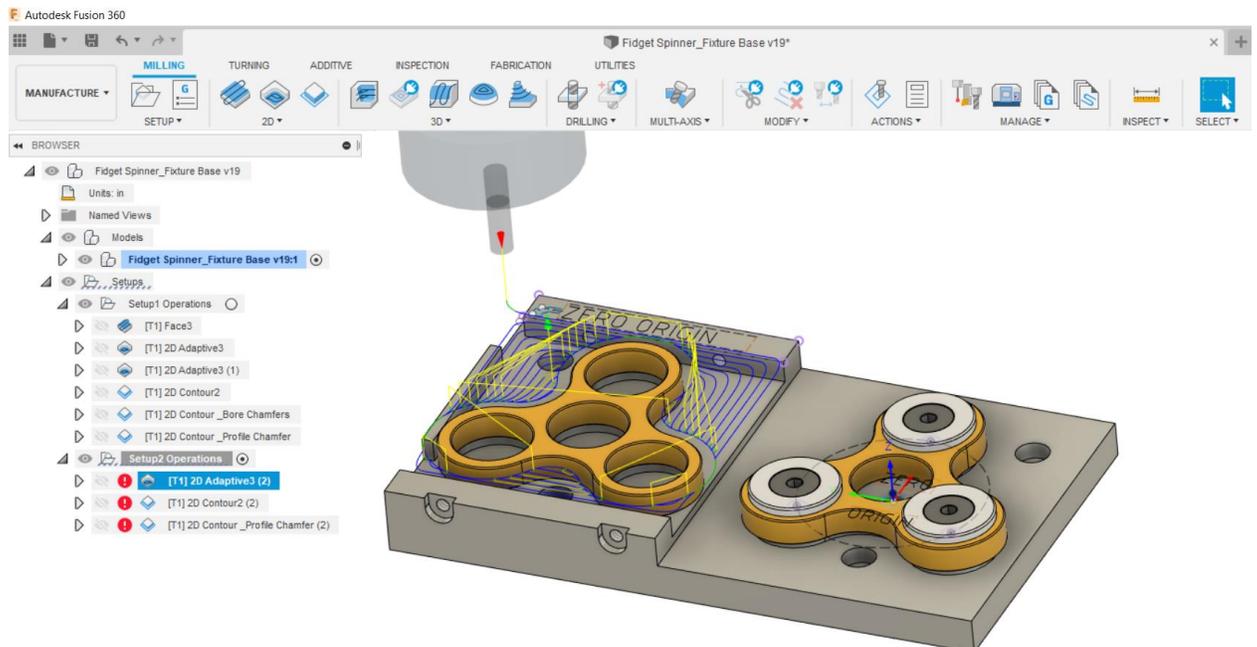
Copying existing toolpaths will improve work flow, reduce programming time and produce consistent toolpath output. We don't need to copy all of the toolpaths, only select toolpaths from Setup1.

Step 1: Copy and Paste Toolpaths

1. With your currently open active document, **Fidget Spinner_Fixture Base**, ensure you are in the Manufacture workspace.
2. Be sure that CAM Setup2 Operations folder is active; and select secondary 2d adaptive operation, 2d contour operation, and 2d contour_Profile Chamfer operation from Setup1 Operations folder, right click, and choose copy.

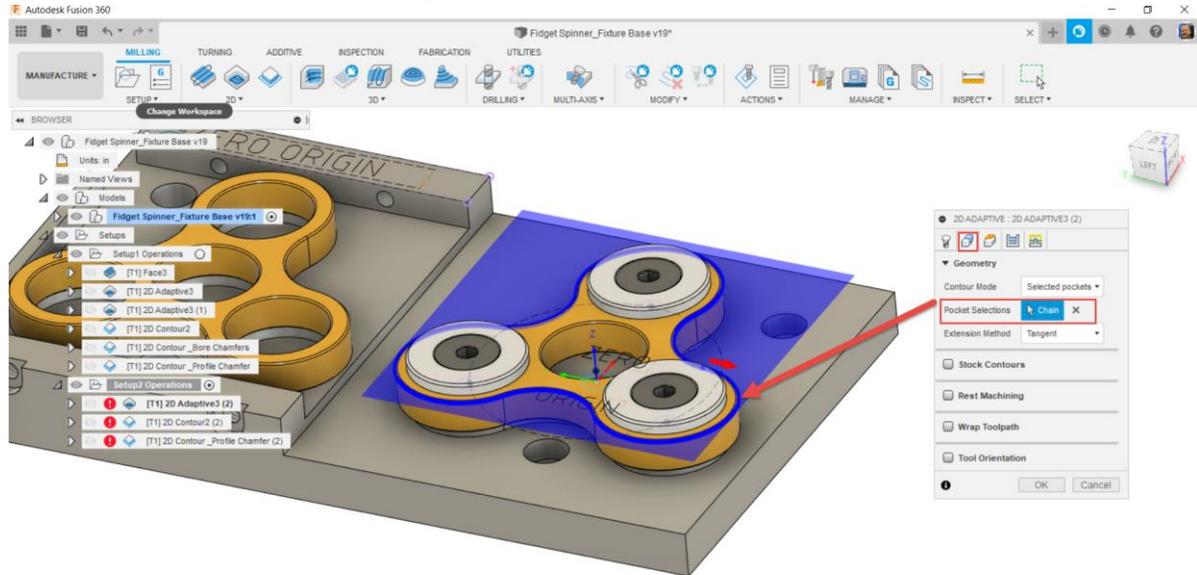


3. Then right-click on Setup2, and choose paste.
4. Once the operations have been copied, you'll notice that they are marked with a red exclamation mark. This is a dirty toolpath, meaning, something has changed in a way that the toolpath is no longer valid.
5. When you highlight the 2d adaptive toolpath you will notice that it really is a copied toolpath as it still uses the geometry selection that was defined for Setup#1.

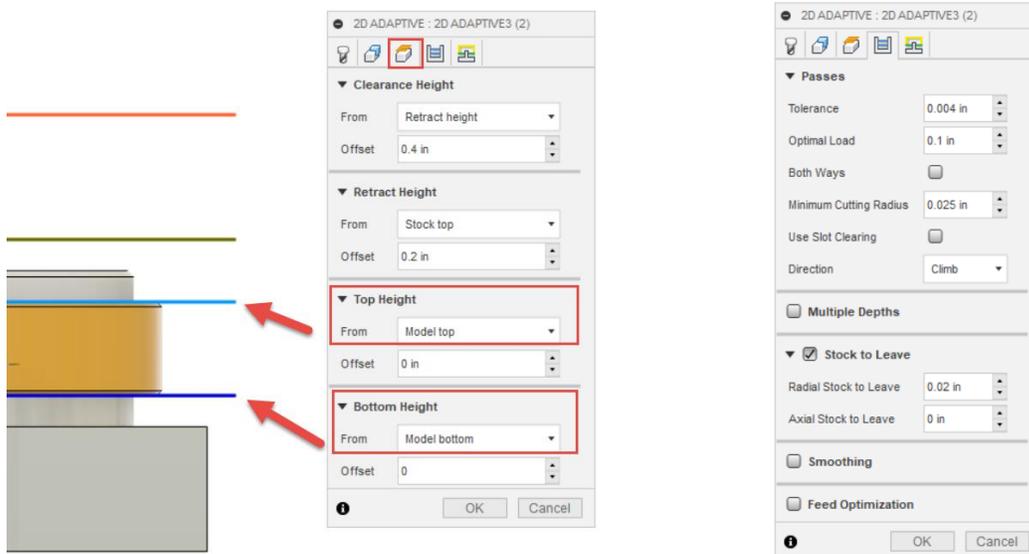


Step 2: Reapply Toolpath Geometry Selections

1. Select the 2d adaptive toolpath in Setup2 Operations folder, right-click, Edit toolpath.
2. In the geometry tab, clear the previous geometry selection, and reselect the outer profile, the lower chamfered edge.

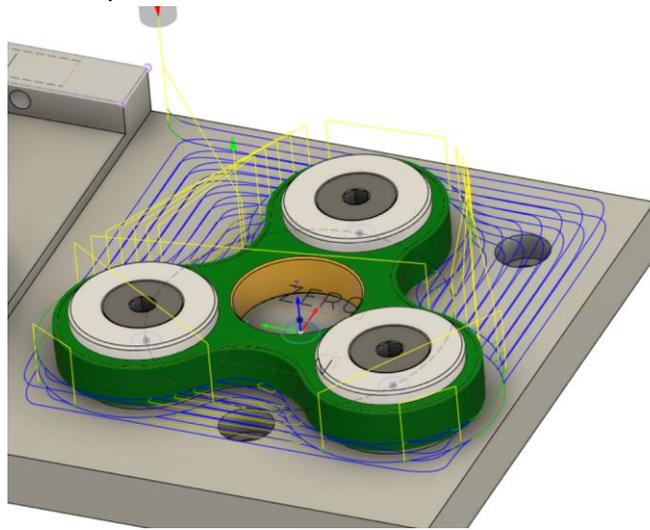


3. In the Heights tab, adjust the Bottom height to be from Model Bottom, with 0 Offset, and adjust the Top Height to be from Model Top.

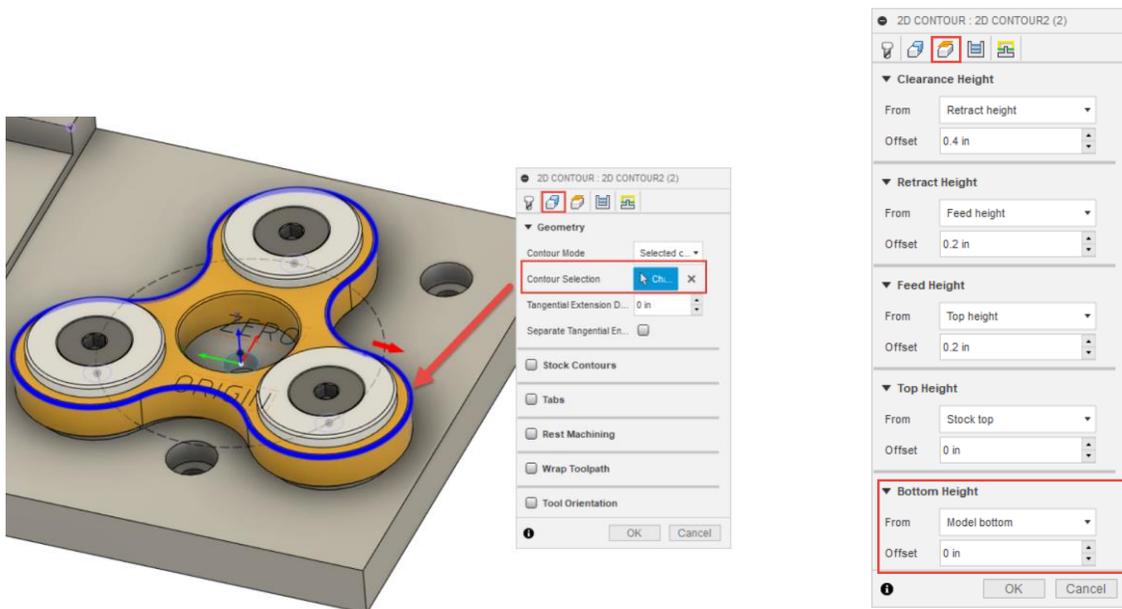


4. The 4th tab, the passes tab, will have all the same parameter settings as the adaptive operation in Setup#1 because this is a copied toolpath.
5. However, considering this new operation will be roughing away lots of material from the outside profile of the part; it's best to add some depths of cut so that tool can machine safely.

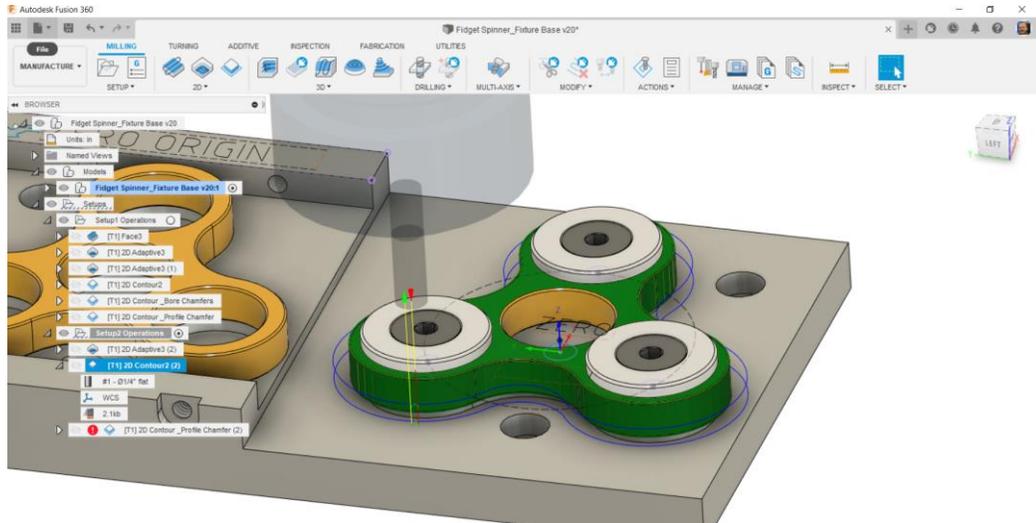
6. In the Passes tab, enable Multiple Depths and set Maximum Roughing Stepdown to 0.150”.
7. Select Ok to calculate toolpath.



8. Repeat this process but for the 2D contour toolpath operation.
9. When editing the 2D contour toolpath, reselect new driving geometry in the geometry tab, select the same outer edge profile.
10. Ensure the red arrow is on the outside of the geometry selection.



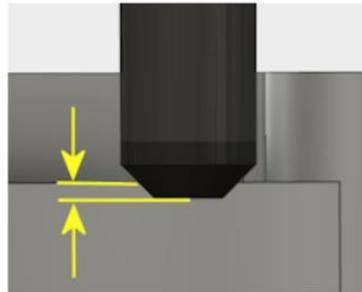
11. In the Height tab, adjust the Bottom Height to be from Model Bottom with a 0 offset amount.
12. Adjust the Maximum Roughing Depths cut to be 0.150” in the Passes tab.
13. Select Ok to calculate toolpath.



14. Repeat this process but for the 2D contour_Profile Chamfer toolpath operation.
15. When editing the 2D contour toolpath, reselect new driving geometry in the geometry tab, select the outer edge profile.
16. Ensure the red arrow is on the outside of the geometry selection.
17. Select Ok to calculate toolpath.
18. All 3 toolpaths should now have updated toolpath geometry that effectively machines the for Setup#2 Operations.
19. Click the **Save** icon  to save your design.

Step 3: Setting Chamfer Tip Offset Amount

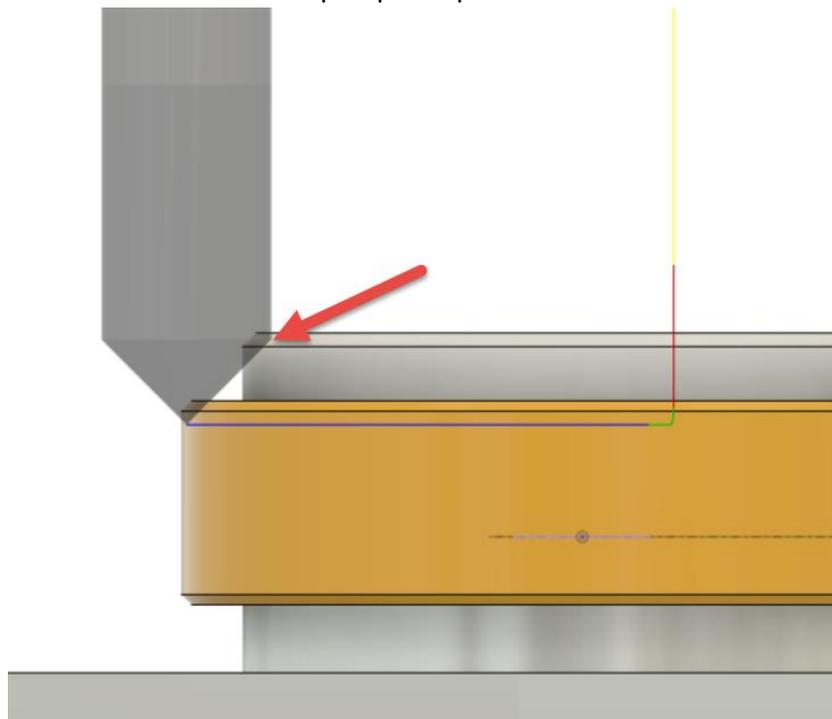
The Chamfer Tip Offset parameter located in the Passes tab; adds an amount to the toolpath depth, while keeping the tool in contact with the selected edge by adjusting the toolpath radial amount.



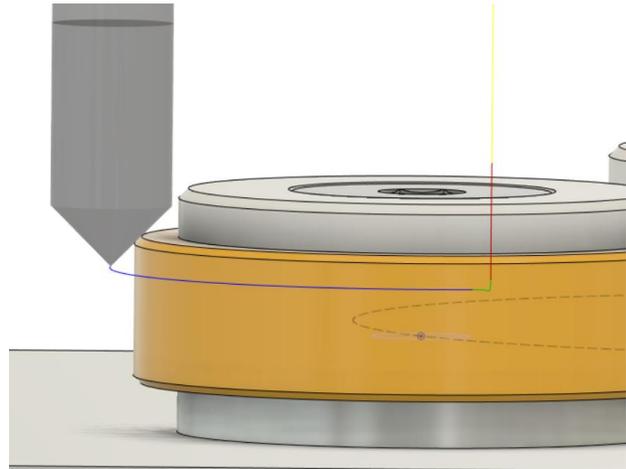
Chamfer Tip Offset
Tool Tip Clearance Extension

If the CAD model has already been chamfers, the easiest way is to select the lower edge of the chamfer. This puts the tool chamfer corner at this position, and you only need to adjust the tip offset if you want to go some distance below the lower edge.

1. Now when you right-click on toolpath, and simulate; you will notice that the tool actually interferes with the Fixture OP2 Clamp Cap component.



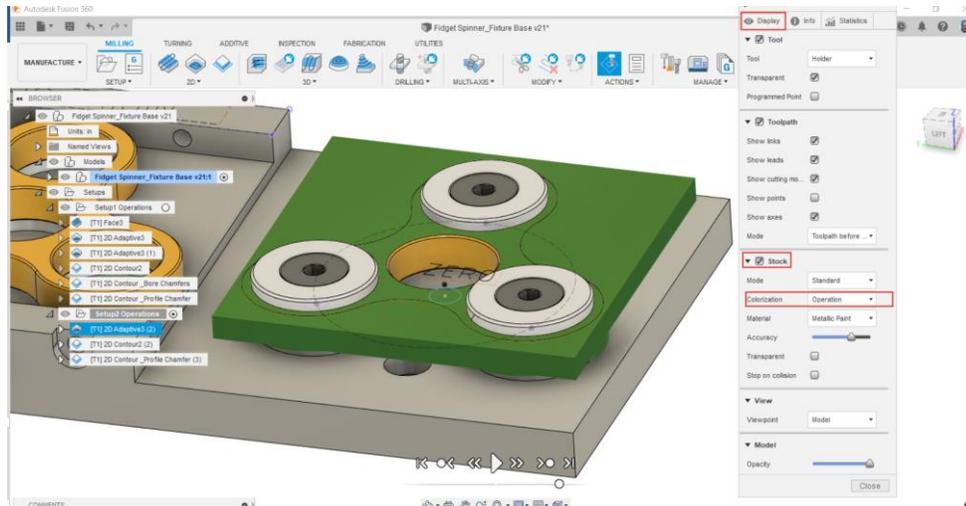
2. Edit the toolpath, in the Passes tab, adjust the Chamfer Tip Offset amount to be 0.050", and hit OK to regenerate the toolpath.
3. Now you will see that the toolpath is forced down deeper, effectively machining the chamfer further up the cutting flutes. This, inherently, adjusts the toolpath radial amount to provide clearance for the tool to not interfere with the Fixture OP2 Clamp Cap component.



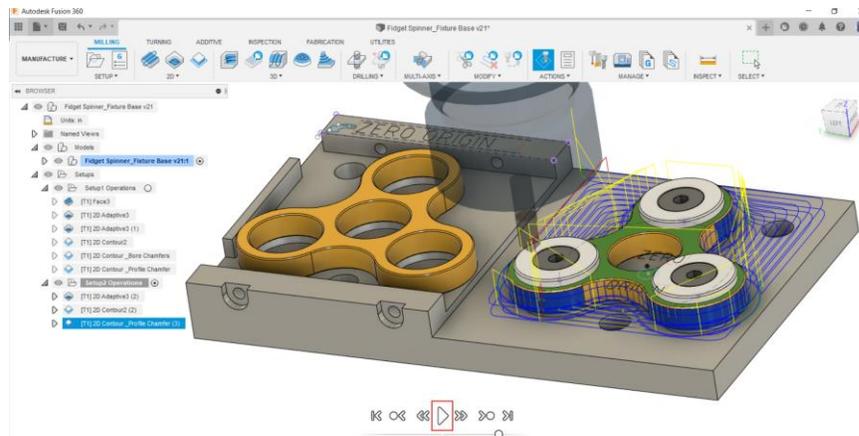
4. Click the **Save** icon  to save your design.

Step 4: Verify Operations with Toolpath Simulation

1. To verify toolpaths with stock simulation, right-click on Setup2 Operations folder and choose Simulate or alternatively you can activate simulation under the Actions drop down menu.
2. In the Simulation dialogue window, ensure Stock is enabled and colorization is set to Operation.



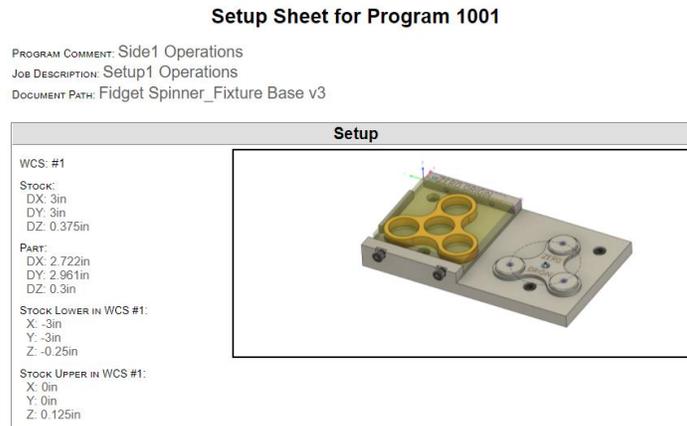
3. Play the toolpath simulation and observe toolpath behavior.



4. Once complete, and you are satisfied with expected toolpath behavior, close the simulation.
5. Click the **Save** icon  to save your design.

Step 5: Create Setup Sheet Documentation

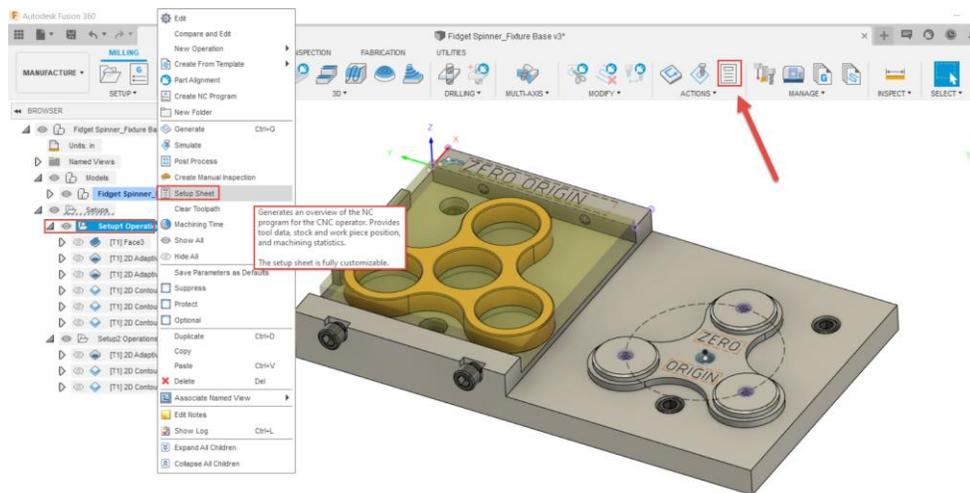
The Setup Sheet is generally output for the machine operator and can be viewed in any standard web browser.



This is the documentation about the stock size, job description, zero position, tool, feeds, speeds and work offsets. It can be printed and sent out to the shop floor or, if the operator has a PC at the machine, it can be made available on the network without the need to print a document.

You'll be given a choice of where to save the Setup Sheet file. If you're on a network you can save this to a folder that the operator can access. This will save paper and insure the operator has access to the most current Setup Sheet information.

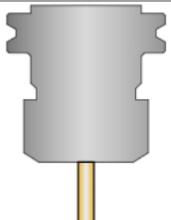
1. To create Setup Sheet documentation, right click on Setup 1 Operations, choose Setup Sheet. Another way of creating a setup sheet, choose the setup sheet icon in the Actions menu in ribbon bar as shown below.



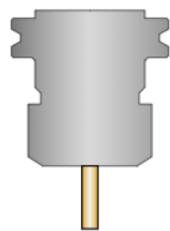
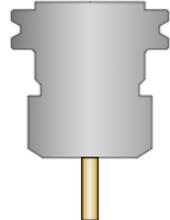
2. Confirm your Setup Sheet Output folder destination.

3. A new setup sheet html document should appear on your screen.
4. Notice, how it creates a screen capture of your Fusion 360 design showing WCS orientation and transparent stock.
5. Setup Sheet Documentation contains useful information such as: stock dimensions compared to part dimensions, number of operations, number of tools being used, cutting distance, and estimated cycle time.
6. Further information is provided that includes selected tooling information and toolpath operation order.

Tools		
T1 D1 L1		
TYPE: flat end mill	MINIMUM Z: -0.26in	HOLDER: Maritool CAT40-ER32-2.35
DIAMETER: 0.25in	MAXIMUM FEED: 20in/min	VENDOR: Maritool
LENGTH: 1in	MAXIMUM SPINDLE SPEED: 15000rpm	PRODUCT: CAT40-ER32-2.35
FLUTES: 3	CUTTING DISTANCE: 269.622in	
	RAPID DISTANCE: 68.23in	
	ESTIMATED CYCLE TIME: 22m:37s (98.9%)	



Operations		
Operation 1/6		
DESCRIPTION: Face3	MAXIMUM Z: 0.725in	T1 D1 L1
STRATEGY: Facing	MINIMUM Z: 0.05in	TYPE: flat end mill
WCS: #1	MAXIMUM SPINDLE SPEED: 15000rpm	DIAMETER: 0.25in
TOLERANCE: 0in	MAXIMUM FEEDRATE: 20in/min	LENGTH: 1in
MAXIMUM STEPOVER: 0.2in	CUTTING DISTANCE: 52.887in	FLUTES: 3
	RAPID DISTANCE: 1.05in	
	ESTIMATED CYCLE TIME: 5m:16s (23%)	
	COOLANT: Flood	
Operation 2/6		
DESCRIPTION: 2D Adaptive3	MAXIMUM Z: 0.725in	T1 D1 L1
STRATEGY: Adaptive 2D	MINIMUM Z: -0.26in	TYPE: flat end mill
WCS: #1	MAXIMUM SPINDLE SPEED: 15000rpm	DIAMETER: 0.25in
TOLERANCE: 0.004in	MAXIMUM FEEDRATE: 20in/min	LENGTH: 1in
STOCK TO LEAVE: 0.02in/0in	CUTTING DISTANCE: 110.113in	FLUTES: 3
MAXIMUM STEPDOWN: 0.3in	RAPID DISTANCE: 15.922in	
OPTIMAL LOAD: 0.1in	ESTIMATED CYCLE TIME: 7m:22s (32.2%)	
LOAD DEVIATION: 0.01in	COOLANT: Flood	

LESSON 8

Haas Desktop CNC Mill

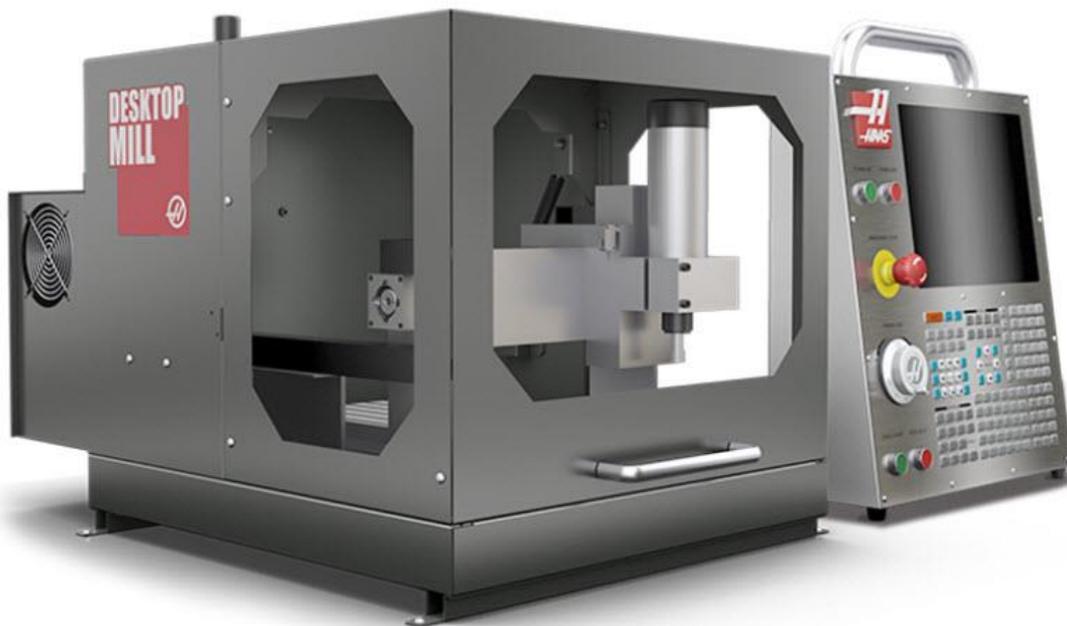
&

Work Holding Setup

After completing this lesson, you will be able to:

- Secure Work Holding Fixture

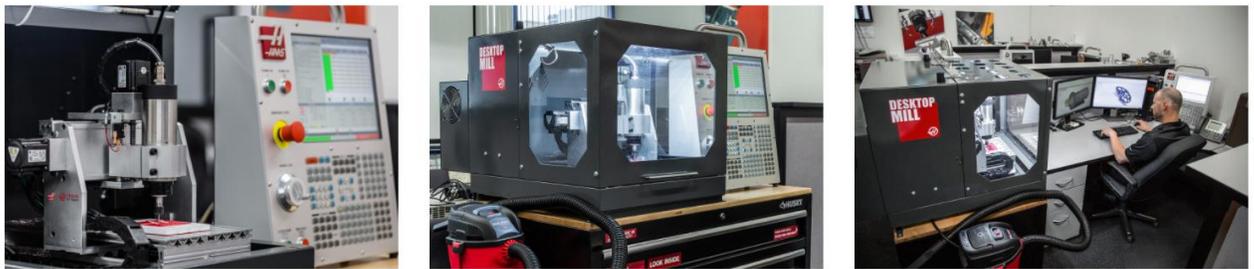
The Haas Desktop Mill is a compact CNC milling training machine for schools and technical colleges. Powered by a full-function Haas CNC control housed in a portable simulator enclosure. The Desktop Mill is perfect for teaching the basic principles of programming and operating a CNC milling machine. Designed to cut plastics and machinable wax, it is also perfect for hobbyists and benchtop machinists.



Although the Desktop Mill is small, it has the same functionality and controls as full-size Haas machines, so nothing is lost in translation when transitioning from the Desktop Mill to a production machine.

So, students and hobbyists can learn actual machine operation and programming techniques, without needing the space or resources required for a production machine. Operators have full access to, and quickly learn, all the powerful functions of the Haas control, such as the easy-to-use Visual Programming System (VPS), M130 media display, WiFi networking, HaasDrop and more.

Full Function Haas Control



Teaching CNC setup, operation, and programming has never been easier or more economical. This desktop milling machine may look small but has big features that include:

- Same full-function Haas control and interface as all Haas machines
- Can be placed on a desk, table, cabinet, or mounted to a rolling toolbox
- Turn any space into a CNC learning lab
- Does not need compressed air
- Does not need coolant, coolant maintenance, or disposal
- Runs on single-phase 120V or 220V power (must be specified at time of order)
- No dedicated power drop needed, just plug into the wall

For additional product information please visit:

<https://www.haascnc.com/machines/vertical-mills/desktop-mill.html>

Step 1: Secure Work Holding Fixture

CNC machining requires effective work holding that ensures the part will remain stable, fully secure and can withstand the forces exerted during the machining process. Work holding can be as simple as using a vise clamped to the table or fixtures can be specifically designed to secure unique part shapes.

1. The Haas Desktop CNC milling machine may come with an additional learning kit that includes a specially designed and manufactured work holding fixture as shown below.



2. This training guide has shown in earlier lessons the Fusion 360 CAD model work holding fixture.
3. Proceed to mount the work holding fixture on the machine table so that the front edge of the fixture is a minimum of 1-1/8" from the front edge of the machine table as shown above.
4. Note that the bottom side of the fixture is keyed to align with the t-slot of the machine so that when its clamped tightly to the table, the fixture is aligned perfectly with the X axis of the machine.
5. This keyed feature on the fixture removes the need for dialing in the fixture to ensure alignment with the machine axis.

LESSON 9

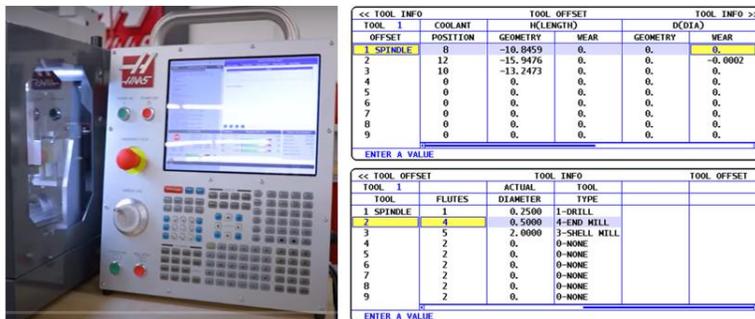
Qualify Tool Lengths & Work Offsets

After completing this lesson, you will be able to:

- Understand Machine Offsets
- Set Tool Length Offsets
- Qualify G54 Fixture Work Offsets- X & Y axis
- Qualify G54 Fixture Work Offsets Z axis
- Calculate G55 Fixture Work Offset

CNC motion is based on the Cartesian coordinate system. A CNC machine cannot be successfully operated without an understanding of how the coordinate systems are defined in CAM software in conjunction with the CNC machine controller and how the two systems work together.

Typically, the machine work coordinate system (WCS) is set on the CNC machine so the machine knows where the part is located within the machinable workspace.



When a CNC machine is first turned on, it does not know where the axes are positioned in the workspace. Home position is found by the Power On Restart sequence initiated by the operator by pushing a button on the machine control after turning on the control power.

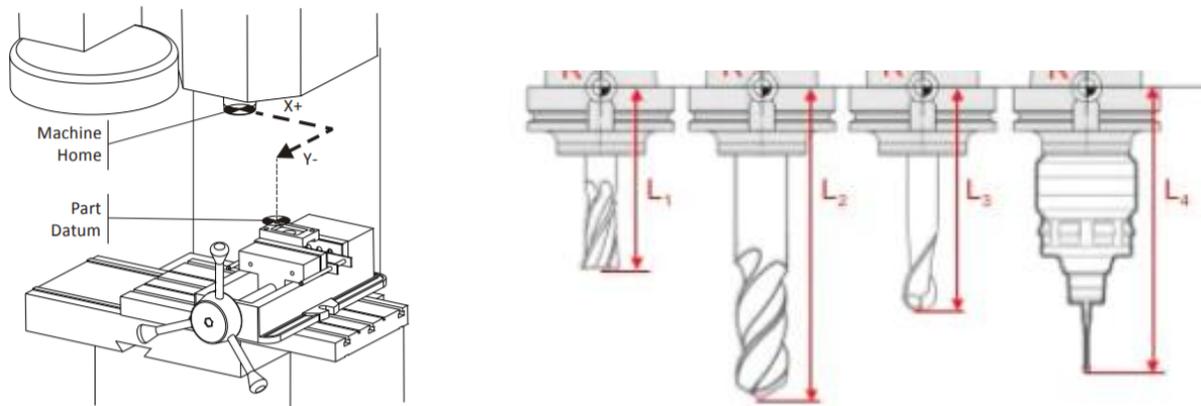
The Power On Restart sequence simply drives all three axes slowly towards their extreme limits (-X, +Y, +Z). As each axis reaches its mechanical limit, a microswitch is activated. This signals to the control that the home position for that axis is reached. Once all three axes have stopped moving, the machine is said to be "homed".

Machine coordinates and offsets thereafter are in relation to this home position.

Step 1: Understanding Machine Offsets

For the most part, it is difficult to place a vise or fixture in the exact same position on the machine each time; the distance from the Home position to the WCS is usually not known until the vise/fixture is set and aligned with the machine axis.

To complicate matters further, different tools extend out from the machine spindle at different lengths. For example, a long end mill extends further from the spindle face than a stub length drill. If the tool wears or breaks and must be replaced, it is almost impossible to set it in the exact same length out of the tool holder each time.



Therefore, there must be some way to relate the Machine Coordinate system to the part WCS and take into account varying tool lengths.

This is done using machine Tool and Fixture Offsets.

Step 2: Tool Length Offsets

Every tool loaded into the machine is a different length. In fact, if a tool is replaced due to wear or breaking, the length of its replacement will likely change because it is almost impossible to set a new tool in the holder in exactly the same place as the old one.

The CNC machine needs some way of knowing how far each tool extends from the spindle to the tip. This is accomplished using a Tool Length Offset (TLO).

Setup: Jog | 13:54:57

MEM Memory/101.nc N195

000101 (Desktop);
 (Using high feed G1 F650. instead of G0.);
 (T1 D=0.25 CR=0, - ZMIN=-0.3 - flat end mill);
 N10 G90 G94 G17;
 N15 G20;
 N20 G53 G0 Z0.0;
 ;
 (2D Adaptive2);
 N25 T1 ;
 N30 S15000 M3;
 N35 G54;
 N40 M8;
 N45 G1 X-2.3416 Y-2.4851 F100. ;
 N50 G0 G43 Z0.85 H1;
 N55 G0 Z0.45;
 N60 Z0.35;
 N65 G3 X-2.3496 Y-2.4785 Z0.3243 I0.071 J0.0952
 F13.333;
 N70 X-2.357 Y-2.4713 Z0.2986 I0.079 J0.0887;
 N75 X-2.3638 Y-2.4635 Z0.2729 I0.0864 J0.0815;
 N80 X-2.3699 Y-2.4551 Z0.2472 I0.0932 J0.0736;
 N85 X-2.3752 Y-2.4462 Z0.2215 I0.0992 J0.0652;
 N90 X-2.3797 Y-2.4368 Z0.1958 I0.1045 J0.0563;
 N95 X-2.3834 Y-2.4271 Z0.1701 I0.1091 J0.047;
 N100 X-2.3862 Y-2.4172 Z0.1445 I0.1127 J0.0373;
 N105 X-2.3881 Y-2.407 Z0.1188 I0.1156 J0.0273;
 N110 X-2.3892 Y-2.3967 Z0.0931 I0.1175 J0.0172;
 N115 Y-2.3892 Z-2.3967 Z0.0674 I0.1186 J0.0060.

Offsets

Active Tool: 1 Coolant Position: 0

Tool Offset	Length Geometry(H)	Length Wear(H)	Diameter Geometry(D)	Diameter Wear(D)	Flutes
1 Spindle	-2.3580	0.	0.	0.	2
2	0.	0.	0.	0.	2
3	0.	0.	0.	0.	2
4	0.	0.	0.	0.	2
5	0.	0.	0.	0.	2
6	0.	0.	0.	0.	2
7	0.	0.	0.	0.	2
8	0.	0.	0.	0.	2
9	0.	0.	0.	0.	2
10	0.	0.	0.	0.	2
11	0.	0.	0.	0.	2
12	0.	0.	0.	0.	2
13	0.	0.	0.	0.	2
14	0.	0.	0.	0.	2
15	0.	0.	0.	0.	2
16	0.	0.	0.	0.	2
17	0.	0.	0.	0.	2

TOOL OFFSET MEAS Tool Offset Measure ALTER Tool Presetter F1 Set Value
 ENTER Add To Value F4 Work Offset

Main Spindle

STOP

Overrides

Feed: 100%
 Spindle: 100%
 Rapid: 100%

Spindle Speed: 0 RPM
 Spindle Power: 0.0 KW
 Surface Speed: 0 FPM
 Chip Load: 0.000 IPT
 Feed Rate: 0.0000 IPM
 Active Feed: 0.0000 IPM

Spindle Load(%) 0%

Hand Jog

Position: (IN) Jog Rate: 0.0010
 Work G54 Distance To Go Machine Operator

	Work G54	Distance To Go	Machine	Operator
X	0.9440	0.0000	0.0000	0.0000
Y	2.7790	0.0000	0.0000	0.0000
Z	-2.3580	-2.6060	-2.6060	-2.6060

Setup Jog Opt Stop

Input:

In it's simplest use, the TLO is found by jogging the spindle with tool from the machine home Z-position to the part Z zero position.

1. To begin, mount the 1/4" endmill into the spindle collet so that the tool length sticks out approximately 1.125" and securely tighten with wrenches. You can use the 3d printed tool gage provided in your workholding fixture kit to help with setting the approximate tool length below holder.



2. On the Haas controller, select the Offset button to display the Offsets page, scroll to highlight the tool 1 offset values.

Offsets

Tool Work Tool Info

Active Tool: 1 Coolant Position: 0

Tool Offset	Length Geometry(H)	Length Wear(H)	Diameter Geometry(D)	Diameter Wear(D)	Flutes
1 Spindle	-2.3580	0.	0.	0.	2
2	0.	0.	0.	0.	2
3	0.	0.	0.	0.	2
4	0.	0.	0.	0.	2
5	0.	0.	0.	0.	2
6	0.	0.	0.	0.	2
7	0.	0.	0.	0.	2
8	0.	0.	0.	0.	2
9	0.	0.	0.	0.	2
10	0.	0.	0.	0.	2
11	0.	0.	0.	0.	2
12	0.	0.	0.	0.	2
13	0.	0.	0.	0.	2
14	0.	0.	0.	0.	2
15	0.	0.	0.	0.	2
16	0.	0.	0.	0.	2
17	0.	0.	0.	0.	2

TOOL OFFSET MEAS Tool Offset Measure ALTER Tool Presetter F1 Set Value

ENTER Add To Value F4 Work Offset

3. In manual Jog mode, jog the Z axis down carefully towards the 1-2-3 block resting on the table as shown below.



4. Make sure to move the 1-2-3 block out of the way slightly so that you can jog the tool down lower than the 1-2-3 block on purpose. Do not jog the tool into the 1-2-3 block causing damage.
5. Then gently jog the Z-axis **upward** in 0.001" increment mode until you can slide the 1-2-3 block under the tool smoothly and safely, taking care not to damage the tool cutting flutes.
6. Once satisfactory, on the Haas controller, press Tool Offset Measure button.
7. Notice that the Z length of the tool (from Z home position) is recorded in the Tool Offsets Length column, establishing the recorded length for tool 1.

Step 3: Qualify G54 Fixture Work Offsets- X & Y axis

Fixture offsets provide a way for the CNC control to know the distance from the machine home position and the part WCS. In conjunction with Tool Offsets, Fixture Offsets allow the programs to be written in relation to the WCS instead of the Machine Coordinates.

They make setups easier because the exact location of the part in the machine envelope does not need to be known before the CNC program is written.

The screenshot shows the CNC control interface with the following data:

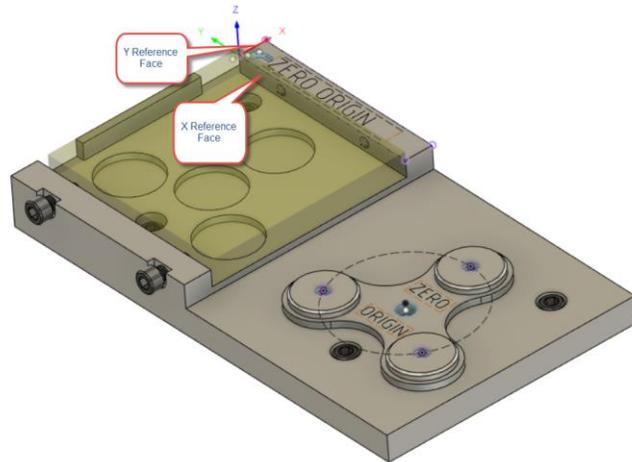
G Code	X Axis	Y Axis	Z Axis	Work Material
G52	0.	0.	0.	No Material Selected
G54	-0.9440	-2.7790	-0.2480	No Material Selected
G55	-2.7020	-7.7780	-0.4480	No Material Selected
G56	0.	0.	0.	No Material Selected
G57	0.	0.	0.	No Material Selected
G58	0.	0.	0.	No Material Selected
G59	0.	0.	0.	No Material Selected
G154 P1	0.	0.	0.	No Material Selected
G154 P2	0.	0.	0.	No Material Selected
G154 P3	0.	0.	0.	No Material Selected
G154 P4	0.	0.	0.	No Material Selected
G154 P5	0.	0.	0.	No Material Selected
G154 P6	0.	0.	0.	No Material Selected
G154 P7	0.	0.	0.	No Material Selected
G154 P8	0.	0.	0.	No Material Selected
G154 P9	0.	0.	0.	No Material Selected
G154 P10	0.	0.	0.	No Material Selected
G154 P11	0.	0.	0.	No Material Selected
G154 P12	0.	0.	0.	No Material Selected
G154 P13	0.	0.	0.	No Material Selected

Position: (IN)	Distance To Go	Machine	Jog Rate: 0.0010 Operator
X	0.9440	0.0000	0.0000
Y	2.7790	0.0000	0.0000
Z	-2.3580	-2.6060	-2.6060

As long as the part is positioned where the tool can reach all machining operations it can be located anywhere in the machine envelope. Once the Fixture Offset values are found, entered into the control, and activated by the CNC program, the CNC control works behind the scene to translate program coordinates to WCS coordinates.



1. With this project, we will use the paper shim method for qualifying a Work Offset fixture position.
2. With tool 1, 1/4" endmill, installed in the spindle, (spindle off) ; proceed to manually jog the tool position to the left edge of the fixture as shown below.



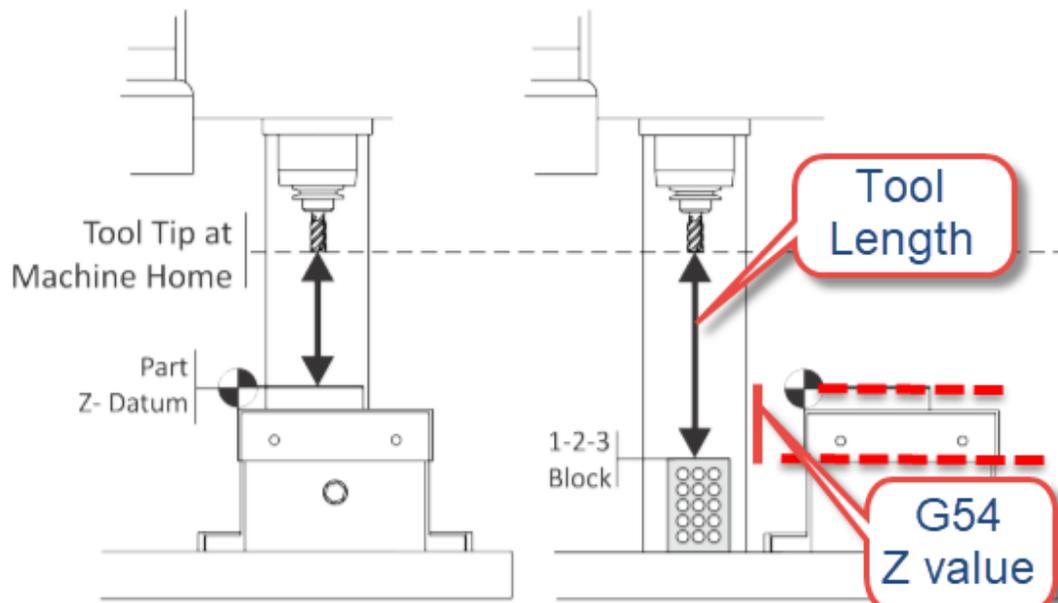
3. Set jog increment to 0.001" mode, and jog in the Y direction very carefully until the paper drags.
4. On the control, in the Work Offsets page, ensure that G54 Work offset Y position is highlighted, press F1 to Set Value.
5. Repeat this process for qualifying the X position for the G54 Work Offset, which will be touching the tool and paper on the inside face of the fixture.
6. Once G54 X and Y positions have been qualified, you will need to adjust this offset position accordingly to compensate for the radius of the tool.
7. In this case, you will need to adjust these X and Y offset positions by 0.125" in their rightful direction.
8. Typically, you will need to add or subtract an amount equal to the tool radius; this is because you touched the side of the tool using the paper against the fixture reference face and recorded the X position of the tool. You want to have the G54 X and Y position recorded using the centre position of the tool.

Step 4: Qualify G54 Fixture Work Offset Z -axis

The Fixture Offset Z value is combined with the Tool Length offset to indicate to the machine how to shift the Z-datum from part home to the part Z-zero, taking into account the length of the tool.

Fixture Offset Z may or may not be used, depending on how the machine is setup and operated.

For example, if you had qualified the tool length with a paper feeler gage off the top reference surface of the fixture, then the tool length recorded would be directly referencing the WCS, Z-datum, of the part.



Considering you qualified the tool length off a 1-2-3 block resting on the machine table, the Z amount difference between the top of the 1-2-3 block and the Z reference face on the fixture becomes the G54 Z-offset value.

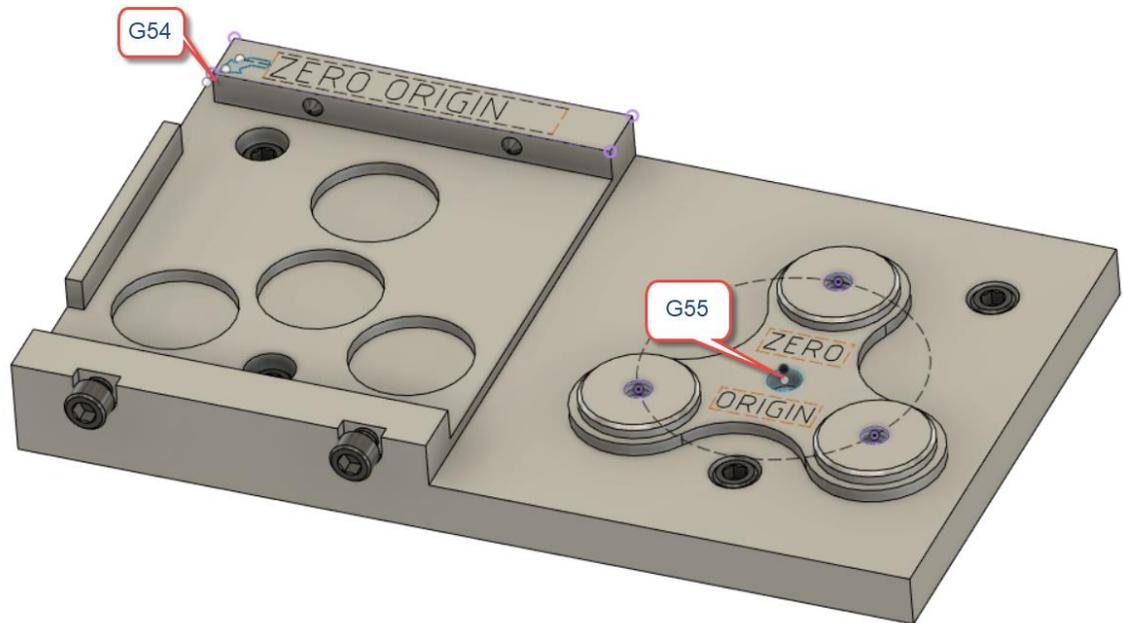
In some situations, the G54 Z-offset value can be either be negative or positive.

1. With tool 1, 1/4" endmill, installed in the spindle; proceed to manually jog the tool position down towards the 1-2-3 block on table.
Note that the 1-2-3 block will be flat on machine table and will look a little different than the image above.
2. When close, Set jog increment to 0.001" mode, and jog in the Z direction very carefully until the appears to be lower than the 1-2-3 block (make sure 1-2-3 block in not in the way) than jog up slowly until you can slide the 1-2-3 block under the bottom edge of tool.

3. On the Haas control, in positional display, you can zero out the Z axis position so that we can use the controller to measure the distance from the 1-2-3- block up to the top surface of the fixture.
4. Zero out position on operator display.
5. Then, jog z-axis upward and over towards the top reference face on fixture.
6. Carefully jog Z-axis down until the bottom of the tool drags on the paper resting between the bottom of the tool and the top of the fixture.
7. On the control, in the Work Offsets page, ensure that G54 Work offset Z position is highlighted, and enter the positional value. This Z value for G54 should be approximately -0.250”.
8. Once complete, you will now have G54 Work Offset X , Y and Z position accurately qualified so that this position is easily called upon in your NC program when it calls G54.

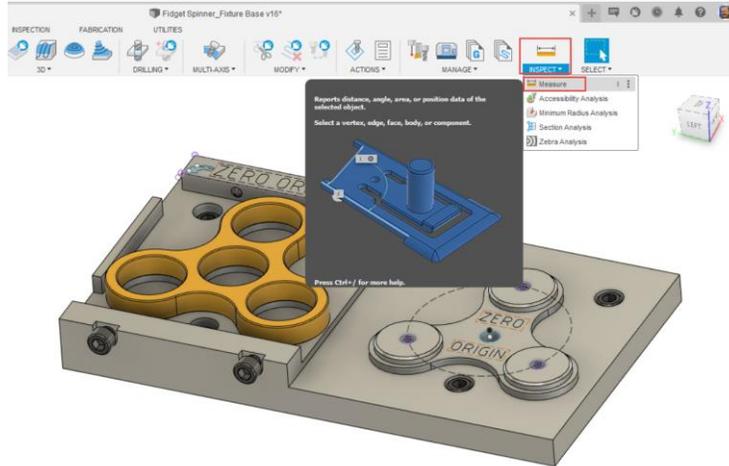
Step 5: Calculate G55 Fixture Work Offsets

The advantage of using Fixture Works offsets on a CNC control is that you can identify multiple positions within the machining envelope and easily activate them in your nc program by calling work offset G-code commands that can normally range between G54-G59.

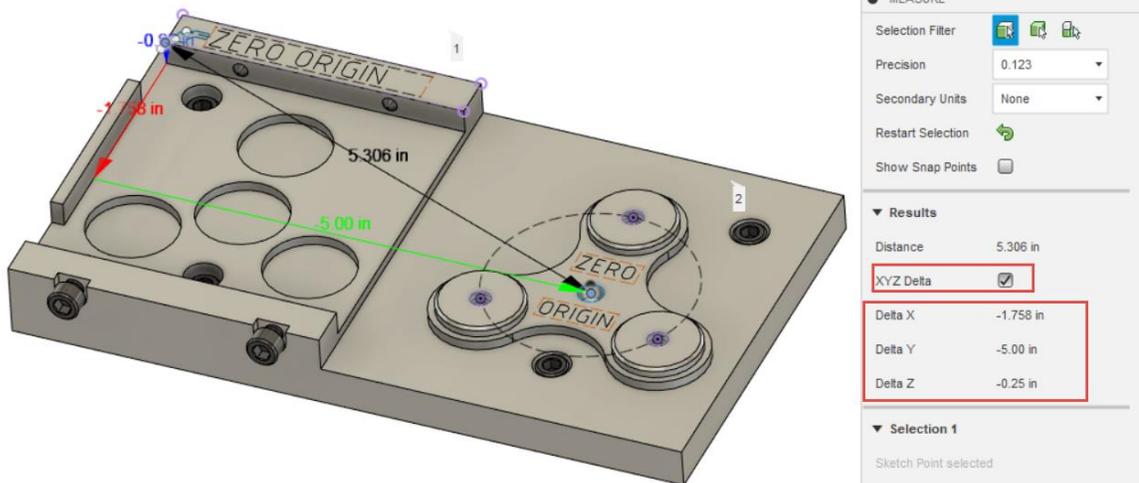


When specialty work holding fixtures are used to accurately position and securely hold parts for machining you can calculate from known datum positions on the fixture to easily define other work offset positions that can be called upon in different nc programs.

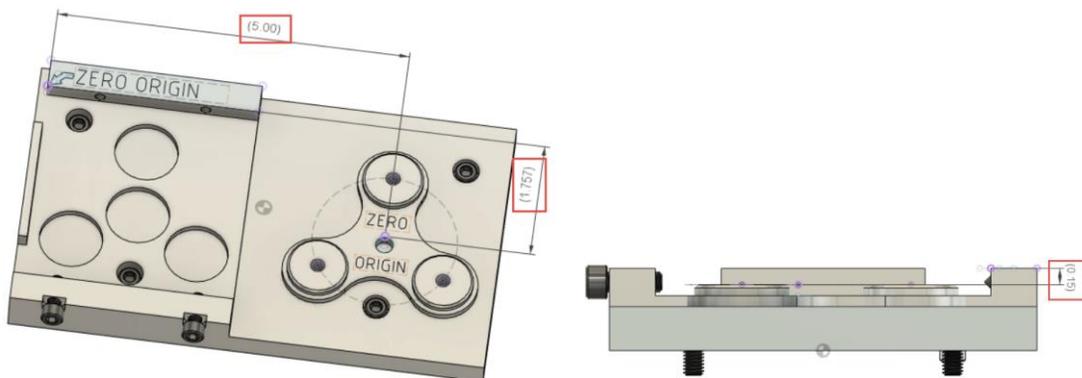
1. To calculate the difference between the G54 X/Y/Z position and the G55 X/Y/X position we can easily use the 3d CAD model fixture designed in Fusion 360.
2. Launch the Fusion 360 application and open the Fidget Spinner_Fixture Base document file.
3. In either the Design or Manufacturing workspace, you can use the Inspect, Measure command shown below.



4. With the Measure command active, ensure XYZ Delta check box is turned on.
5. Select both Zero Origin Points labelled on the fixture body.



6. Or, alternatively in the Design workspace, you could create a new sketch on the top face of the fixture component and create dimensions between both Zero Origin points as shown below.



7. You can repeat this process to identify the difference in Z distance between both Zero Origin positions as well.
8. Through either process you can identify from the CAD model geometry of the Fixture; that the difference between G54 and G55 is X 1.757, Y 5.0, and Z0.150.
9. Now having this accurate information identified, we can use these values to calculate the Work Offset position on the controller for G55.
10. On the Haas Controller, in the Work Offsets page, copy the G54 X/Y/Z values into the G55 X/Y/Z parameters to be the same.
11. Now that both G54 and G55 work offset positions are the same, now apply the CAD calculated values to the G55 X/Y/X offsets.
12. Once complete you will see that G54 has unique X/Y/Z coordinates that are relative to machine home position.
13. Likewise, the G55 X/Y/Z coordinates are relative to machine home position as well.

LESSON 9

Post Processing G-code

After completing this lesson, you will be able to:

- Understand NC Programs
- Select a Post Processor
- Adjust NC Program Settings
- Load NC Programs to Machine Control

Post Processing is when you convert the image of the toolpath on your screen, into the language for a specific NC machine. This output is usually in the form of G-Code, an alpha numeric format used to drive the machine tool. Different machines will use variations of this G-Code format for their controls. So while there is some standardization of codes, not all machines use the identical G-Code format. Because of that, Fusion 360 has a variety of Post Processors available.

Think of the Post Processor as a language translator; so you can Post Process for any single toolpath or any combination of toolpaths.

In the Fusion 360 manufacturing workspace you can post process toolpaths for additive 3d printing, milling and turning operations.



The image shows a screenshot of the Fusion 360 manufacturing workspace. On the left, the 'Setup1 Operations' tree is expanded, showing several operations: [T1] Face3, [T1] 2D Adaptive3, [T1] 2D Adaptive3 (1), [T1] 2D Contour2, [T1] 2D Contour_Bore Chamfers, and [T1] 2D Contour_Profile Chamfer. Below this, 'Setup2 Operations' is also expanded, showing [T1] 2D Adaptive3 (2), [T1] 2D Contour2 (2), and [T1] 2D Contour_Profile Chamfer (3). On the right, a G-code output window displays the following code:

```
1 %
2 O1001 (1ST SETUP TOOLPATHS)
3 (T1 D=2, CR=0.05 - ZMIN=-0.04 - BULLNOSE END MILL)
4 (T2 D=0.5 CR=0. - ZMIN=-0.695 - FLAT END MILL)
5 (T3 D=0.2491 CR=0. - ZMIN=-0.695 - FLAT END MILL)
6 (T4 D=0.312 CR=0. TAPER=45DEG - ZMIN=-0.327 - CHAMFER MILL)
7 (T6 D=0.5 CR=0. TAPER=90DEG - ZMIN=-0.1225 - SPOT DRILL)
8 (T9 D=0.125 CR=0. TAPER=118DEG - ZMIN=-0.7326 - DRILL)
9 N10 G90 G94 G17 G49 G40 G80
10 N15 G20
11 N20 G28 G91 Z0.
12 N25 G90
13
14 (FACE MILL TOP)
15 N30 T1 M06
16 (MAX DEPTH OF CUT = 0.100)
17 N35 T2
18 N40 S2500 M03
19 N45 G54
20 N50 M08
21 N60 G00 X5.68 Y-0.79
22 N65 G43 Z0.6 H01
23 N70 G00 Z0.2
24 N75 G01 Z0.16 F40.
25 N80 G18 G03 X5.48 Z-0.04 I-0.2
26 N85 G01 X5.38
27 N90 X-1.
28 N95 G03 X-1.2 Z0.16 K0.2
29 N100 G00 Z0.6
30 N105 G17
31 N110 M05
32 N115 G28 G91 Z0.
33 N120 G90
34 N125 G49
35
36 (OUTSIDE MILL)
37 N130 M09
38 N135 M01
39 N140 T2 M06
40 N145 T3
41 N150 S1500 M03
```

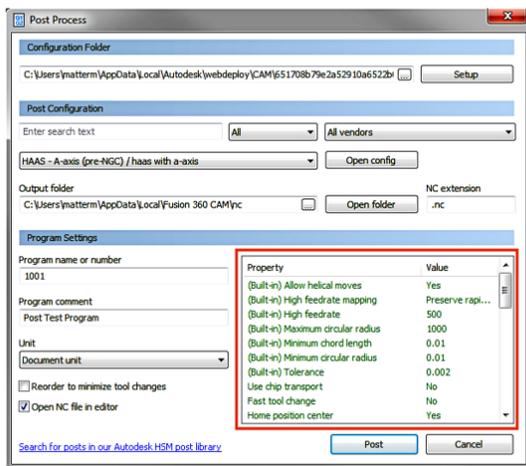
Step 1: Understanding NC Program

NC Program is the new, preferred way of creating your NC code. The old “legacy” style of Post Processing will be retired at some point in the future.

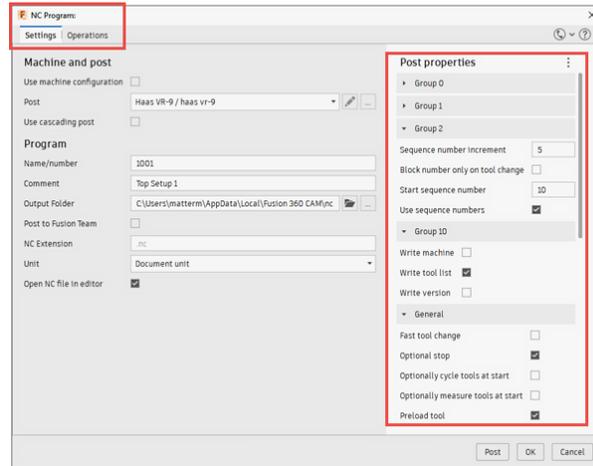
NC Program performs several tasks. All of these have to do with the output of your toolpath operations such as:

- Post Processing your operations to create the NC program for your machine tool
- Grouping multiple setups for output as a single NC program with multiple fixtures and offsets
- Creating documentation setup sheets and tool lists of your selected operations

Legacy Style Posting

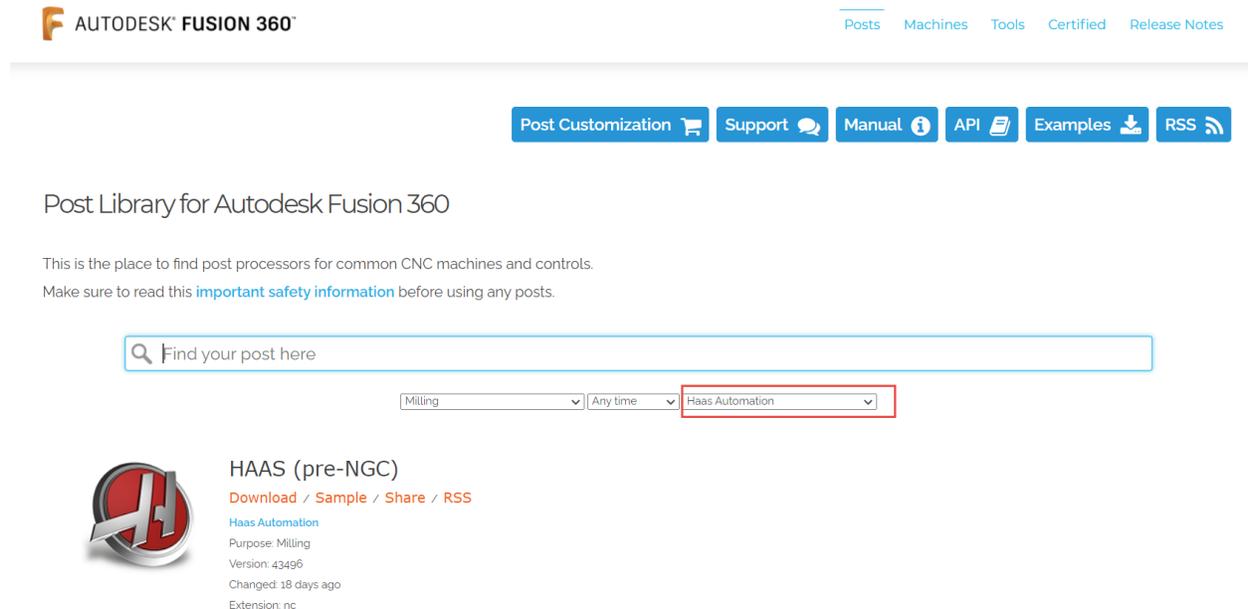


NC Program



Step 2: Selecting a Post Processor

1. For this fidget spinner project, please browse to the online Fusion 360 post library found here: <https://cam.autodesk.com/hsmposts>



2. In the Any Vendor drop down list, choose Haas Automation
3. Scroll down list, and download current Haas -Desktop Mill post processor; this post processor should download into your local download folder



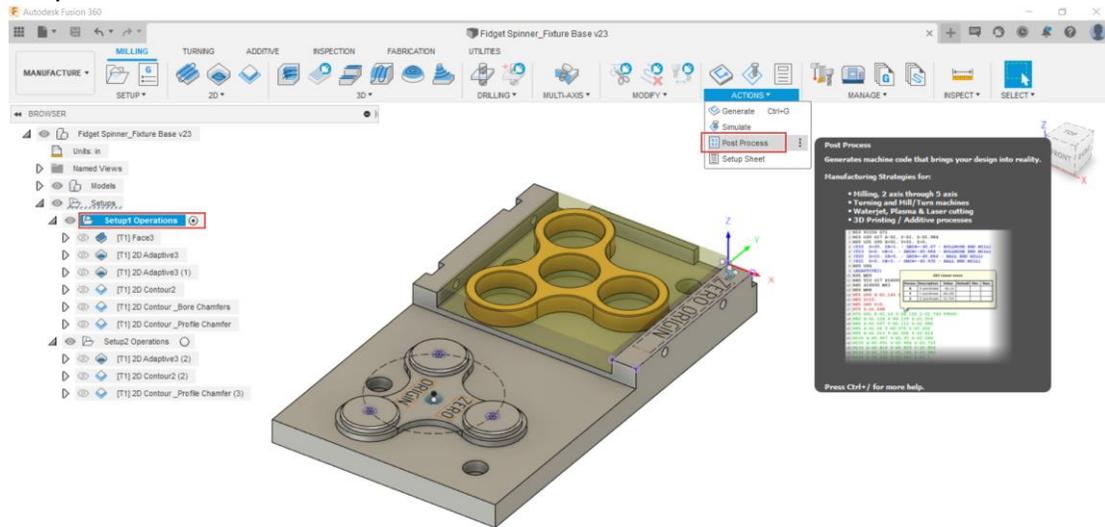
4. We will save this post into your local post directory upon post processing

Step 3: Adjusting NC Program Settings

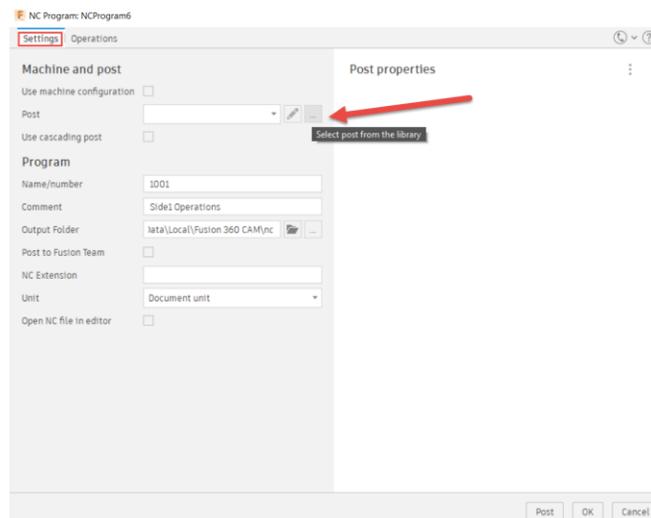
The Settings tab is where you can define presets for post processing. The items you see on this page will be determined by the machine you selected in your Setup or the Post Processor you selected.

If no Post Processor is shown, you will be prompted to select a Post from the library.

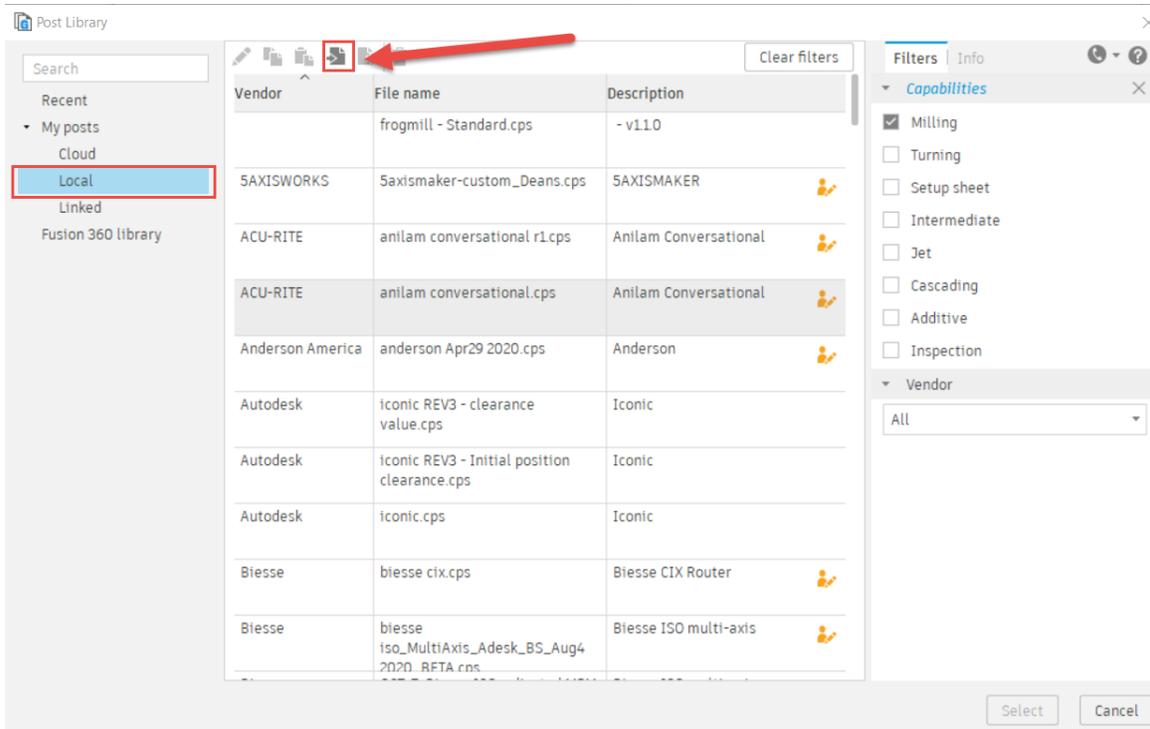
1. Right Click on Setup1 and choose Post Process or, from the toolbar, expand the Actions drop down menu and choose Post Process.



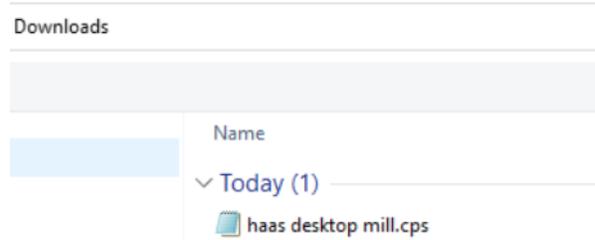
2. In the NC Program Settings parameters, proceed to select post from library as shown below.



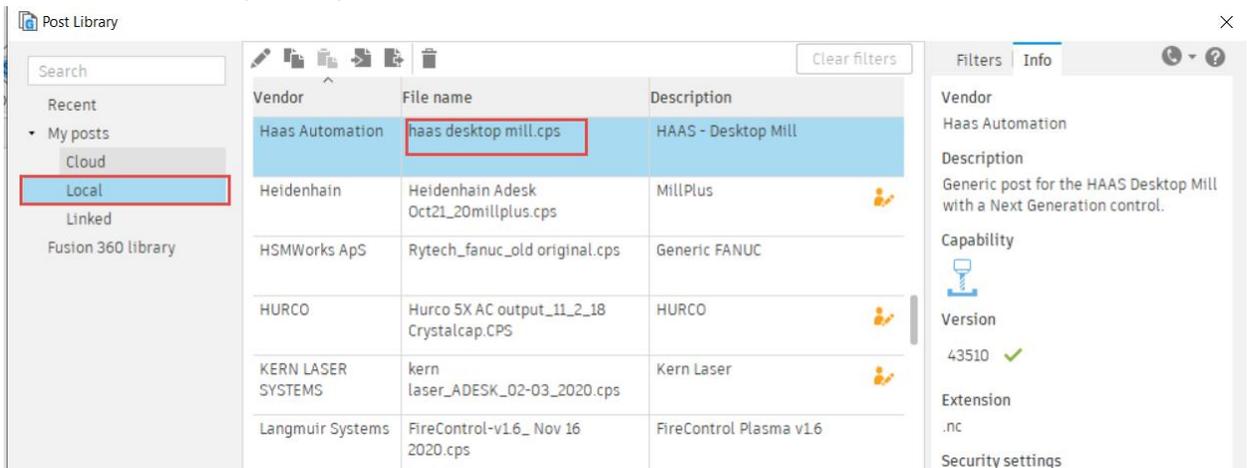
3. Select your Local libraries, and choose import.



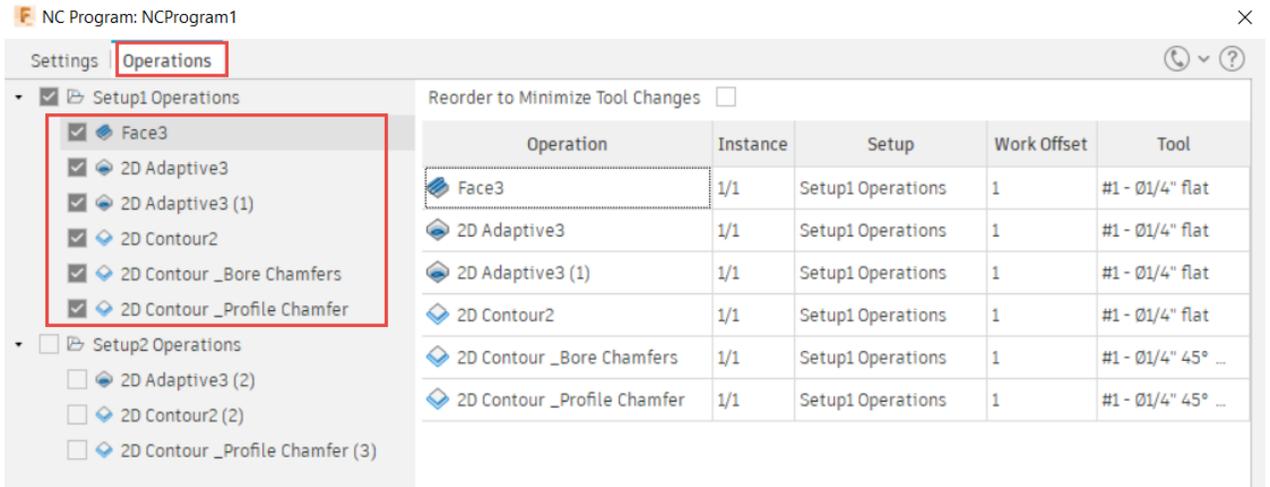
4. Browse to your local downloads folder, and choose haas desktop mill.cps post processor.



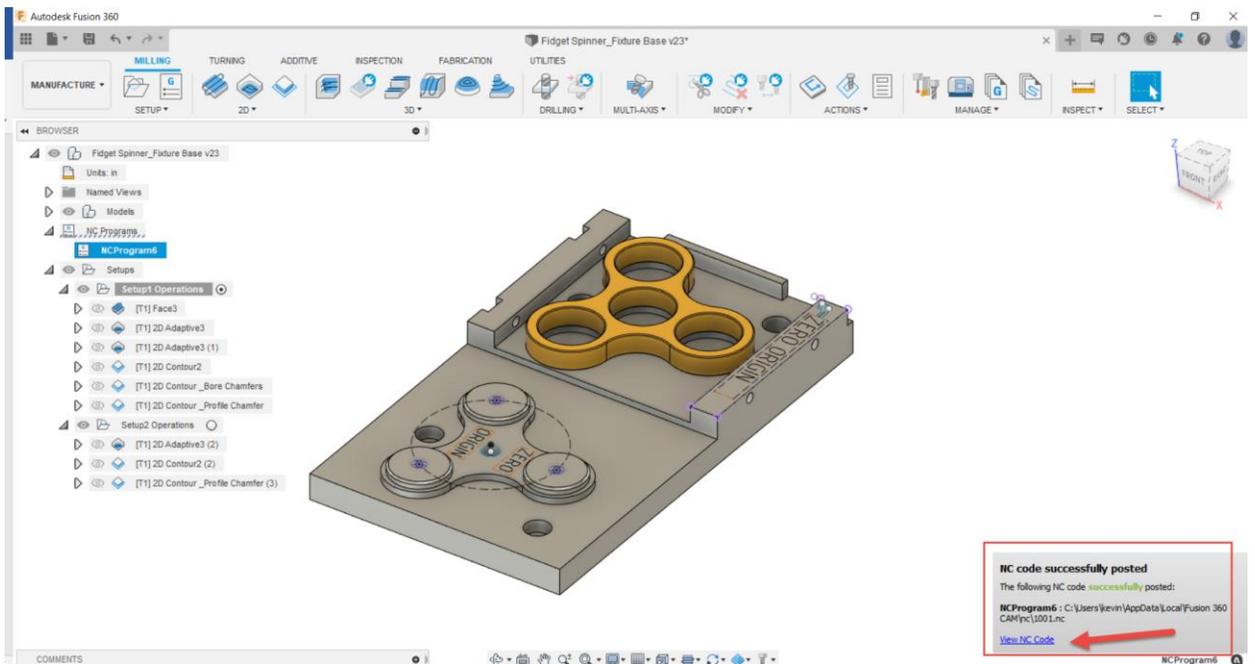
5. Select haas desktop mill.cps



- You will notice that some details update such as Name/number and Comment, which match what you confirmed when defining your CAM Setup1.
- Move into the Operations tab and ensure that only Setup 1 Operations are selected.



- Proceed with selecting Post.
- Upon confirmation of where your nc program will save, you should notice a temporary note in the lower corner of screen confirming NC code successfully posted.
- You can select the link to View NC Code.



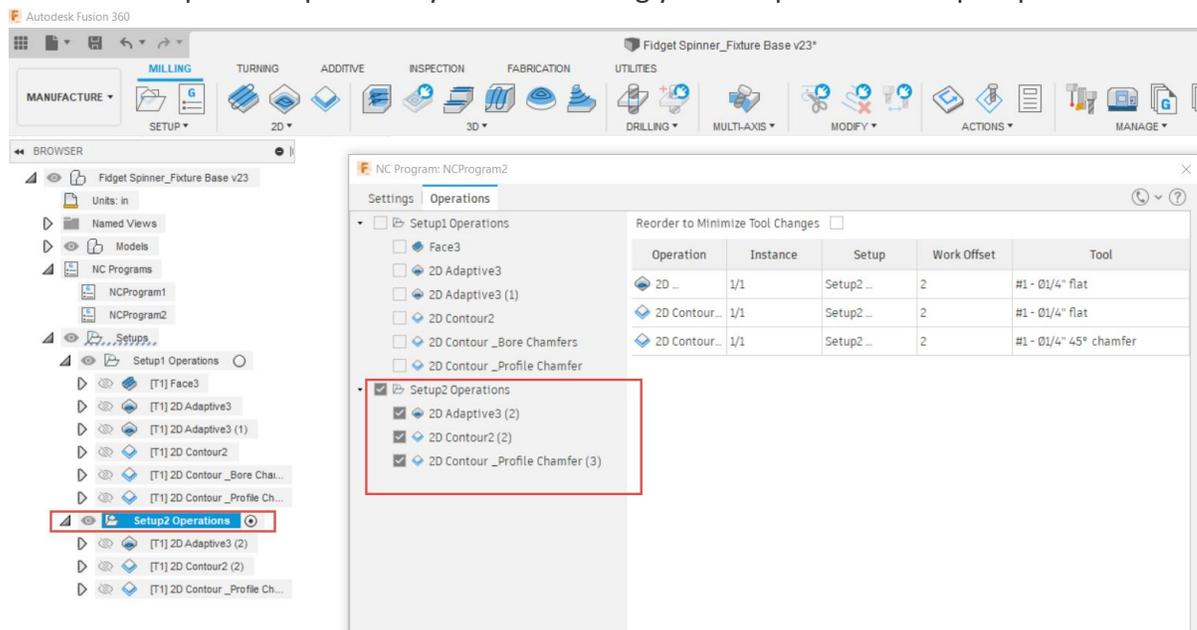
- Your NC g-code program should open inside of Visual Studio Code viewer.

```

1  #
2  O01001
3  (Using high feed G1 F3600, instead of G0.)
4  (Multiple tools disallowed. Please ensure the tool in the spindle is T1)
5  (T1 D=0.25 CR=0. - ZMIN=-0.26 - flat end mill)
6  N10 G90 G94 G17
7  N15 G20
8  N20 G53 G0 Z0.
9
10 (Face3)
11 N25 S15000 M3
12 N30 G54
13 N35 M8
14 N40 G1 X0.1628 Y-2.9354 F3600.
15 N45 G0 G43 Z0.725
16 N50 G0 Z0.325
17 N55 G1 Z0.075 F20.
18 N60 G18 G3 X0.1376 Z0.05 I-0.025 K0.
19 N65 G1 X0.0001
20 N70 X-2.9999 F10.
21 N75 G17 G2 Y-2.7477 I0. J0.0938
22 N80 G1 X0.0001
23 N85 G3 Y-2.5601 I0. J0.0938
24 N90 G1 X-2.9999
25 N95 G2 Y-2.3724 I0. J0.0938
26 N100 G1 X0.0001
27 N105 G3 Y-2.1848 I0. J0.0938
28 N110 G1 X-2.9999
29 N115 G2 Y-1.9972 I0. J0.0938
30 N120 G1 X0.0001
31 N125 G3 Y-1.8095 I0. J0.0938
32 N130 G1 X-2.9999
33 N135 G2 Y-1.6219 I0. J0.0938
34 N140 G1 X0.0001
35 N145 G3 Y-1.4342 I0. J0.0938
36 N150 G1 X-2.9999
37 N155 G2 Y-1.2466 I0. J0.0938
38 N160 G1 X0.0001
39 N165 G3 Y-1.059 I0. J0.0938
40 N170 G1 X-2.9999

```

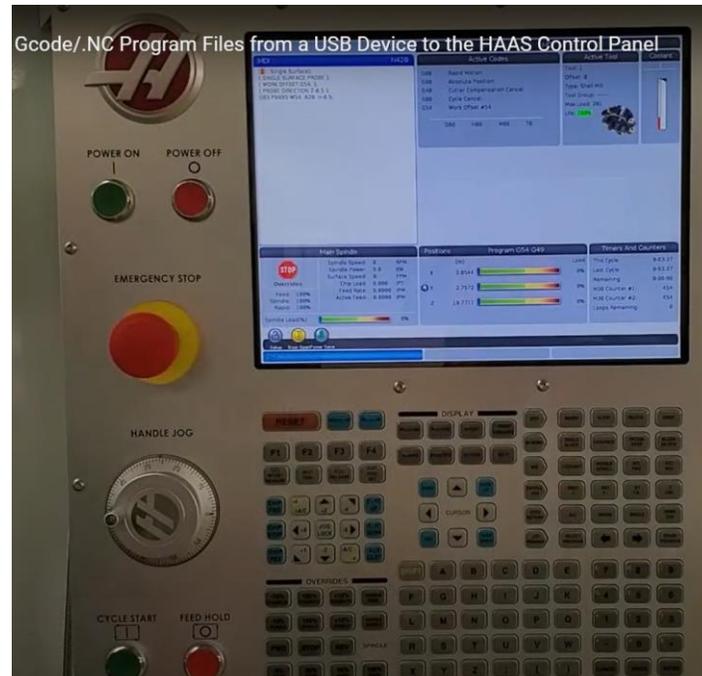
12. It's always safe practice to review Gcode if possible, to ensure both beginning and ending program structure is safe and ready for use.
13. Choose file Save As, and save onto a usb key, or network location where you easily can access your NC g-code program. Save your program as 1001.nc
14. Proceed to repeat this process by Post Processing your toolpaths for Setup2 Operations



15. Choose file Save As, and save onto a usb key, or network location where you easily can access your NC g-code program. Save your program as 1002.nc

Step 4: Loading NC Programs to Machine Controller

Your Haas Desktop CNC milling machine comes with a standard Haas cnc machine controller.



1. Insert your USB device into side of your Haas control panel
2. Proceed to select List Program Button



3. Use the cursor keys to scroll up and over to the USB tab and down to the program your interested in.
4. Select nc program 1001.nc and copy program to controller memory
5. Repeat this process for nc program 1002.nc and copy program to controller memory
6. With caution, proceed to cycle start program 1001.nc followed with 1002.nc

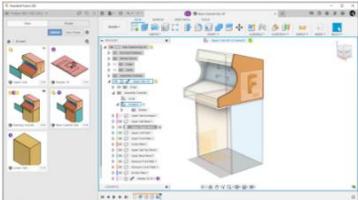
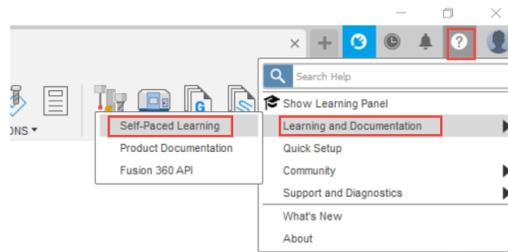
Additional Learning Resources

- Self-paced Learning for Fusion 360
- Autodesk Learning Partner Faculty Training Workshops
- Autodesk Fusion 360 Certifications
- Haas Operator Certification Program

Step 1: Self-paced learning pathways for Fusion 360

Get started with self-paced learning pathways that include videos, step-by-step tutorials, and downloadable 3D models to help you learn Fusion 360 at your own pace.

In Fusion 360, select the help icon in top right corner, choose Learning and Documentation, then Self-Paced Learning.



Introduction to Fusion 360

- Video series
- 10 videos
- Beginner

Highlights the powerful capabilities of Fusion 360, and introduces key features to help you get the most out of Fusion 360.

Start learning

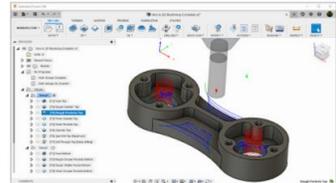


Introduction to 3D modeling

- Video series
- 18 videos
- Beginner

This beginner-level CAD video series focuses on the transition from 2D to 3D design. Whether you're a designer, engineer, machinist, maker, artist, or just curious, this video series helps you take your first steps towards professional 3D modeling.

Start learning



Milling basics

- Video series
- 17 videos
- Beginner

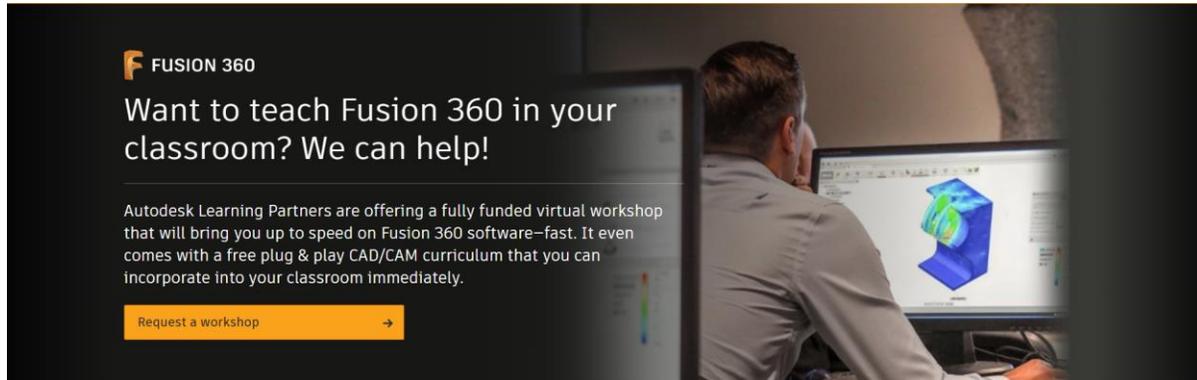
This video series is designed to give you quick answers to common manufacturing questions. These videos are laid out as a process, but you can jump ahead for information on any specific task.

Start learning

<https://help.autodesk.com/view/fusion360/ENU/courses/>

Step 2: Autodesk Learning Partner Faculty Training Workshops

Autodesk Authorized Training Centers and Authorized Academic Partners deliver quality professional training from Autodesk Certified Instructors and industry experts to students, educators and customers.



"The workshop showed me a much easier way to approach teaching my students about design for manufacturability. I am excited to schedule more sessions with my colleagues!"
—Jeritt Williams, Engineering Technology Faculty, Illinois State University

Fully funded personalized virtual workshop with plug and play curriculum

We have some good news for educators who are looking for new ways to improve distance learning. Autodesk Learning Partners are offering a fully funded virtual workshop that will bring you up to speed on Fusion 360 software—fast. It even comes with a free plug and play CAD/CAM curriculum. This will allow you to provide your students with a solid grounding in CAD/CAM fundamentals, plus manufacturing and engineering skills. In this personalized workshop, where you can invite your peers, an Autodesk Learning Partner expert will provide:

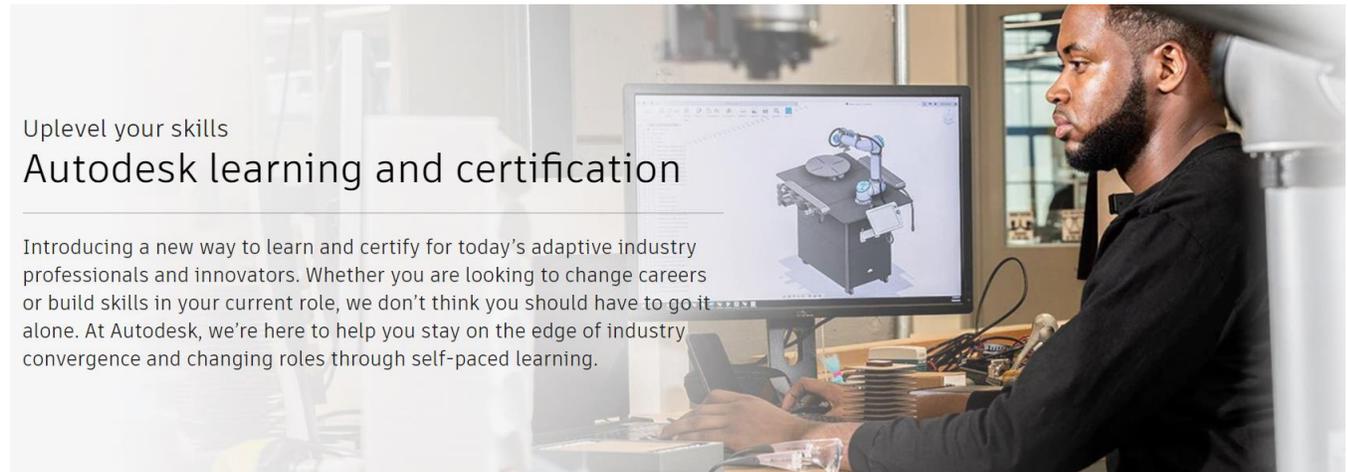
- An introduction to Autodesk Fusion 360 and its cloud-based collaboration features
- Specific curriculum that can augment your existing coursework
- Ways to distinguish between traditional task-based mindsets and holistic engineering

The Fusion 360 software, the curriculum, and the workshop are all provided by Autodesk at no cost to your school, your colleagues, and your students. Finally, when you've finished your half-day workshop, you'll have a dedicated partner to guide you and answer any questions that come up. Don't miss out on this opportunity!

<https://www.autodesk.com/campaigns/education/workshop-registration>

Step 3: Autodesk Fusion 360 Certifications

Autodesk offers certifications that map to your skill level and career goals allowing you to earn recognition for your expertise, accelerate your career, and continue your lifelong learning journey.



Uplevel your skills **Autodesk learning and certification**

Introducing a new way to learn and certify for today's adaptive industry professionals and innovators. Whether you are looking to change careers or build skills in your current role, we don't think you should have to go it alone. At Autodesk, we're here to help you stay on the edge of industry convergence and changing roles through self-paced learning.



Autodesk Certified Associate in CAD for Mechanical Design

Use your CAD skills and knowledge of rapid prototyping methods to earn your certification in CAD for mechanical design.



Autodesk Certified Associate in CAM for 2.5 Axis Milling

Put your expertise into practice with a certification in CAM for 2.5 axis milling. Prove your knowledge and skills for programming simple parts as well as operating CNC milling and lathe machines.



Autodesk Certified Expert in Generative Design for Manufacturing

Show your mastery of expert-level skills in defining and running generative design studies for manufacturing applications with a certification in generative design in manufacturing.

<https://www.autodesk.com/certification/overview>

Step 4: Haas Operator Certification Program

Team up with Haas Automation and stand out in the industry as a qualified CNC operator, or ensure your team has the skills they need to properly and safely operate a CNC machine – by becoming Haas Certified.

Haas’s online CNC Certification Program provides you, or your team members, with the knowledge and skills necessary for basic CNC machine operation.



Start the Haas CNC Certification Program

GET CERTIFIED
Stand out in the manufacturing industry.



For New Operators

- ✓ Stand out in the industry
- ✓ Prove your knowledge
- ✓ Professional credibility

<https://learn.haascnc.com/>