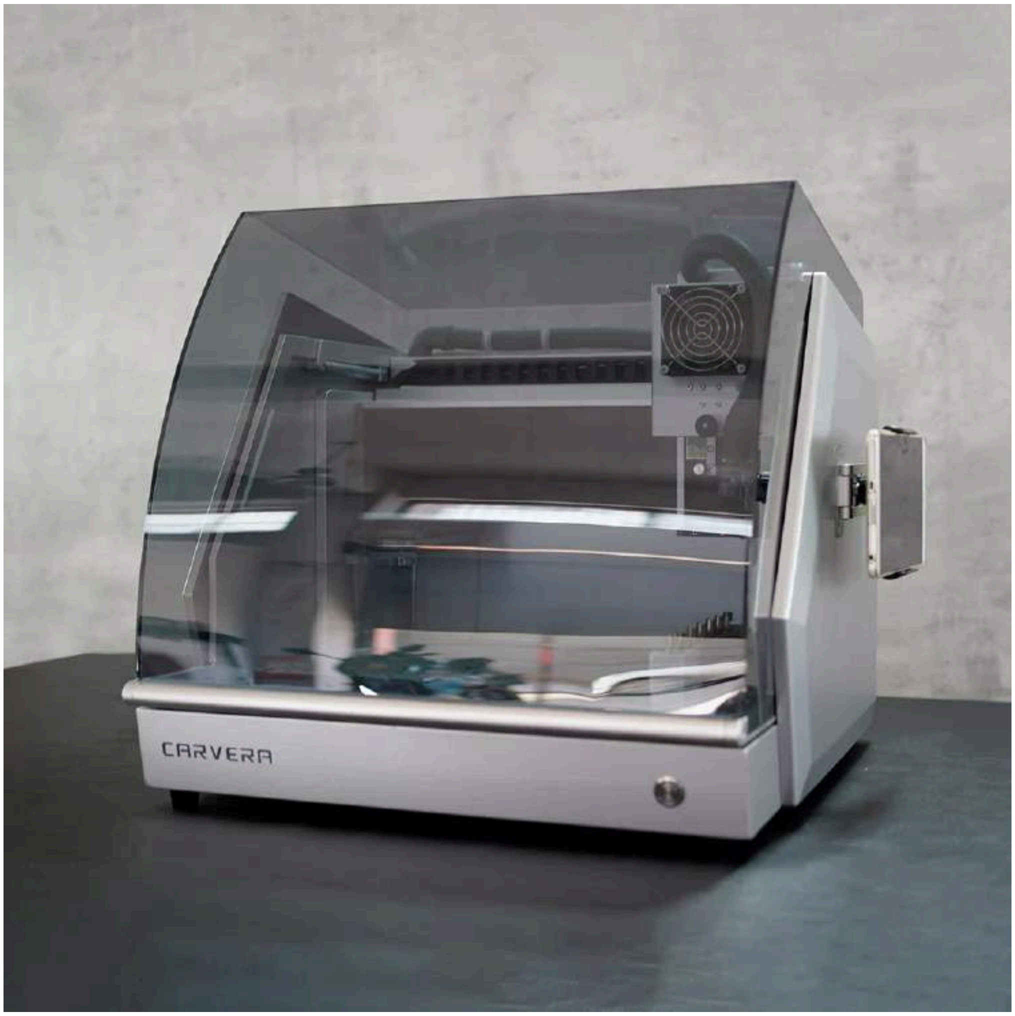
Makera Carvera



April 2024

Included Items:

- Tool Kit

- Manual Tool Setter
- Laser Goggles
- Safety Goggles
- Emergency Stop Button

Disclaimer

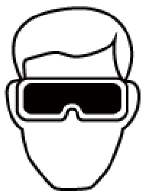
Please read this manual carefully before using the Carvera. Failure to read the manual may lead to personal injury, inferior results, or damage to the Carvera products.

This manual is provided for reference purposes only

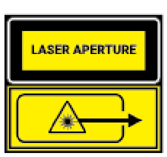
Carvera is equipped with multiple sensors (Motor stall sensor, Spindle abnormal sensor, etc.). However, you should always pay attention in operation that milling cutters revolving in high-speed, unstable components or laser in operation are dangerous to people without protection. Please carefully read the safety instructions below to avoid unnecessary damage to the machine or injury to the user.

Safety Information

1. Always wear safety goggles when operating Carvera, especially when the protective cover is open.
2. Always wear laser protection goggles when using laser function.
3. Please wear hearing protection when Carvera is machining on the hard materials.
4. Do not put your hands close to the spindle or machining area. Keep the protective cover closed when operating Carvera.
5. Do not leave your Carvera unattended while it is machining.
6. Please be aware of the sharpness of the milling bits during installation, dust collection, and other operations.
7. Milling and laser carving will generate heat. Inappropriate parameters will cause fire hazards. Make sure an extinguisher is in your vicinity.
8. Some materials are harmful to people when machining or laser craving, such as carbon fiber and epoxy resin. Please wear a face mask and turn on the automatic dust collection.
9. Do not expose this machine to rain or wet conditions.
10. Keep children and bystanders away while operating this machine. It requires supervision and the assistance of an adult when children use this machine.

<table>
<tr><th>Safety Labels</th><th>Meaning</th></tr>
<tr><td></td><td>Keep hands clear of moving parts: machine axis, spindle, etc.</td></tr>
<tr><td></td><td>Wear safety goggles when operating the CNC function or laser goggles when operating the laser.</td></tr>
<tr><td></td><td>Caution sharp cutters when installing milling bits and doing dust collection.</td></tr>
<tr><td>

IEC 60825


FDA</td><td>Laser radiation - Class 4 laser product, avoid eyes or skin exposure to direct or scattered radiation.</td></tr>
<tr><td>

IEC 60825


FDA</td><td>Laser aperture - Laser radiation is emitted from this aperture.</td></tr>
</table>

Video Tutorials

The Makera Youtube channel has video tutorials on almost every aspect of the Carvera, these come highly recommended to watch before or during your use of the machine, including how to use CAM softwares such as Fusion with the Carvera

Tool Preparation

For easy transportation, the tools are not loaded by default. Please put the wireless probe and milling bits into right positions as shown in the figure. You can find the wireless probe in the accessory box, milling bits in the tool box and the optional solder mask removal tool in the PCB pack. The default tool installed is a test rod (No 6 tool).

This 1-6 tool positions are for the examples. If you run your own toolpath in the future, you should layout your tools accordingly.

While you can put another tool in tool spot 6, it is not recommended as the test rod is used with the XYZ probe.

It is crucial to know the exact tool that is in which slot, within your CAM software you can designate a tool to a certain position in the ATS, incorrectly locating or numbering a tool in your CAM software can lead to the wrong tool being selected and causing damage to your stock material, the bit, and possibly the machine.

If the machine picks up the wrong tool for your operation hit the red stop button immediately double check your tool numbering in your CAM software.

Adjust Dust Shoe

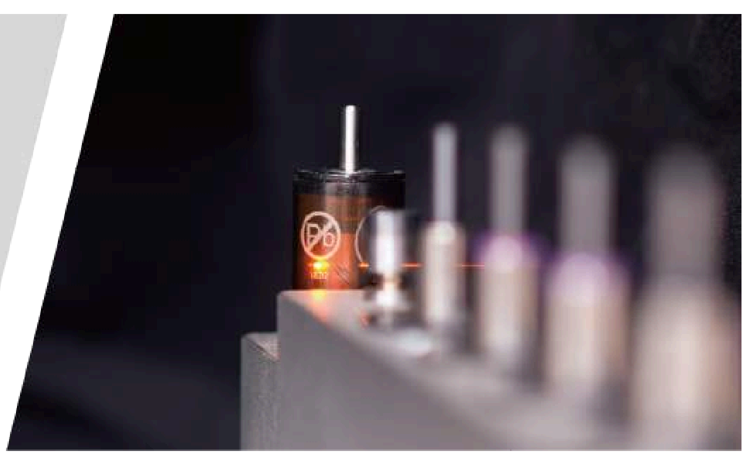
The default state of the dust shoe is locked. To use the vacuum system correctly you need to unlock the dust shoe first. Pull the plug outwards, rotate it for about 30 degree, make sure it is unlocked, thus the dust shoe can go up and down freely (Locking and unlocking modes are toggled every 30 degrees of rotation).

Charge Wireless Probe

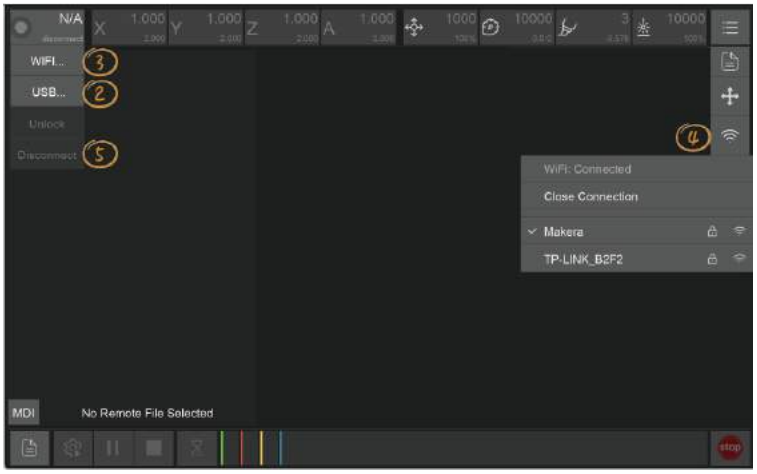
The wireless probe is one of the main components of Carvera. It is an indispensable component for Z-axis tool setting and automatic leveling. Please fully charge it before using it for the first time.

1. After the device is turned on, the yellow indicator light will light up as the wireless probe automatically starts charging.

2. Please charge the wireless probe for at least 30 minutes or the yellow indicator light runs out which means it’s fully charged. If the machine is used frequently (at least once a week), then there is no need to charge it every time.



Connecting to the machine



Connect via USB: Connect the USB cable and open the Carvera control software. Click the status button in the status bar, click “USB...”, you will see your USB device and choose one to connect. Then the status button will be updated, displaying machine is connected in USB mode.

Test Auto Tool Changer

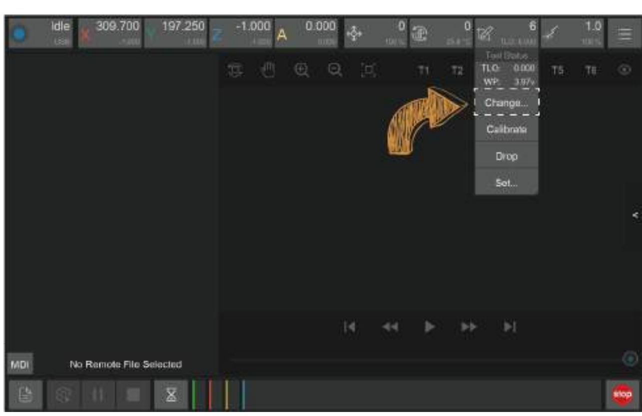
Auto tool changing is one of the key features of Carvera and also the base of all the automated function. Please test it at the first time.

1. Open the control software and connect to Carvera.

2.Change to tool 1. The tools that pre-installed in the machine is tool 6. Click tool status; click “Change…”; choose “Tool: 1”, it will automatically change to tool 1. Please observe the process to see if there is any error.

3.Change to wireless probe. If no error occurred, then test wireless probe. Click tool status; click “Change”; select “Probe”, it will automatically change to wireless probe and check its signal. Please observe the process to see if there is any error.

4.Change back to tool 1. Please follow step 2.



Compatible CAM software

We continually expand our collection of CAM software profiles to cater to our customers' requirements. Here are some that we have already developed.

Software	Description	Video Guide
Fusion 360	Fusion 360 is a cloud-based CAD/3D modeling, CAM, CAE, and PCB software platform for both product design and manufacturing. Compare to other industrial software, Fusion 360 is relatively easy to learn and is an all-in-one solution. If you are new to CAD and CAM, and you are interested in making mechanical/ electronic/robot/drone parts or something like them, Fusion 360 is a good option.	
VCarve Desktop	VCarve Desktop gives you the power to produce complex 2D patterns with profile, pocket, drill, and inlay toolpaths, plus gives you the ability to create 3D and rotary relief by importing 3d model files. It can also do laser engraving with an add-on module. VCarve Desktop is suitable for woodworking, jewelry making, art making, etc.	
Kiri: Moto	Kiri:Moto is free, open-source, browser-based application that bridges 3D design to fabrication with tools for CAM, 3D Printing, and Laser output. There is no software to install, and all data and processing happen privately and locally in the browser window. Import STL, OBJ, and 3MF files and output GCode or SVG. 3 and 4-axis CAM operations include: roughing, pocketing, contouring, tracing, and more. Preview and animate your jobs ahead of time to see what you will get before the milling begins.	

Software	Description	Video Guide
LightBurn	LightBurn is a layout, editing, and control software for your laser cutter. With LightBurn you can: - Import artwork in a variety of common vector graphic and image formats. - Arrange, edit, and even create new vector shapes within the editor. - Apply settings like power, speed, number of passes, cut order, brightness & contrast, dithering mode, and much more.	
SolidCAM	SolidCAM is an integrated CAM software for CNC applications, offering advanced milling and turning tools while seamlessly integrating with SolidWorks.	
SolidWorksCAM	SolidWorks CAM is a fully integrated, knowledge-based technology for automating design and manufacturing processes within the SolidWorks environment.	
MasterCAM	MasterCAM is a versatile CAM software solution, offering advanced 2D and 3D design tools, precision machining, and efficient programming capabilities.	

Speeds & Feeds

The following recommended parameters are based on current tests. The machining speed with small diameter tools/hard materials should be slow and fast in opposite. We will conduct more tests and provide more detailed parameter recommendations in the future on our website.

Please start the test from the lower limit of the parameter, and adjust them based on the test results.

Material	**Tools**	**Milling depth (mm)**	**Feed Speed (mm/min)**	**Plunge speed (mm/min)**	**Spindle speed (RPM)**
PCB	V-bit	0.1	200-500	200	12000
Wood	Single flute spiral bit	0.5-2	500-1000	300	10000
Plastic	Single flute spiral bit	0.5-2	500-1000	300	10000
Carbon Fiber/ Glass	Corn bit	0.3-0.5	500-1000	300	10000
Aluminum/ copper	Single flute spiral bit for metal	0.1-0.2	300-500	200	12000

Software Introduction

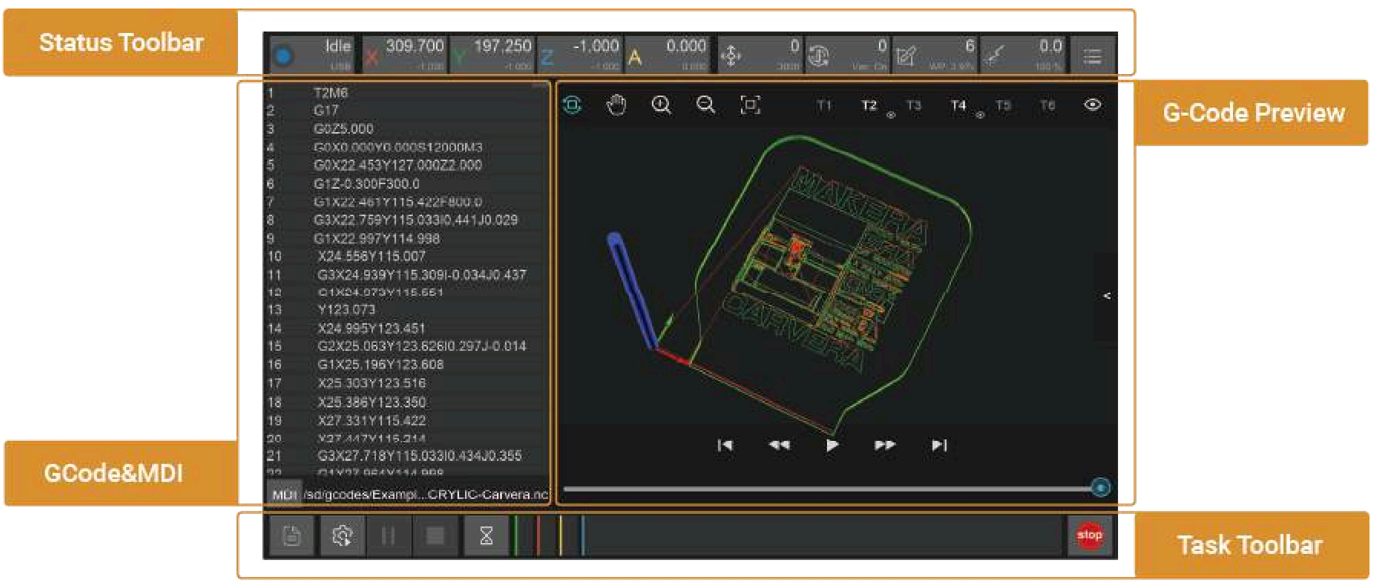
Overview

1. Status Toolbar

The status toolbar is on top of the interface. It shows the real-time data of key indicators and can be used to control these indicators.

2. Task Toolbar

Task toolbar is at the bottom of the interface. In the task toolbar, you can manage G-Code files, configure, track and control machining process.



3. G-Code & MDI

By default, this interface displays the G-Code of the currently opened file. It can be switched to the command information sent/received by the machine.

4. G-Code Preview

The G-Code preview graphically displays the currently opened file G-Code.

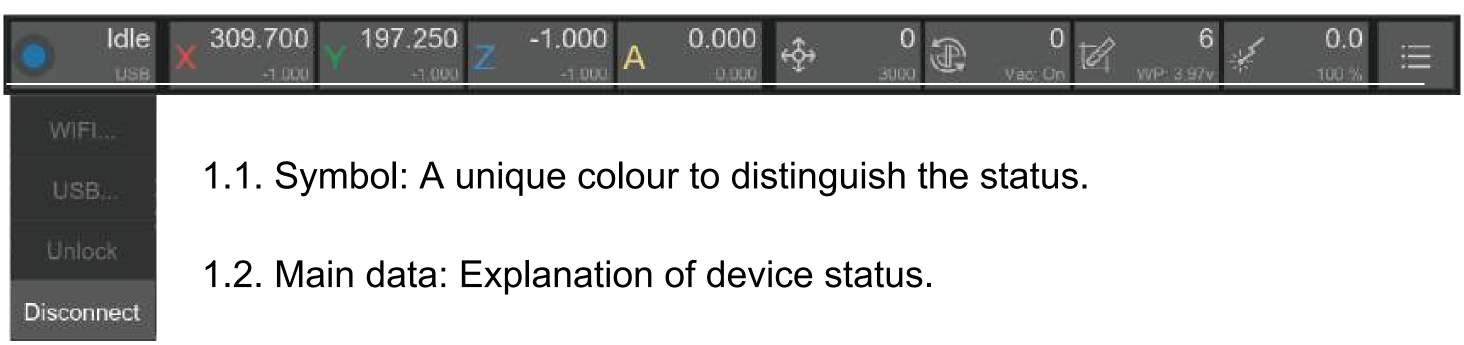
5. **Manual Operation**

Manually control Carvera's movement and execute other commands. Because Carvera does most of the jobs automatically, the manual control interface is hidden by default. Click the arrow on the right side of the interface to switch whether to display it.

Status Toolbar

All indicators include 3 items: a symbol, main data and sub-data. Click the button can open the corresponding drop-down list.

1. **Machine status and control**



1.1. Symbol: A unique colour to distinguish the status.

1.2. Main data: Explanation of device status.

1.3. Sub-data: Current connection mode.(No connection, WiFi or US0B)

1.4. Drop-down list:

1.4.1. WIFI: Connecting Carvera via WiFi.

1.4.2. USB: Connecting Carvera via USB.

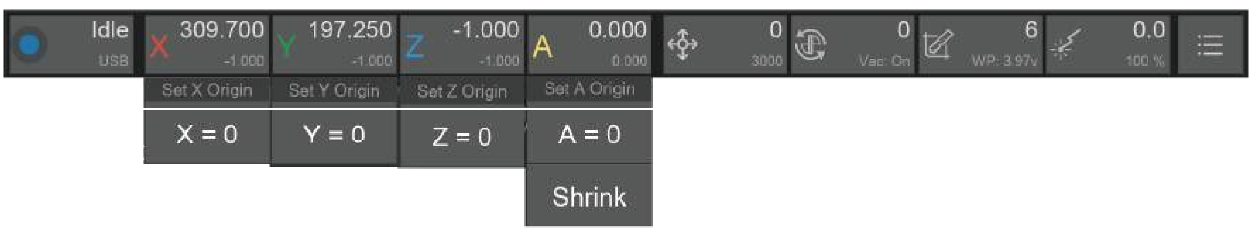
1.4.3 Unlock/Reset: Unlock or reset Carvera.

1.4.4. Disconnect: Disconnect Carvera from your device.

Explanation of different Carvera statuses :

Colour	Status	How it been triggered/How to quit
	Idle	Carvera is idle
	Run	Carvera is working
	Alarm	Carvera has an alarm/Unlock it to restore
	Home	Carvera is resetting the coordinate
	Hold	Click the hold button/Click again to resume
	Wait	Carvera is emptying the buffer
	Disable	No device has been connected
	Sleep	Timeout/Reboot Carvera to restore
	Pause	Click the pause button/Click again to resume

2. Coordination status and control



2.1. Symbols: X/Y/Z/A

2.2. Main data: Work coordinate (The position of the tool relative to the work zero point). The position depends on where you put your workpiece and where you want to start on the workpiece.

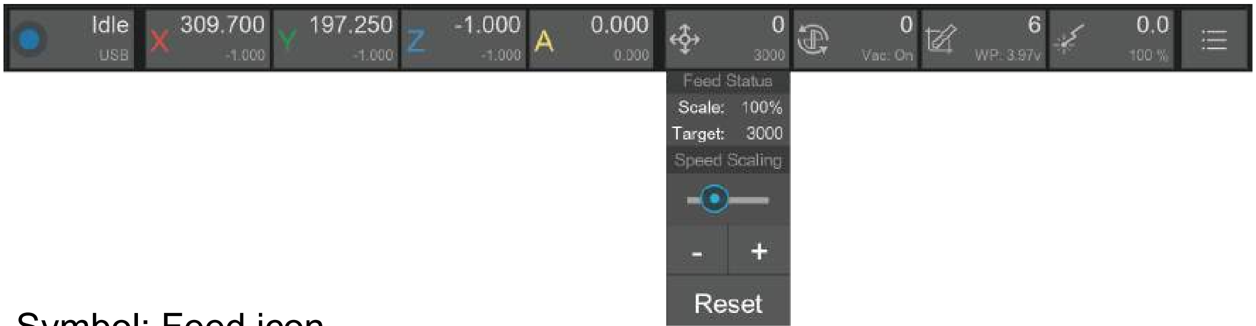
2.3. Sub-data: Machine coordinates (The position of the tool relative to the machine's zero point). The position is fixed and located in the upper right corner of the machine, so the coordinates are generally negative.

2.4. Drop-down list:

2.4.1 Set Origin: The working coordinates of four axes - X/Y/Z/A can be set to zero. Carvera usually uses 2 fixed anchor points to set the starting point automatically, so we do not recommend setting the origin point by yourself.

2.4.2 Rotary axis (A axis) does not distinct work coordinates and machine coordinates, and there is a "Shrink" function in the drop-down list, which can calculate the remainder according to 360 when having large rotation angles.

3. Feed status and control



3.1. Symbol: Feed icon

3.2. Main data: Real-time feeding speed

3.3. Sub-data: Target feeding speed/Feeding speed scale(moving message)

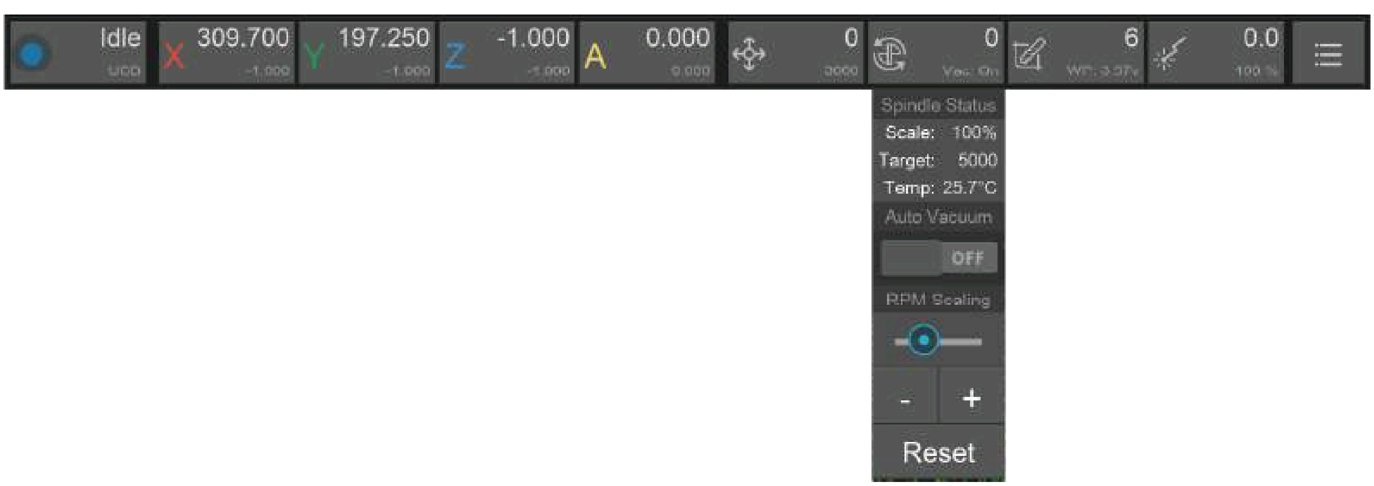
3.4. Drop-down list:

3.4.1. Feed Status: Target feeding speed /Feeding speed scale

3.4.2. Speed Scaling: Set the feed rate by percentage. For safety reasons, the adjustment range is limited to 50% to 200%

Note: Adjusting the feed rate will not take effect immediately. Generally, it will take a few seconds to wait for the current command to finish.

4. Spindle status and control



4.1. Symbol: Spindle icon

4.2. Main data: Real-time spindle rotary speed (RPM)

4.3. Sub-data: Target spindle rotary speed/ Spindle speed scale/Real-time spindle temperature (moving message)

4.4. Drop-down list:

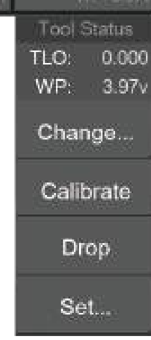
4.4.1. Spindle Status: Target spindle rotary speed/ Spindle speed range /Real-time spindle temperature summary display

4.4.2. Auto Vacuum: Choose to turn on/off auto vacuum while the spindle is rotating. Default - on.

4.4.3. RPM Scaling: Set the spindle speed range by percentage. For safety reasons, the adjustment amount range is limited from 50% to 200%.

Note: Turn on the Vacuum or adjust the spindle rotary speed will not take effect immediately. Generally, it will take a few seconds to wait for the current command to finish.

5. Tool status and control



5.1. Symbol: Tool icon

5.2. Main data: Display the number 1 to 6 of the current tool on the spindle, no tool - “None”, wireless probe -“Probe”

5.3. Sub-data: Current Tool Length Offset（TLO）/Wireless Probe Power(moving message)

5.4. Drop-down list:

5.4.1 Tool Status: TLO/Wireless Probe Power summary display

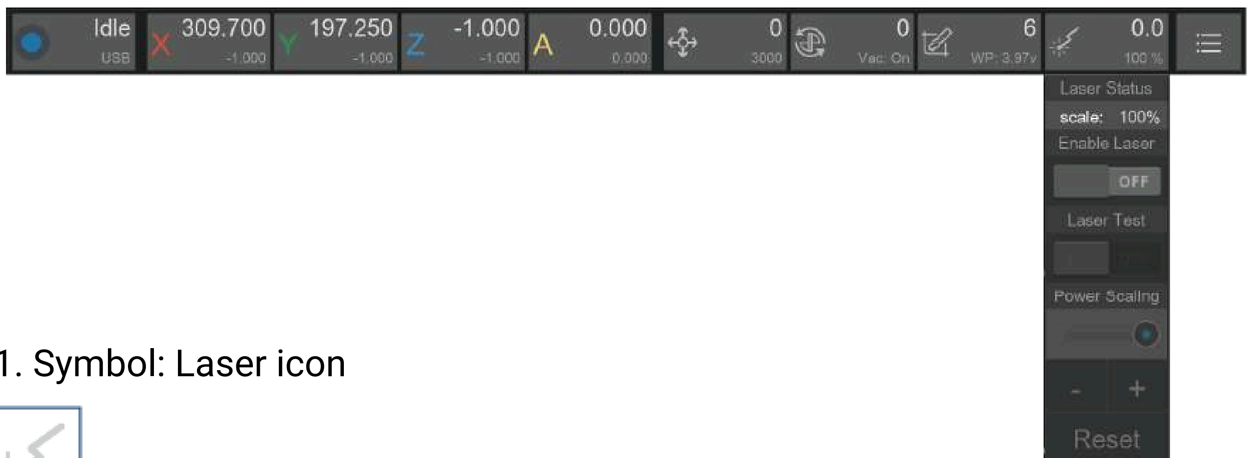
5.4.2. Change tool: Change to the selected tool or wireless probe, and perform automatic calibration.

5.4.3. Calibrate tool: Automatically calibrate the current tool and set TLO.

5.4.4. Drop tool: Drop the current tool.

5.4.5. Set tool: Manually set the current tool number. Only use when the tool number is wrong.

6. Laser status and control



6.1. Symbol: Laser icon

6.2. Main data: Current laser rate 6.3. Sub-data: Laser power scale 6.4. Drop-down list:

6.4.1. Laser Status: Laser power scale

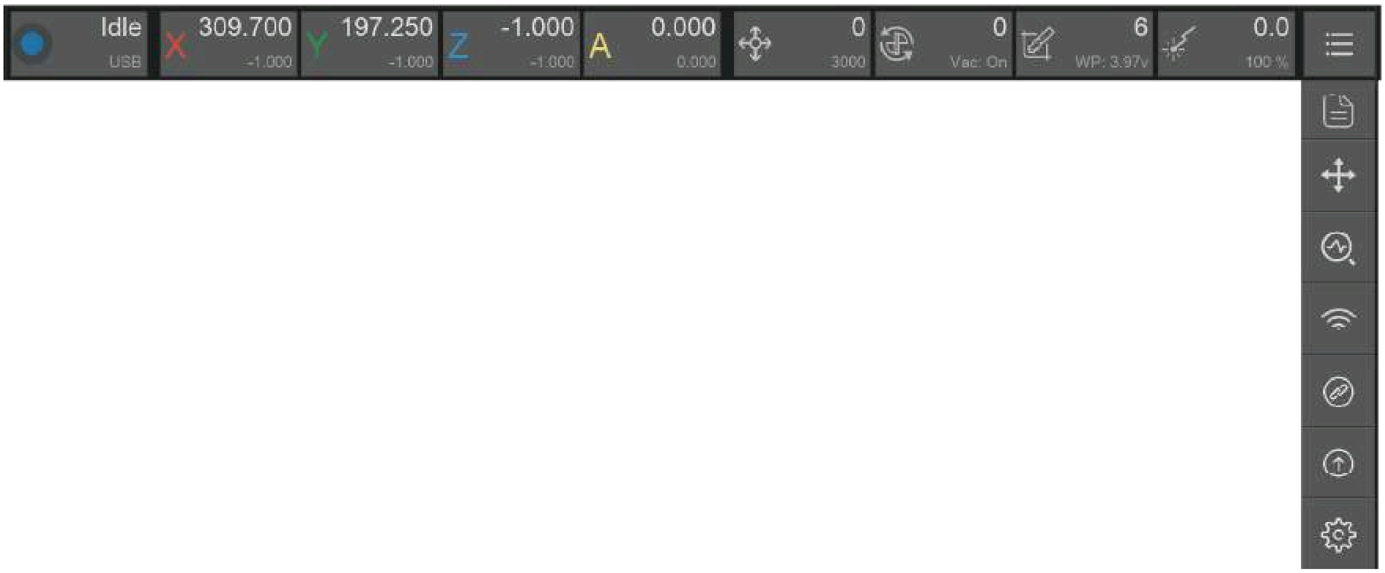
6.4.2. Enable Laser: Switch to laser mode. The working coordinates will be automatically updated according to the preset offset. If there is a tool on the current spindle, Carvera will drop it first and calibrate again to check the laser head’s height.

6.4.3. Laser Test: Performing a laser test after turn on the laser mode will trigger a low-power laser beam for focus calibration.

6.4.4. Power Scaling: Set the laser power range by percentage. For safety reasons, the adjustment amount range is limited from 50% to 200%.

Note: Carvera has already set the coordinate offset of the laser before delivery, only reset it when the coordinates deviate. We will add relevant tutorials in the online instructions.

7. Other functions



7.1. Manual control: Same function as the arrow button on the right side of the interface. Switch to display manual control interface and file preview interface .

7.2. Device status diagnose: Check and your Carvera in detail and debug. No need to use it when Carvera runs in good condition. We will add a tutorial in the formal version of the instruction manual.

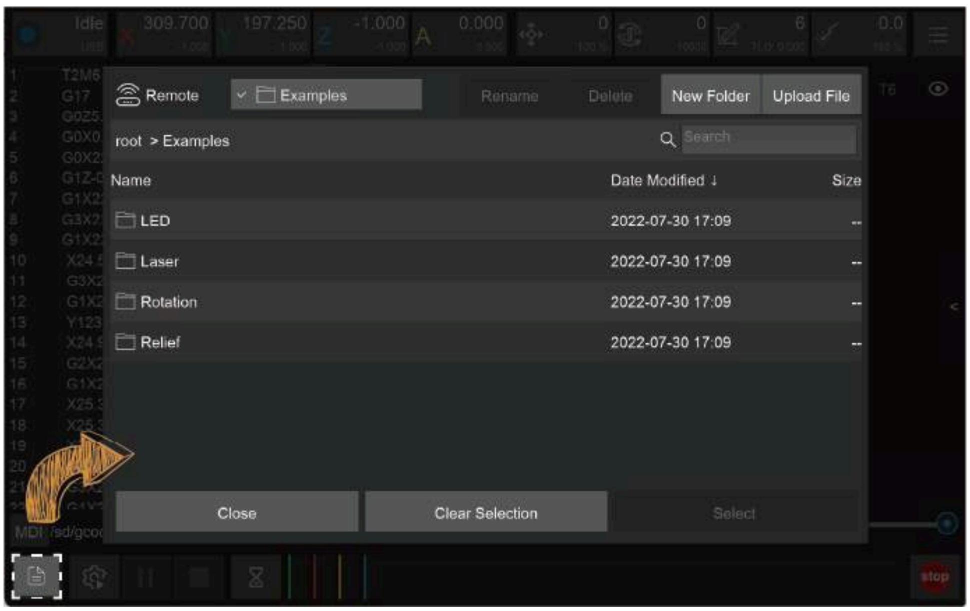
7.3. WiFi configuration: Refer to the previous WiFi setting instruction in chapter 4.

7.4. Wireless Probe Pairing: See the next chapter for detailed information.

Task Toolbar

1. File management and selection

Carvera’s G-Code is executed in the controller to ensure the task’s efficiency and stability and avoid task failures caused by WiFi or USB connection instability. Therefore, the G-Code file needs to be uploaded to the machine before running the program. We have created examples folder on the machine and have already uploaded sample G-Code files.



1.1. Remote file management:

1.1.1. Rename: Rename files or folders in the machine.

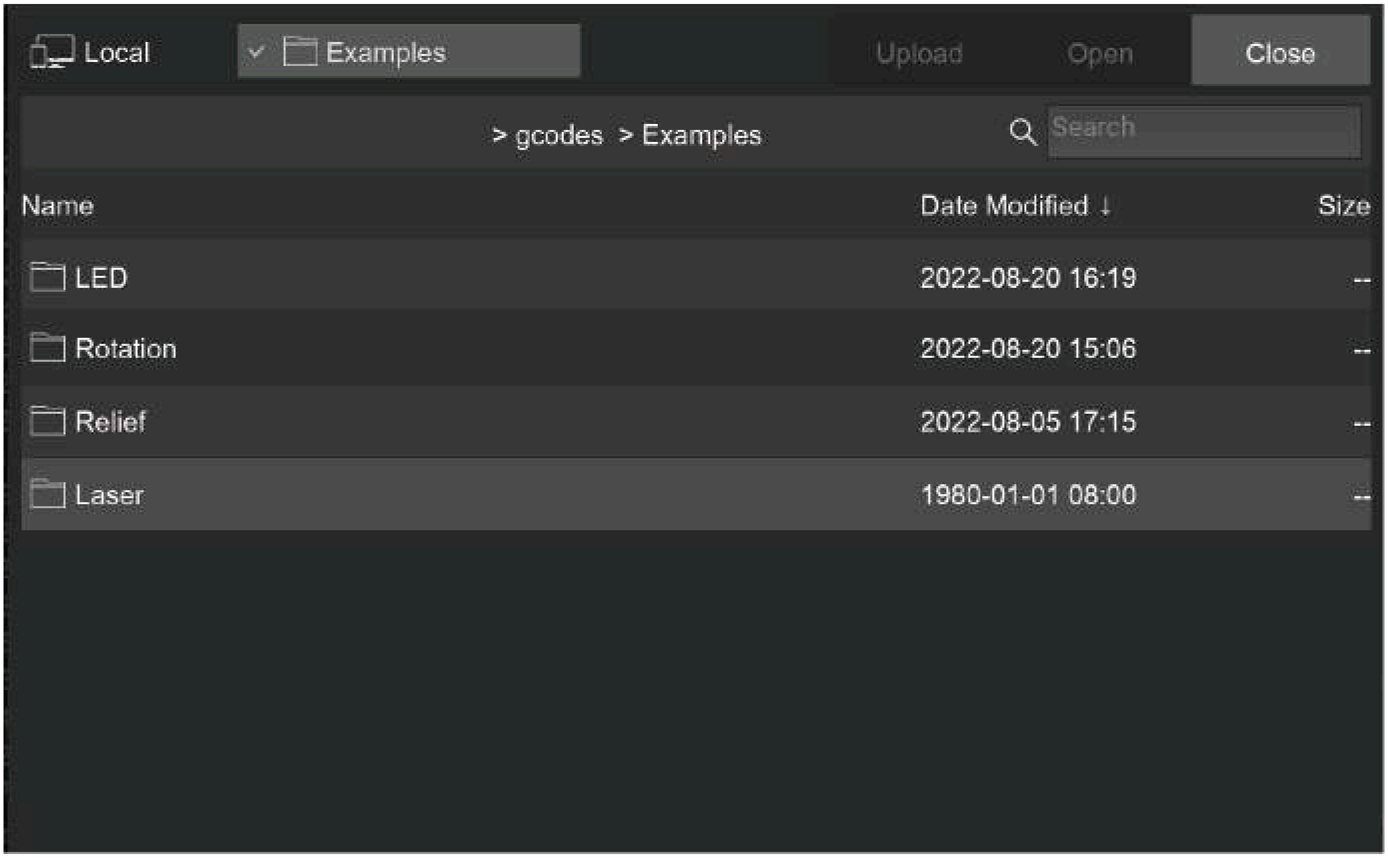
1.1.2. Delete: Delete files or folders in the machine.

1.1.3. New Folder: Create a new folder under the current file path. 1.1.4. Upload File: Switch to local file interface for uploading. 1.1.5. Close: Close the file management interface.

1.1.6. Clear Selection: Clear the currently selected G code file. 1.1.7. Select files: Select the G-Code file to run.

1.1.8. Recent Places: Short cut for recently used directories.

1.2. Local file browsing:



It opens the gcodes subdirectory under the local program installation directory by default. Therefore, we recommend putting your G-Code files here for easy management.

Upload: Select and upload local files.

Open: Open the file and preview it without uploading it (you can also do this without connecting the Carvera).

Close: Back to the remote file management interface

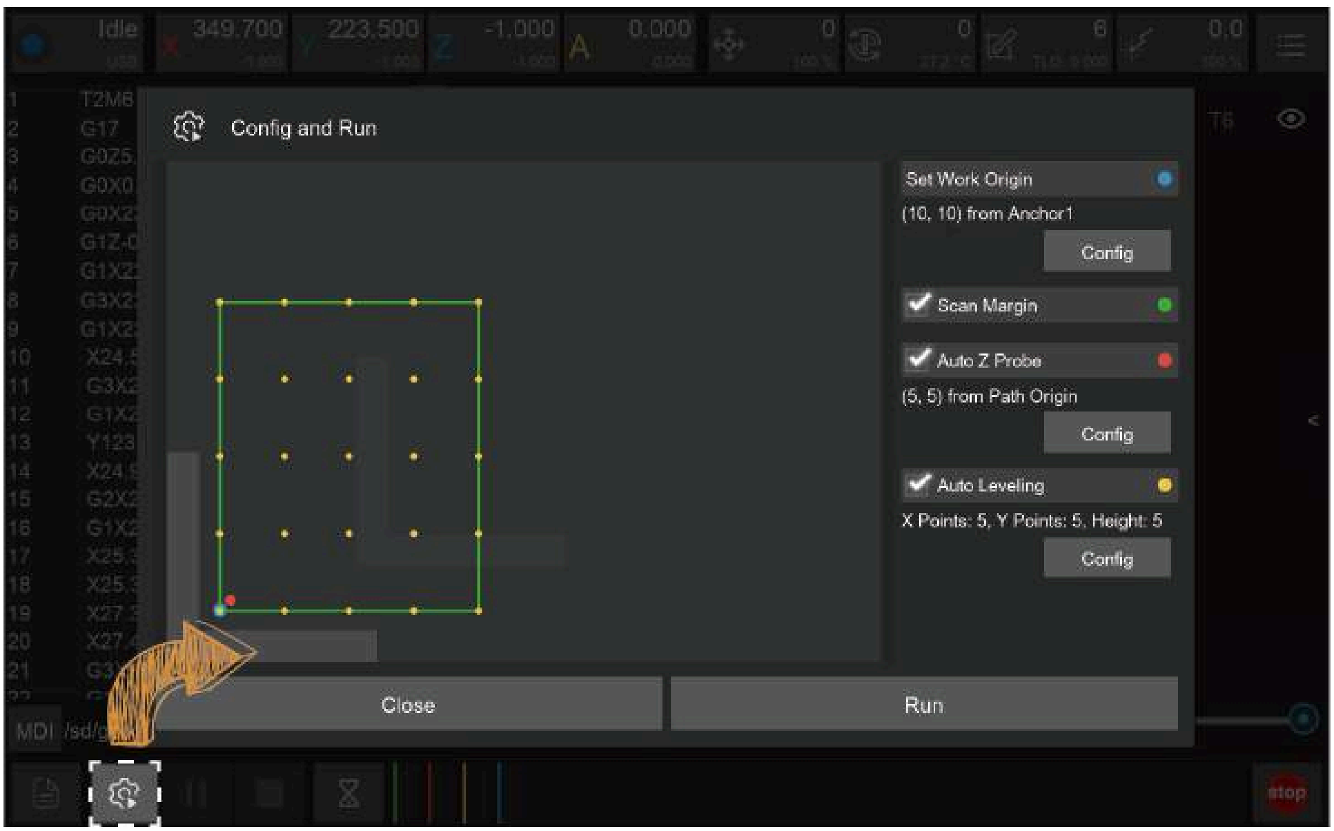
Recent Places: Short cut for recently used directories.

2.Task configuration and execution

If you have used CNC before, you definitely know that a CNC machining task requires a lot of preparation work, including setting the work coordinates (XY axis tool setting), Z-axis tool setting, workpiece levelling (PCB processing) and so on.

Because Carvera has automatic detection and leveling functions, we have integrated these settings and task execution into one interface.

We provide an innovative method that using “anchor points” for work coordinate setting, allowing you to locate XY-axis positions accurately through easy configuration.



2.1. Machining area preview: Display the preview of the machining area according to the current work coordinates and G-Code file.

L-shaped gray symbol: Anchor point position. You can choose to install the L-shaped bracket to be located either to point 1 or point 2.

Blue Circle: Zero position of work coordinate.
Green line area: G-Code file machining range.
Green Bold line area: Indicates that Scan Margin is activated to automatically scan the area before machining.

Red Circle: Z probe position when Auto Z Probe is selected.
Yellow Circle matrix: Levelling matrix when Auto Levelling is selected.

2.2. Set Work Origin: Set the work coordinates zero point relative to anchor points - the X/Y axis distance relative to anchor point 1 or 2. (Only X distance is needed when performing 4-Axis machining). Please note that the work coordinate settings will take effect immediately.

2.3. Scan Margin: Scan the rectangle path area before machining. When scanning, machine will switch to the wireless probe and turn on the red laser for observation. We recommend new users turn on this while running job.

2.4. Auto Z Probe: Z-axis tool setting is required after changing the workpiece or the zero points of the work coordinate. When doing Z probe, machine will automatically switch to the wireless probe and do probe at the set position.

Work Origin: Perform Z axis probe at a certain distance related to the X/Y axis of the working coordinate zero point.

Path Origin: Perform Z axis probe at a certain distance related to the actual X/Y machining starting point
(lower left corner). This method is selected by default.

2.5. Auto Leveling: If requiring uniform machining depth, please select the automatic leveling option such as PCB engraving. You can set the leveling matrix size and the lifting height when moving horizontally during the leveling
process. The less the flatness, the higher the lifting height needs to be. The higher the requirement for machining consistency, the denser the matrix. For PCB engraving, it is better to have matrix dots spaced about 1 cm apart.

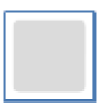
2.6. Run: Click to start machining process. If you set the scanning area, Z-axis tool setting or automatic leveling, the G-Code file will be executed after the automatic detection is completed.

3.5. Emergency stop: Immediately stops the current task, turn off the spindle, the same function as the physical button in front the machine.

3、Task control

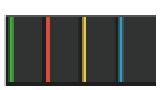
3.1. Task pause: Pause the running G-Code task and you can manually control the machine afterwards. But if you want to resume the task, please remember to turn on the spindle, or it may damage the tool if it resumes without rotating the spindle.

3.2. Task stop: Terminate the current G-Code task.



3.3. Task hold: Similar to task pause, but the pause speed is faster and cannot control the machine manually during holding.

3.4. Task track: Display the current G-Code task name, running time, percentage and other information.



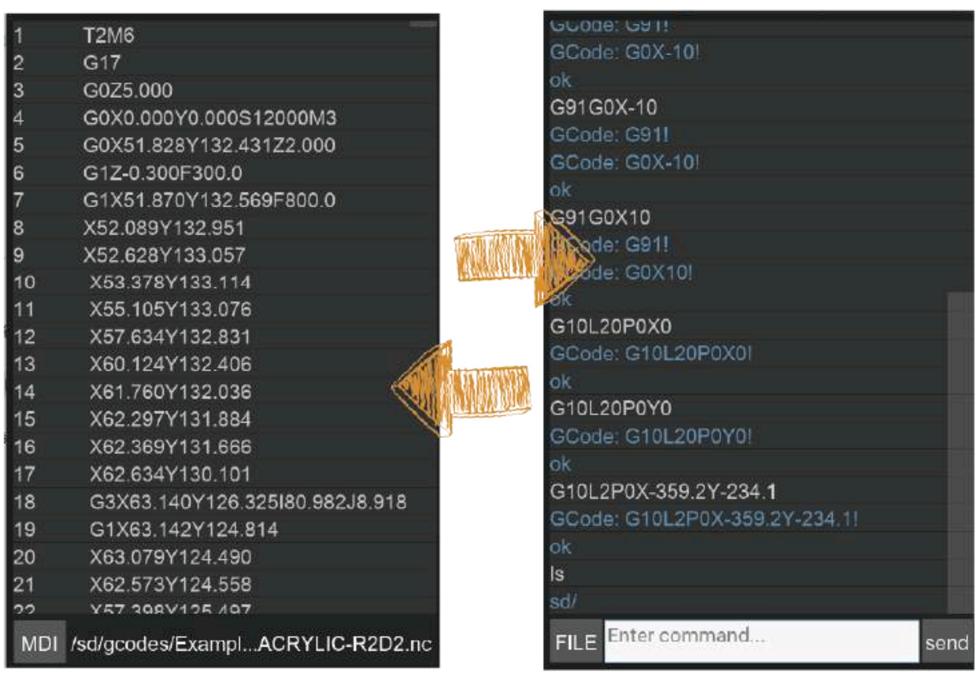
Note: The wireless probe is a precise device that integrates mechanics and electronics, which is easy to damage. Please be careful when using it. Make sure no obstacles can block its way. We suggest new users turn on the scan margin function to preview the route. It is recommended to leave su cient margin between L bracket position and path start point, greater or equal to 10/10mm.

Note: The current task cannot be stopped immediately when the task is paused or held. It has to wait for the buffer zone finish. If emergency occurs that needs to stop the machine immediately, please click the emergency stop button in the software or press the physical emergency stop button on the front panel of the machine. Carvera is built by closed-loop servo motors, and it saves the coordinate and status at each step. Therefore, no needs to reset the tool after reboot/unlock.

G-Code/MDI

1. G-Code interface: Display the currently opened remote or local G- Code file. When the task is running, it will track and highlight the running line in real-time.

2. MDI: Display detailed send/receive commands, similar to the log information. In specific cases, you can manually enter the g-code for operation and diagnosis. Enter “clear” to clear the current command area.

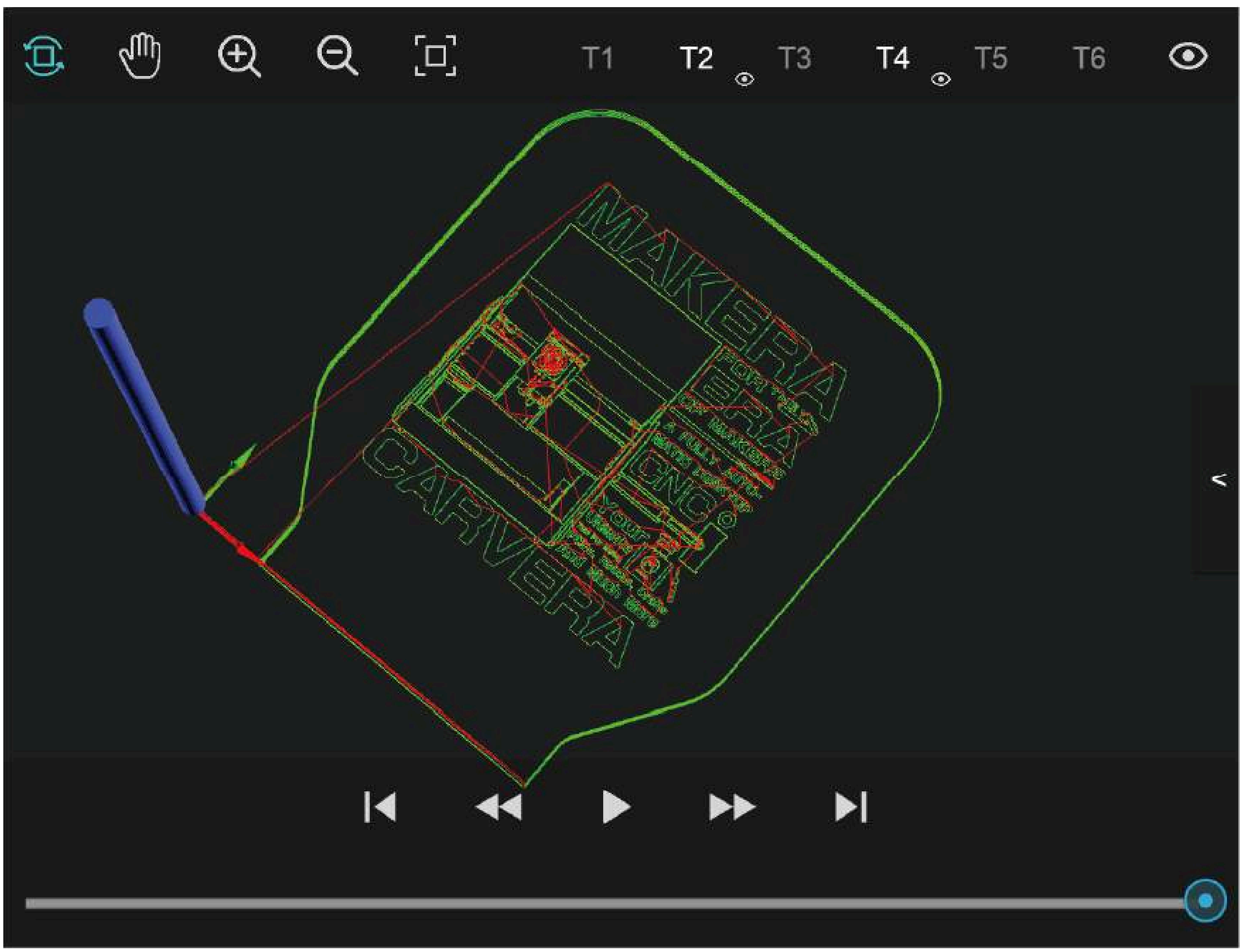


G-Code Preview

1. Graphical preview: Open the G code file to preview the G code graphics in the tool path preview area. Green lines are G1/G2/G3 code, and red lines are fast moving G0 code.

2. Display control toolbar: You can pan (right mouse button), rotate (left mouse button), zoom in (scroll wheel up), zoom out (scroll wheel down), and restore the preview (double-click) . You can also select to show/hide the G-Codes for for different tools.

3. Playback toolbar: When the task is not running, you can play, fast forward, or backward the preview. When the task is running, the preview graph shows the real-time machining trace.



Manual Operation

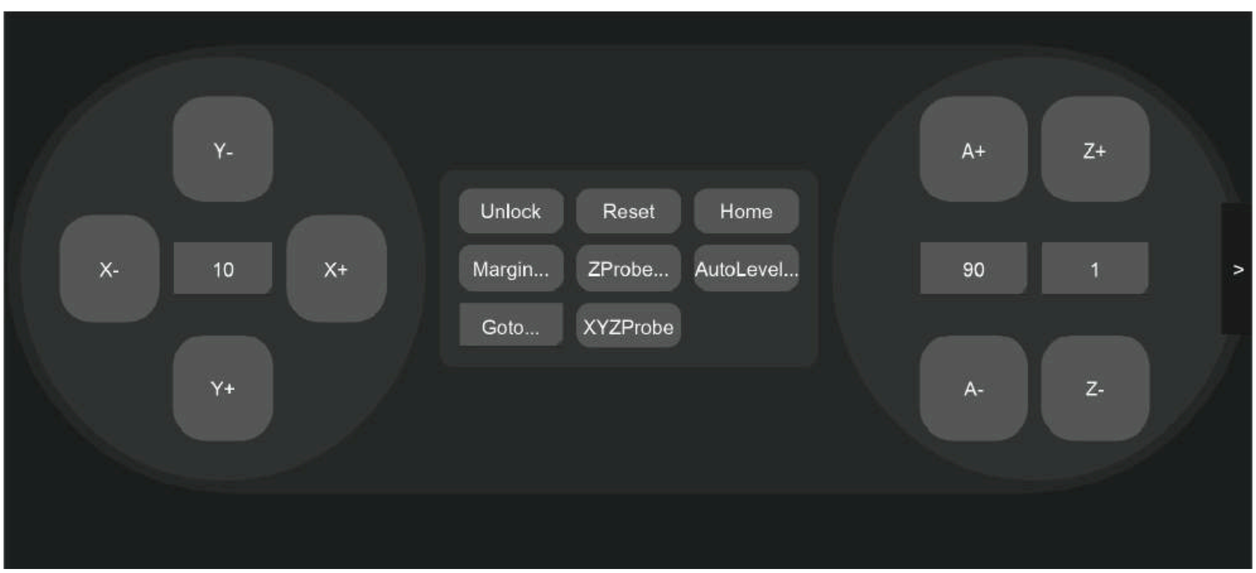
1. Jogging control: Manually control the movement of the X/Y/Z axis and rotate the A axis at G0 fast speed
(3000mm/min by default). You can set movement distance. The

2. Status control: You can unlock, reboot or reset the device.

3. Automatic detection: Automatically scan the machining area; perform Z-axis probe or automatic leveling. The difference between here and the task configuration is that you can just apply one-time detection, without executing the G code file.

4. Move to the specified location: Provide a shortcut to quickly move to the specific location; including anchor points 1, 2, working zero points, G-code starting point, and clearance point (the upper right corner next to the machine zero points by default. You can quickly move the machine to there before cleaning the working surface after machining process end).

5. XYZ Probe： Use the manual probe to do x/y/z axis probing. See detailed introductions in the next chapter.



Errors

The device triggers an alarm when encountering an abnormal. Some alarms can be closed by unlocking. Some require rebooting the machine.

Alarm Type	Reboot	Causes
Halt Manually	No	Press emergency button on the machine or software
Home Fail	No	The return zero limit switch did not trigger
Probe Fail	No	Exceeded the maximum detection distance but has no signal
Calibrate Fail	No	Tool calibration probe malfunction
ATC Home Fail	No	Return zero limit switch failure
ATC invalid tool number	No	Unsupported tool number
ATC Drop Tool Fail	No	Unsuccessfully drop the milling cutter
ATC Position Occupied	No	The position for tools is occupied
Spindle Overheated	No	Spindle overheating
Cover opened	No	The protective cover is opened during machining (cover detection enabled)
Wireless Probe Error	No	No response from the wireless probe
Emergency Stop	No	Emergency stop is pressed
Hard Limit Triggered	Yes	Motion out of range
X/Y/Z Motor Error	Yes	X/Y/Z Servo motor block
Spindle Error	Yes	Spindle stall or other errors
SD card Error	Yes	SD card reading error
Machine Is Sleeping	Yes	Machine is sleeping

General Steps

Different from the cumbersome operation process of general CNC, Carvera greatly simplifies the machining process. The general operation steps are as follows:

1. Turn on the Carvera device and wait for the homing to end.
2. Fix the workpiece to the anchor point.
3. Place the tools.
4. Open the control software and connect to the device.
5. Upload and open the G-Code file.
6. Open the task setting box, set the working zero point and automatic detection rules.
7. Start and wait until the machining process ends.

Tool Kit

Wireless Probe

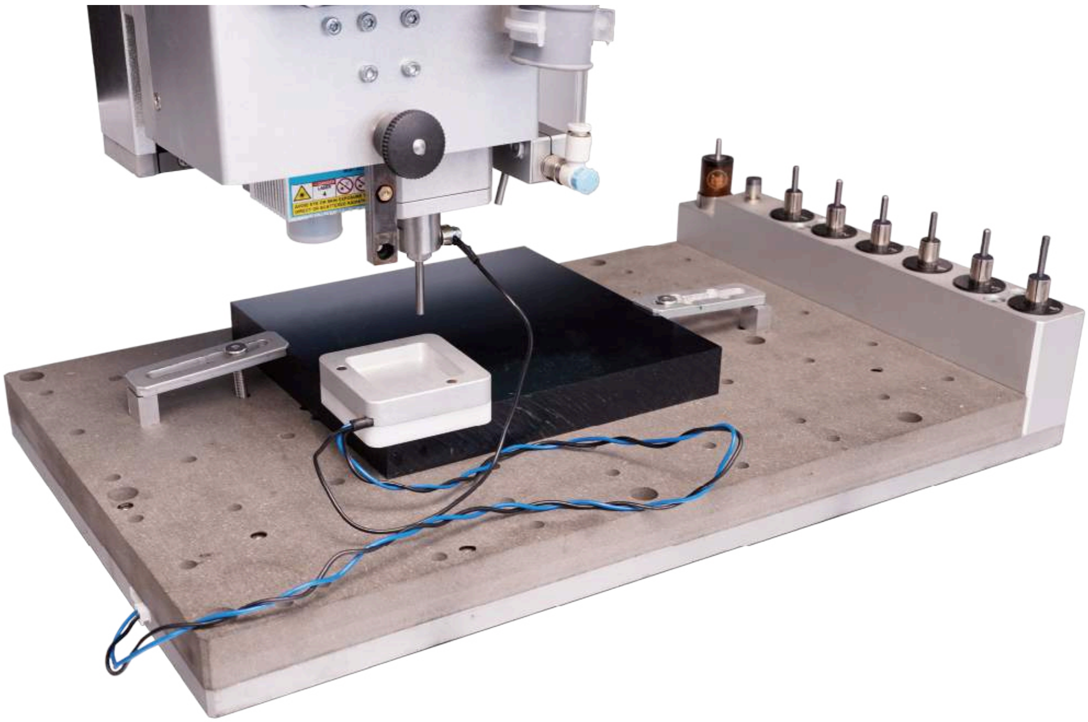
1. Charging: The wireless probe needs to be charged when it is used for the first time or after the machine has not been used for a long time (more than a week). The wireless probe will automatically start charging after turn on the machine. The charging indicator yellow light will be on and go out when it is full. You may start using it after the voltage reach 3.7v, no need to wait for full charge.

2. Probing: When the wireless probe is triggered, the green indicator light turns on.

3. Laser indicator: Press the wireless probe twice to turn on the laser indicator (used for manual tool setting of the XY axis). The laser indicator will also be turned on automatically when scanning the path area

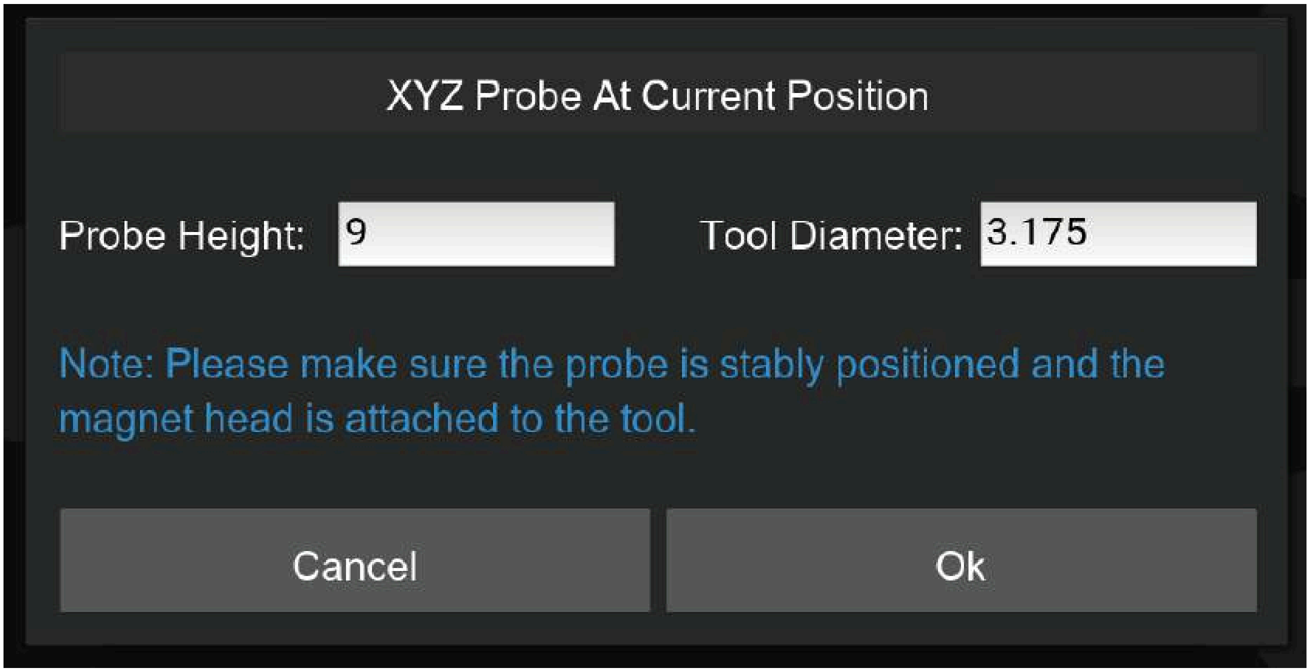
Manual Probe

Use case: Normally, using the wireless probe and the anchor-based positioning system is quite enough for most jobs. But when you need to place the workpiece not at the anchor point and need to accurately find the origin, you can use the manual probe.



Usage:

1. Plug the manual probe into the 2-pin socket on the left side of the machine bed.
2. Place the manual probe (white plastic side) against the lower left corner of the workpiece firmly.
3. Move the machine and let the milling bit be positioned in the square area of the manual probe.
4. Attach the magnetic end of the manual probe to the spindle shaft as shown.
5. Click the "XYZ Probe" function, and set the height offset and the diameter of the milling bit. (It is recommended to use the 3.175mm diameter test rod we provided for probing so that the default parameters can be applied directly)



6. Click OK to start the probing process.

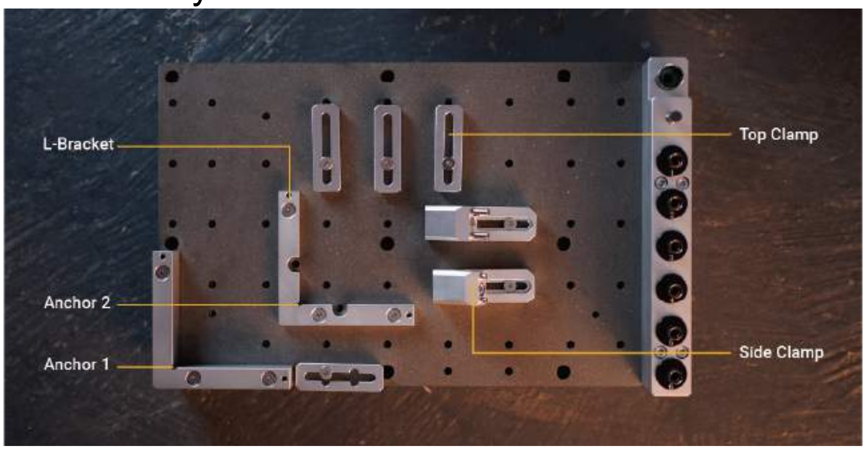
Note: Tip: The manual probe process will automatically set the origin of X, Y, and Z axes. There is no need to set them again.

Emergency Stop Button

Just like the main button in front of the Carvera machine, when any unexpected situation occurs, you can quickly press the emergency stop button to stop the machine, and the machine will stop immediately and enter the alarm state. You need to go to the control software to unlock the machine before continuing to use it. The emergency stop button has a self-locking function, just turn the emergency stop button clockwise to unlock it.

Workholding Tools

Workholding is one of the most important steps when using a CNC machine. Carvera provides two different workholding methods and corresponding tools to adapt to different types, shapes, sizes of workpieces. While holding the workpiece, you can also locate the workpiece to the pre-defined position accurately by using Carvera's anchor based system.



1. L-Brackets: Carvera provides two types of L-shaped brackets, a thin one and a thick one, as shown in the figure. The L-Bracket can be fixed at one of the two anchor points with two 4mm dowel pins and three M5 screws(thick bracket uses long screws). The lower left corner is anchor 1, and the middle position is anchor 2. There are two semi-circular openings of the thin positioner, you can put two M5 screws to fix the lower-left corner of the workpiece there.

2. Top Clamps: The top clamp usually fixes the workpiece with a thickness of less than 2 cm together with the thin L-Bracket.The purpose of the top clamp with a cross groove is to facilitate the use of long sides to fix the workpiece. We recommend using shims at the end of top clamps to fix workpieces greater than 1 cm. together to fix the workpiece with a thickness greater than 2 cm. The side clamp can also be used with the top clamp to fix a thin workpiece to process the surface, as shown in the figure.

Note: the clamping elements are made of aluminum, they will break non-carbide tooling, and will bend if you screw them down too hard.

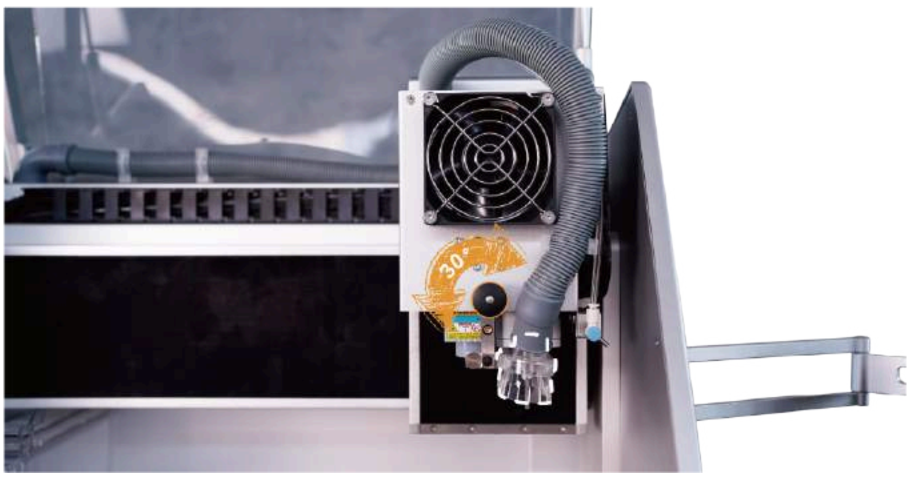
Note: If you need to cut through a workpiece, we highly recommend placing a 1-2mm thick waste board (as the complimentary one) under the workpiece. This can avoid damage to the workbench.

Note: Please select corresponding length screws to fix different workpieces, you do not want to scratch the plate under workbench.

Dust Collection Module

Chip evacuation is an important part of CNC but is usually ignored by other desktop-level CNC machines. Carvera has a built-in dust collecting and filtering system that can literally achieve dust-free machining process when doing light weight task and highly reduce the mess level when doing long time jobs .

1. Use case: The key factor in deciding whether to use the dust collection system is the interference situation. If the machining path has obstacles that block the dust shoe, do not use it. Generally, the dust collection could be used for machining thin and flat workpieces, such as plates. We suggest moving and locking the dust shoe to the highest position when machining thick and irregular workpieces. And remove the dust shoe entirely when using



2. Lock/Unlock: To fit Carvera's automatic tool change function, we designed the dust shoe to be able to slide up and down. As described earlier in the manual, pull out the black knob, and you can switch between lock and unlock states by rotating it every 30 degrees.

3. Install/uninstall: When dust collection is not needed, you can move the dust shoe to the highest position and lock it, or you can loose the hand screw that fixes the dust shoe to the linear rail and remove the dust shoe. Use the pipe holder as shown in the figure to fix the dust pipe.

4. Dust bin: Clean the dust bin in time when the milling job is done.

5. Dust bypass: The built-in dust collection system is not very powerful and the capacity of the dust bin is limited, the Carvera supports the bypass of dust to external dust collection devices such as a vacuum cleaner:

6. Do not lock the dust shoe at the bottom to avoid affecting the automatic tool changer

1.1: Introduction to Fusion 360

Fusion 360 is a cloud-based CAD/CAM tool for collaborative product development. The tools in Fusion enable exploration and iteration on product ideas and collaboration within a product development team.

Fusion 360 enables fast and easy exploration of design ideas with an integrated concept to production toolset. Fusion lets you focus on the form, function, and fabrication of your products. Use the sculpting tools to explore form and modeling tools to create finishing features. These tools let you quickly iterate on design ideas. Once you have settled on a design, you can create assemblies to validate fit and motion in your design or create photo-realistic renderings to verify the appearance. Finally, you need to fabricate your design. Use the 3D print workflows to create a rapid prototype or the CAM workspace to create tool-paths to machine your components.

Fusion 360 also helps bring design teams together for collaborative product development. All your designs are stored in the cloud, which means you and your team always access the latest data. Fusion also tracks versions of your design as you work. You can use Autodesk A360 to view each version in your web browser and promote an old version to the current version. Finally, use Fusion and A360 to share your designs and track design activity. You can even provide controlled access to your designs without requiring an Autodesk ID.

Fusion 360 uses a hybrid environment that harnesses the power of the cloud when necessary and uses local resources when it makes sense. For example, your design data is stored on the cloud and renders amazing images every time you save a new version of your design. This happens in parallel while you are creating and editing designs locally on your machine. This allows you to harness the power of your computer and the power of the cloud at the same time.

Throughout this course, you explore these areas of Fusion 360. This course gets you started designing with Fusion and helps you understand how it can improve your design processes.

1.2: Preferences

Preferences control default settings in Fusion. The Preferences dialog contains many pages of settings. Any changes you make to the preferences are saved with your Autodesk ID and are loaded when you log into another machine.
Some important preferences to review:

- General: general settings such as versioning (saving), pan, zoom, and orbit.
- General > Design: settings for the design workspaces: model, sculpt, patch.
- General > Drawing: settings for the creation of drawings.
- Material: controls the default physical material and appearance.
- Unit and Value Display: sets the precision and display of units.
- Default Units > Design and CAM: sets the default unit type.

Lesson 1: Setting Your Preferences

Learning Objectives

1. Access preferences
2. Modify preferences settings

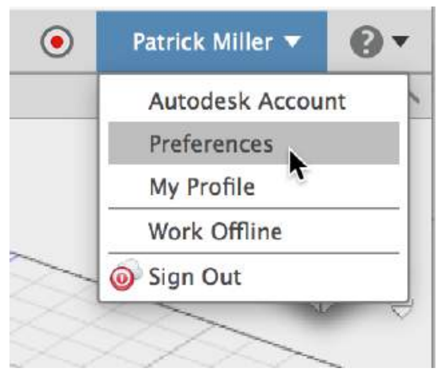
Datasets Required

No dataset are required. You start with a new empty design.

Step-by-step Guides

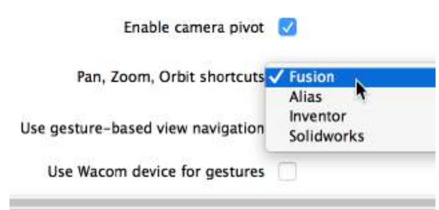
Step 1: Access the preferences dialog box.

1. Click on your name in the upper right corner then select Preferences.



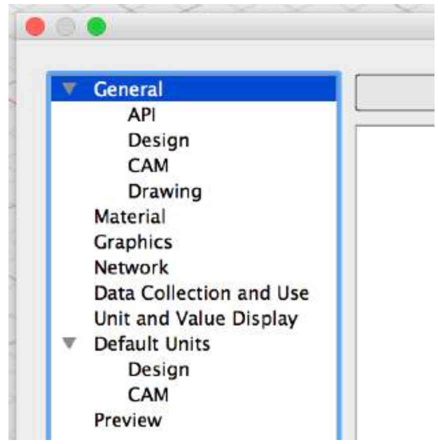
Step 2: Change the General settings.

1. Click General in the preferences list.
2. Scroll to Pan, Zoom, Orbit shortcuts.
3. Select the CAD application you are most comfortable with. This changes the mouse behavior for pan, zoom, and orbit.
4. Scroll through the other options in the General page.



Step 3: – Review the other pages.

1. The Preferences dialog contains multiple pages. Click through the other pages to review the available settings.

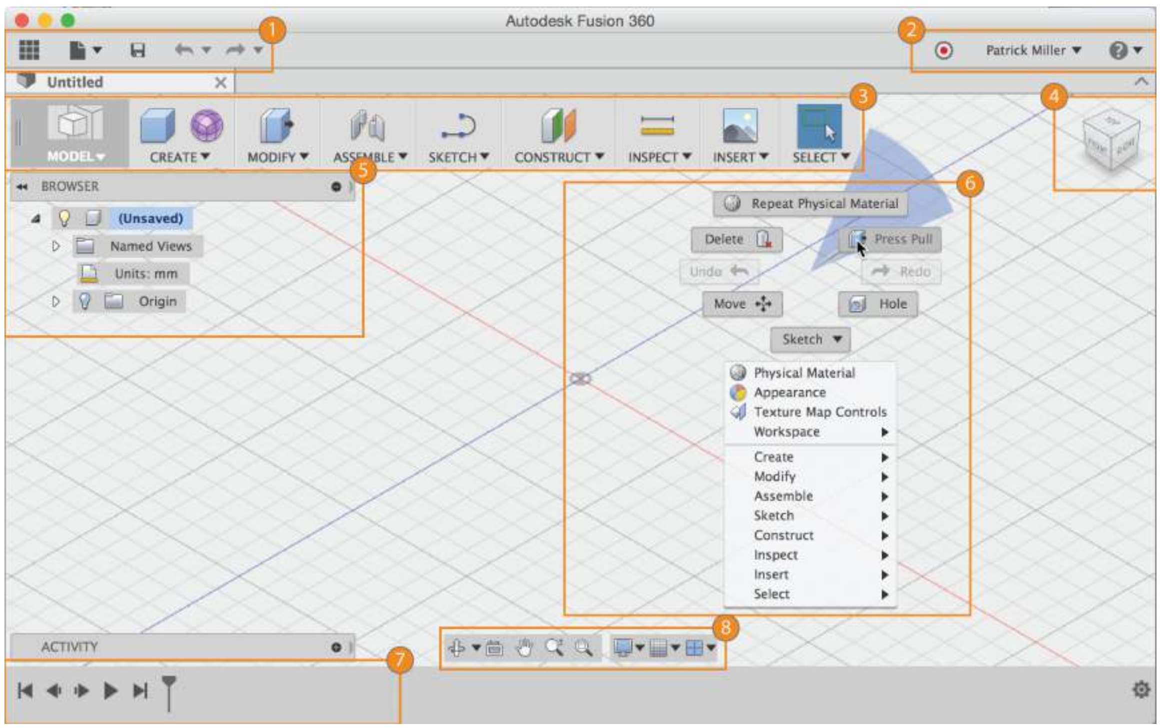


Step 4: – Reset the preferences.

1. If desired, click Restore Defaults in the lower left of the Preferences dialog. This resets all settings to the installed settings.

1.3: User Interface

The UI can be broken up into 8 areas. To help you become familiar with the product we will describe each of these areas and go into more details with how to use it in the following lessons.



Application bar

The Application bar is where you'll find and use the following:

	Data Panel – Used for data management and collaboration.
	File – Create a New Design, Save, Export, and 3D Print.
	Save – Save an untitled design or save the changes to a design as a new version.
	Undo/redo – Undo/redo operations.

Profile and help

In profile you can control your profile and account settings, or use the help menu to continue your learning or get help in troubleshooting.

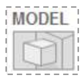
	Profile – In your profile you can access your own personal settings.
	Help – In the help menu you can access online learning content, help, forums, step-by-step tutorials, or link to community content.

Toolbar

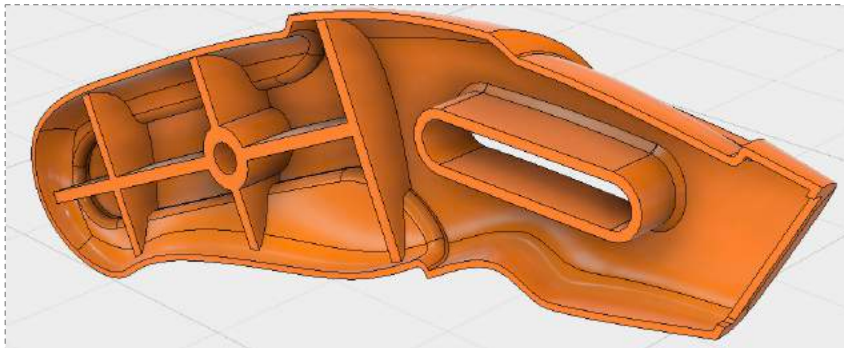
Use the Tool bar to select the workspace you want to work in, and the tool you want to use in the workspace selected.

Your Workspaces

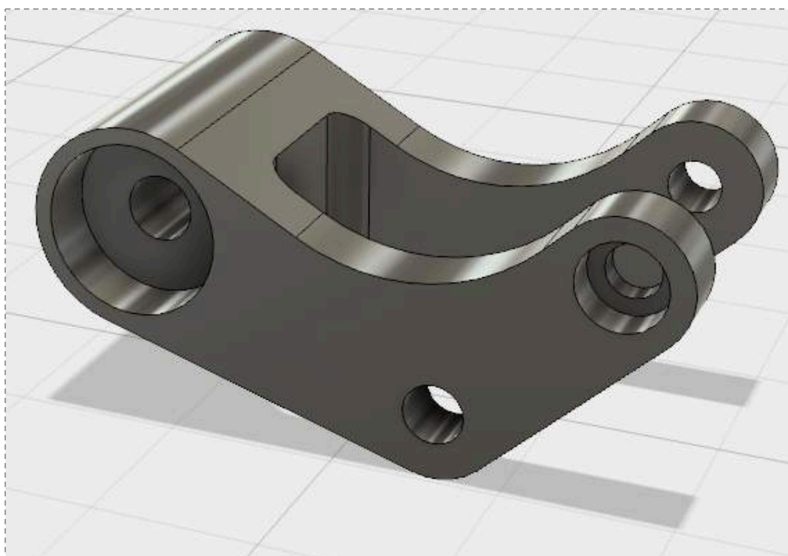
Fusion 360 uses these workspaces to control the commands that are available and the type of data that is created.

	The Sculpt workspace is used to create organic shapes by manipulating faces, edges, and vertices.
	The Model workspace is used to create solids with hard edges and flat faces.
	The Patch workspace is used to create open surfaces to stitch into solid bodies.
	The Render workspace is used to set up the environment and create photo-realistic renderings.
	The CAM workspace is used to create and simulate tool-paths then generate g code for subtractive manufacturing.
	The Animation workspace is used for to create exploded views of an assembly and control over unique animations of parts and assemblies.
	The Drawing workspace is used to generate 2D manufacturing drawings.

> *Note:*
>
> *Very frequently, your designs will require that you work in* ***both sculpt and model workspaces****, back and forth. You might even throw* ***patch in there to stitch surfaces*** *together into a solid. Create the organic shape in sculpt then use model for manufacturing features afterwards.*



Note:
*Use **model** to create designs with hard edges and flat faces. Model creates bodies requiring exact sizes and edges. Entering exact values is not required but is typical.*



4

ViewCube

Use the ViewCube to orbit your design or view the design from standard view positions.

5

Browser

The browser lists objects in your design. Use the browser to make changes to objects

	and control visibility of objects.
6	**Canvas and marking menu** Left click to select objects in the canvas. Right-click to access the marking menu. The marking menu contains frequently used commands in the wheel and all commands in the overflow menu.
7	**Timeline** The timeline lists operations performed on your design. Right-click operations in the timeline to make changes. Drag operations to change the order they are calculated.
8	**Navigation bar and display settings** The navigation bar contains commands used to zoom, pan, and orbit your design. The display settings control the appearance of the interface and how designs are displayed in canvas.

Lesson 1: User Interface Overview

Learning Objectives

1. Familiarize yourself with the UI by creating a simple design
2. Use the toolbar
3. Use the marking menu
4. Control objects in the browser
5. Control operations in the timeline
6. Change workspaces

Datasets Required

No dataset required. You start with a new empty design.

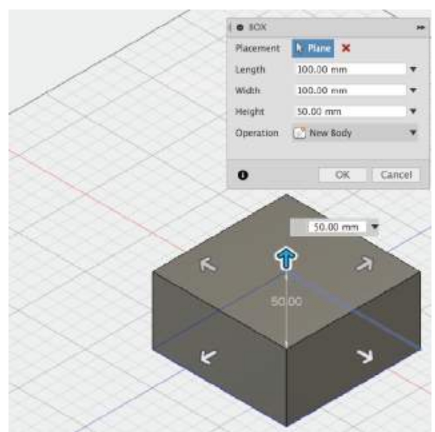
Step-by-step Guides

Step 1: Create a new design

1. Click  then select **New Design**.

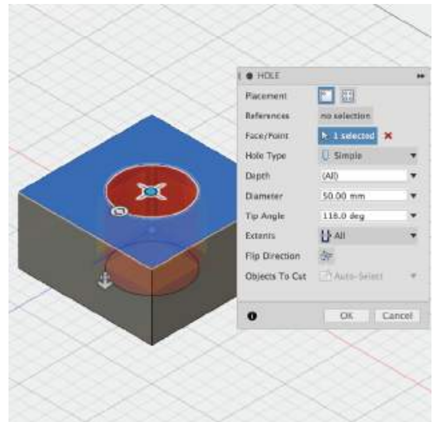
Step 2: Create a box

1. Click Model > Create > Box to start the box command.
2. Select the XZ Plane along the bottom of the canvas.
3. Pick two points to define the length and width of the box.
4. In the Box dialog, use these values:
   Length: **100 mm**
   Width: **100 mm**
   Height: **50 mm**
5. Click **OK**.



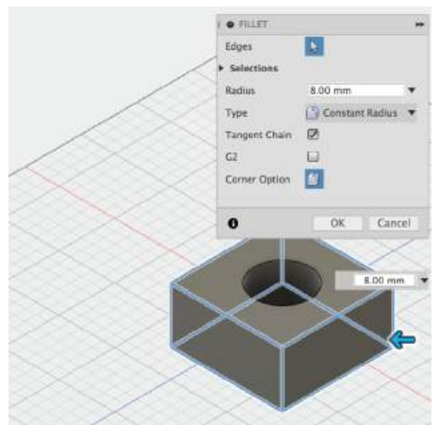
Step 3: Add a hole to the box

1. Click Model > Create > Hole.
2. Select the top face of the box.
3. Drag the center of the hole to the center of the box.
4. Set Diameter to a value of **50 mm**.
5. Change Extents to **All**.
6. Click **OK**.



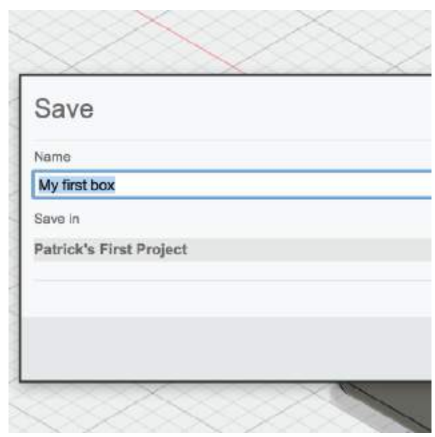
Step 4: Round the edges of the box

1. Right-click an empty area in the canvas then select **Press Pull**.
2. Hold the left mouse button then drag to window select the entire box.
3. Click the top and bottom edges of the hole to deselect the edges.
4. Enter **8 mm** for the Radius.
5. Right-click then select **OK**.

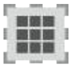
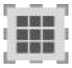


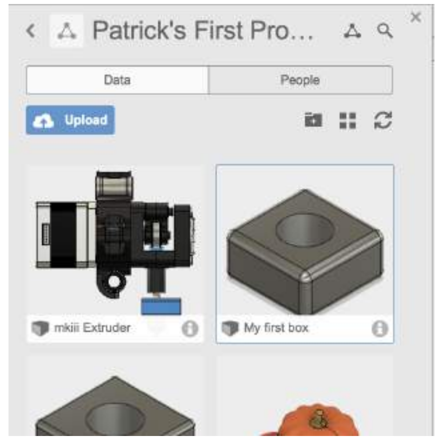
Step 5: Save your design

1. Click  to save the design.
2. Enter **My first box** in the Name field.
3. Set Save in to **<your name>'s First Project**.
4. Click **Save**.



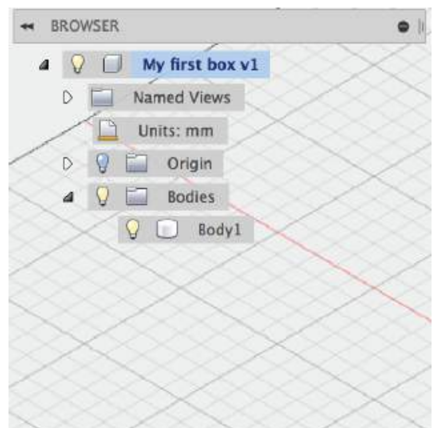
Step 6: Display the Data Panel

1. Click  in the upper left corner to display the Data Panel.
2. The active project name is displayed at the top. Thumbnails of all the designs in the project are listed. All data is stored in A360 in the cloud.
3. Click  again to hide the Data Panel.



Step 7: Use the browser

4. Click  next to Origin to display the origin planes.
5. Click  the light bulb again to turn the origin planes off.
6. Click  next to Bodies in the browser to expand the folder. There is one body in this design.



Step 8: Use the timeline

1. Click  to replay the operations in the design.
2. Right-click the  fillet operation in the timeline.
3. Change the Radius to **5 mm** and click **OK**.

Step 9: Change workspaces

1. Select  then select  to switch to the render workspace. Notice the canvas appearance

changes and the timeline is replaced with the Rendering Gallery. This workspace is used to render images of your design.

2. Notice the Rendering Gallery at the bottom of the interface. This gallery displays a thumbnail of your cloud renderings and shows the progress of renderings that are in process.
3. If the renderings have been processed, click on of the thumbnails to display the image.
4. Close the Cloud Rendering dialog.
5. Select 

then select 

to return to the model workspace.
6. Keep the design open. You will use it in the next lesson.

1.4: Navigation

There are three ways to manipulate the view of your design:

- Navigation bar
- ViewCube
- Wheel button on a mouse

Navigation bar

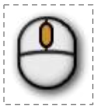
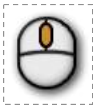
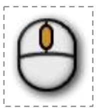
Use the commands in the Navigation bar to pan, zoom, and orbit the canvas. The menus on the right end control Display Settings and Layout Grid options.

ViewCube

Use the ViewCube to orbit the design in the canvas. Drag the ViewCube to perform a free orbit. Click faces and corners of the cube to access standard orthographic and isometric views.

Mouse

Use mouse shortcuts to zoom in/out, pan the view, and orbit the view.

	**Scroll middle mouse button** to **zoom in or zoom out**.
	**Click and hold middle mouse button** to **pan the view**.
SHIFT + 	**Shift Key + middle mouse button** to **orbit the view**.

Mac Trackpad

	Use the **2 finger pinch** to **zoom out**.
	Use the **2 finger spread** to **zoom in**.
	Use the **2 finger swipe** to **pan the view**
	**Hold Shift + the 2 finger swipe** to **orbit the view**

Lesson 1: Navigate the canvas

Learning Objectives

1. Use the commands in the Navigation Bar
2. Use the mouse to zoom and orbit the design

3. Use the ViewCube to navigate the design

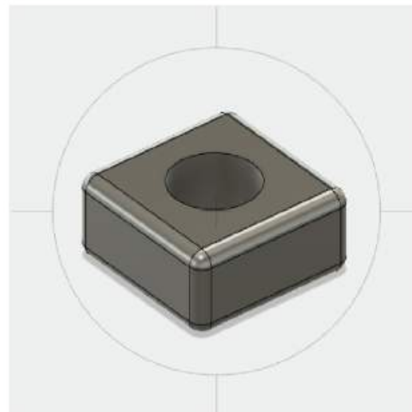
Datasets Required

Use the design from the previous lesson.

Step-by-step Guides

Step 1: Use the Navigation Bar

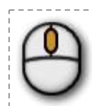
1. Click  Orbit then drag within the circle.
2. Click  Pan then drag in the canvas to pan.
3. Click  Zoom then drag up and down in the canvas to zoom in and zoom out.



Step 2: Use the mouse

1. Roll the wheel forward and backward to zoom in and zoom out.
2. Click and hold then drag to pan.
3. Hold the Shift key then drag with the middle mouse button to orbit the design.
4. Double-click the middle mouse button to zoom extents.

Note: if you changed the Pan, Zoom, Orbit shortcuts preference then your mouse wheel will behave differently.

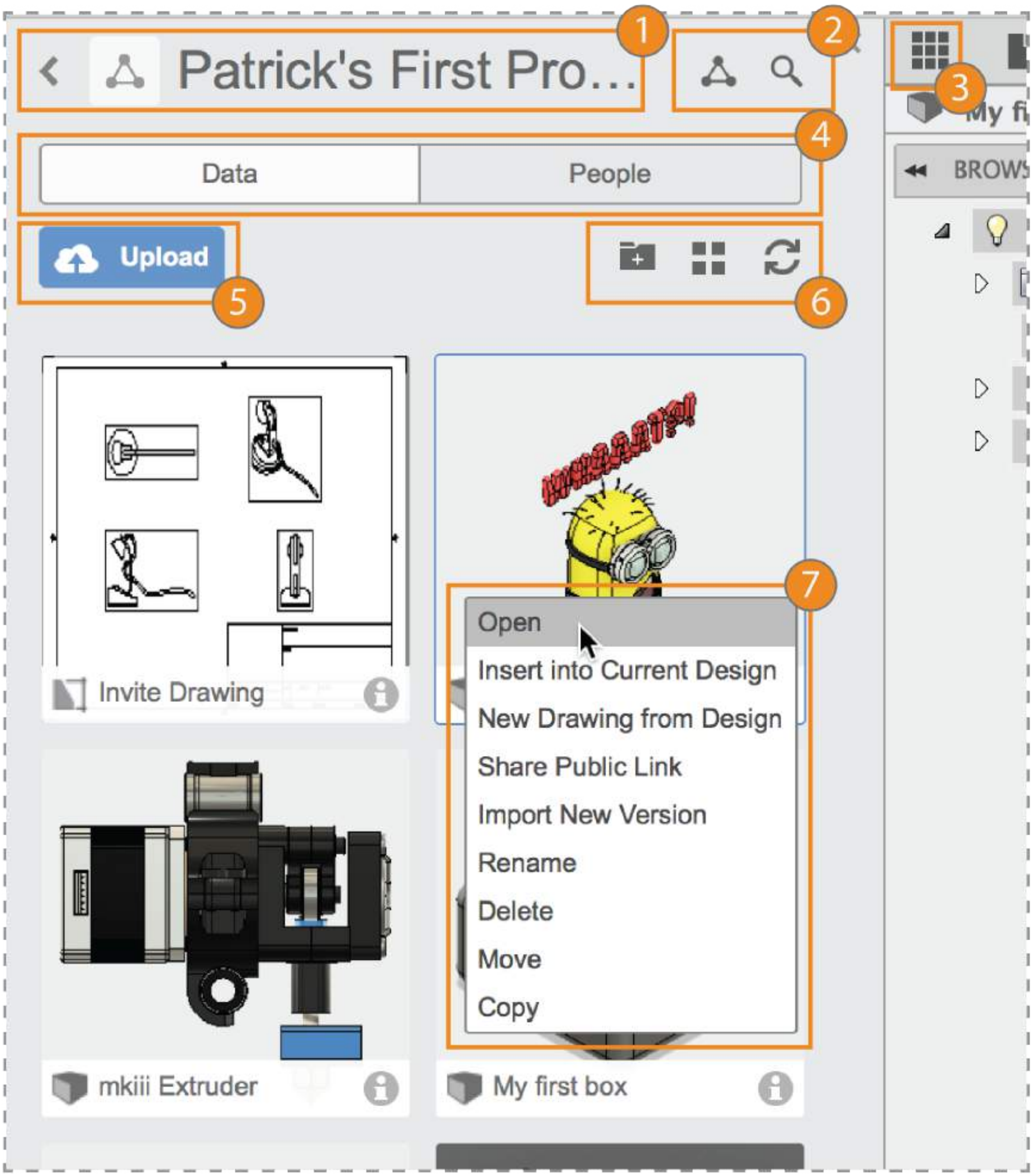


Step 3: Use the ViewCube

1. Left click and drag the ViewCube to orbit the design.
2. Click one of the corners of the cube to go to an isometric view.
3. Click on the FRONT face to go to the front orthographic view.
4. Click  Home to return to the home view.

1.5: Data Panel Interface

Use the Data Panel on the left of the application to access your designs and manage projects.



1 **Project selector**

Displays the name of the active project. Click the back arrow to display a project list.

2 **Project tools**

Displays project data in Autodesk A360 or searches the active project.

Project Details – Opens the active project in Autodesk A360 in your default internet browser.

Search – Search the active project or all projects you have access to.

3 Data Panel show/hide

Click to show or hide the Data Panel.

4 Data or People

Controls the display of data or users in the Data Panel.

5 Upload

Click to upload data.

6 View tools

Create a new folder or change the display of items in the Data Panel.

New Folder – Create a new folder in the active project.

View Options – Choose how to sort and list data.

Refresh – Refresh the data in the Data Panel.

7 Thumbnails

Right-click a thumbnail to access commands for that specific design.

Lesson 1: Manage your design

Learning Objectives

1. Open a design
2. Change the active project

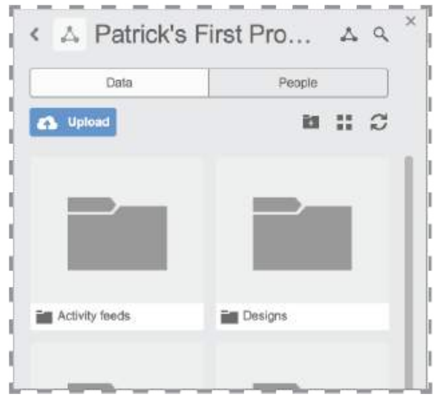
Datasets Required

Use the design from the previous lesson.

Step-by-step Guides

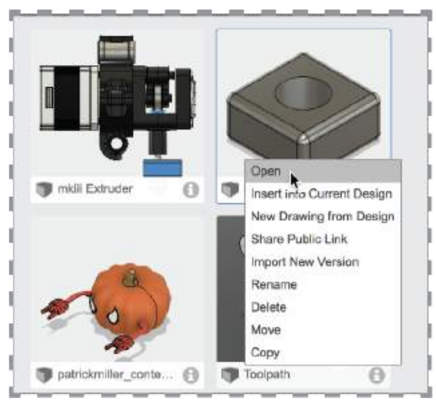
Step 1: Set the active project

1. Click the back button next to the active project in the Data Panel.
2. Double-click **<your name>'s First Project** to make that the active project.



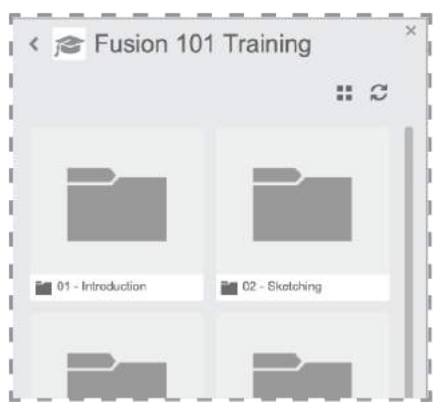
Step 2: Open a design

1. Click the **X** on the tab for My first box. The design is closed and you are presented with an empty design.
2. In the Data Panel, right-click on My first box then select **Open**.



Step 3: Access the Fusion 101 Training project

1. Click the back button next to the active project in the Data Panel.
2. Scroll to the **SAMPLES** category.
3. Double-click Fusion 101 Training to set it as the active project. This project contains the data you will use for the other lessons in this course.



Keyboard Shortcuts

Command	Windows	Mac
Undo	Ctrl + Z	Command + Z
Redo	Ctrl + Y	Command + Y
Copy	Ctrl + C	Command + C
Paste	Ctrl + V	Command + V
Cut	Ctrl + X	Command + X

Sculpt Workspace Selection	Windows	Mac
Grow selection	Shift + Up arrow	Shift + Up arrow
Shrink selection	Shift + Down arrow	Shift + Down arrow
Loop selection	Alt + P	Control + P
Loop grow selection	Alt + O	Control + O
Ring selection	Alt + L	Control + L
Ring grow selection	Alt + K	Control + K
Ring shrink selection	Alt + J	Control + J
Previous U	Alt + Left arrow	Control + Command + Left arrow
Next U	Alt + Right arrow	Control + Command + Right arrow
Previous V	Alt + Down arrow	Control + Command + Down arrow
Next V	Alt + Up arrow	Control + Command + Up arrow
Range selection	Alt + M	Command + M
Invert selection	Alt + N	Command + N
Toggle box mode	Ctrl + 1	Ctrl + 1
Toggle control frame mode	Ctrl + 2	Ctrl + 2
Toggle smooth mode	Ctrl + 3	Ctrl + 3
Select edge ring	Double-click an edge	Double-click an edge
Select face ring	Select two faces then double-click a third face	Select two faces then double-click a third face

Edit Form Command	Windows	Mac
Add geometry	Alt + Drag	Option + Drag
Add geometry and keep creases	Alt + Ctrl + Drag	Option + Command + Drag

2.1: Sketching

Many features that you create in Fusion 360 start with a 2D sketch. In order to create intelligent and predictable designs, a good understanding of how to create sketches and how to apply dimensions and geometric constraints. Fusion does support 3D sketches although, in this module we will cover basic sketching tools to create and edit a 2D sketch. In this lesson we will be building the housing for a hypocycloidal gearbox. Visit robotarm.org to learn more about it.

Lesson 1: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Create geometry in a sketch
3. Use constraints to position geometry
4. Use dimensions to set the size of geometry

Datasets Required

In Samples section of your Data Panel, browse to:

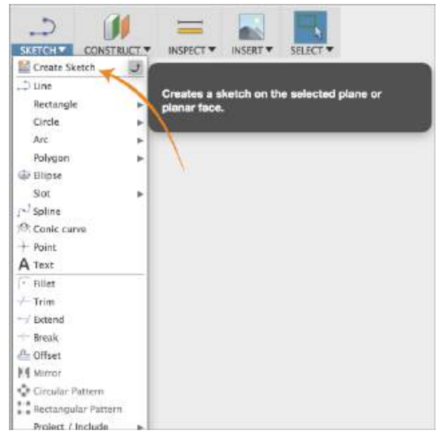
Fusion 101 Training > 02 – Sketching > 02_Sketching

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

Step 1: – Start the **Sketch** command

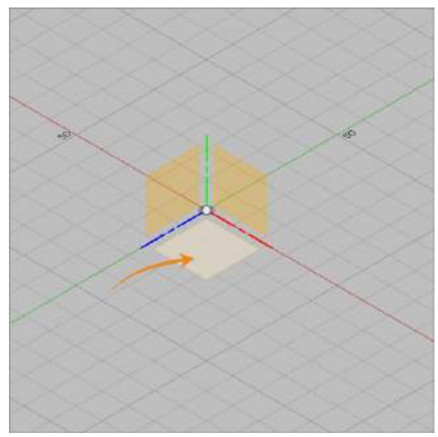
1. Select **Sketch** > **Create Sketch**.



Step 2: – Select the sketch plane

1. You are now prompted to select a “plane” to sketch on.
2. Select the “Front” (XY) plane.

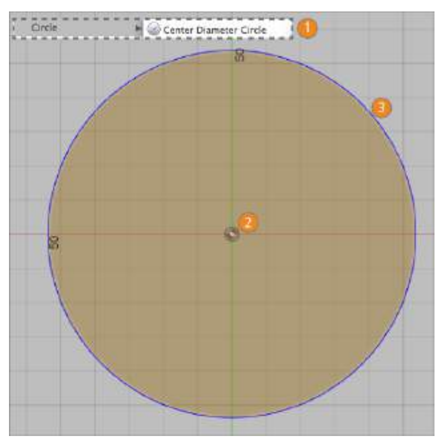
Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.



Step 3: – Create a circle

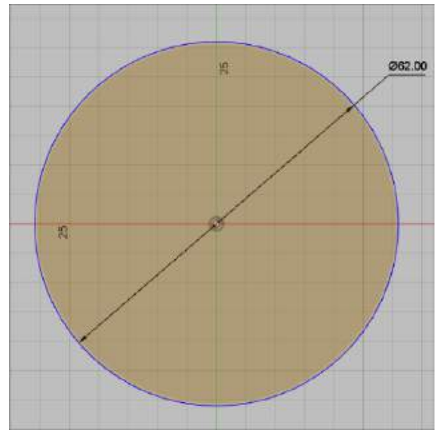
1. Select **Sketch > Circle > Center Diameter Circle**
2. Now hover over the **origin** (center) of the sketch. You should see the cursor “snap” to this location.
3. Click once to begin placing the circle.
4. Move the mouse away from the center to define the size.
5. Click again to place the circle.

Note: In Fusion 360 it is important to “snap” entities to the origin when possible. This accurately grounds the objects and ensures they will behave as expected.



Step 4: – Dimension the circle

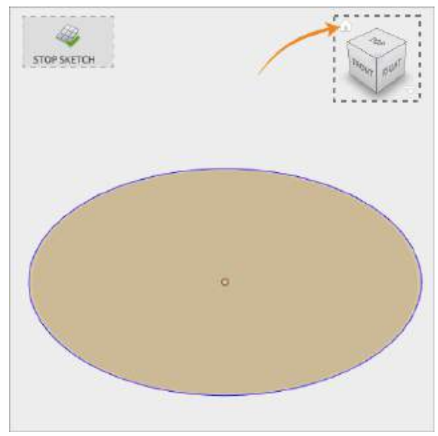
1. Select **Sketch > Sketch Dimension.**
2. Select the circle sketch.
3. Click again to place the dimension.
4. Type in a value of 62 mm.
5. Press **Enter** to accept the value.



Step 5: – Finish the **Sketch** command

1. **Finish Sketch** by clicking on Stop Sketch.
2. Select the home view icon to the let of the view cube.

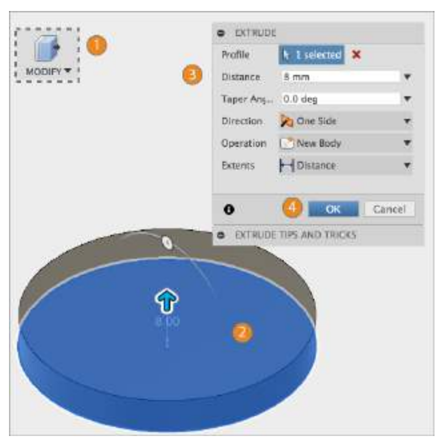
Note: You must "stop" a sketch before you can continue building geometry since Sketch is a mode that you enter and exit within workspaces.



Step 6: – Extrude the Circle

1. Select **Modify > Press Pull.**
2. Select the Profile.
3. Drag the arrow up or type in a value of **8 mm** to set the depth.
4. Press **OK to finish.**

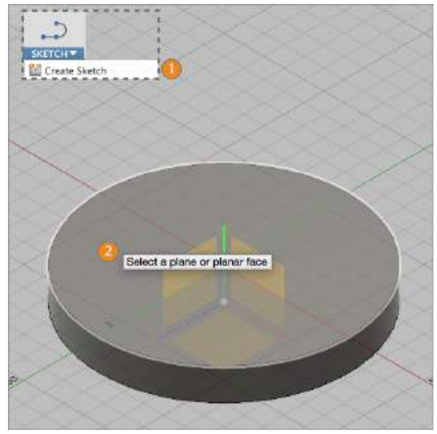
Note: In Fusion 360 you need to select the shaded area in the middle of the circle, NOT the edge of the circle to define profile for the extrude. Later we will look more closely at sketch profiles but for now simply select the shaded area.



Step 7: – Create another sketch

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.

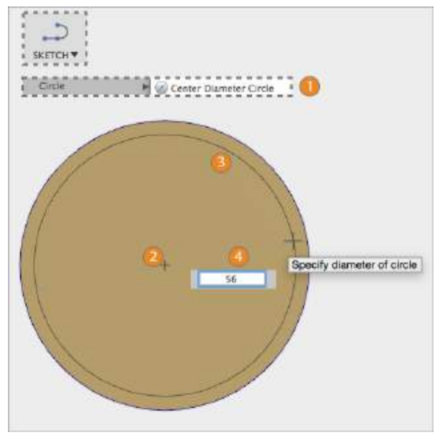
Note: This starts a new sketch on the top face of the cylinder. You will see the outer edge of the cylinder is already captured in the sketch.



Step 8: – Create an inner circle.

1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select the center point of the top face.
3. Start dragging the circle.
4. Type a value of **56 mm.**
5. Press **Enter** twice.
6. Select **Stop Sketch.**

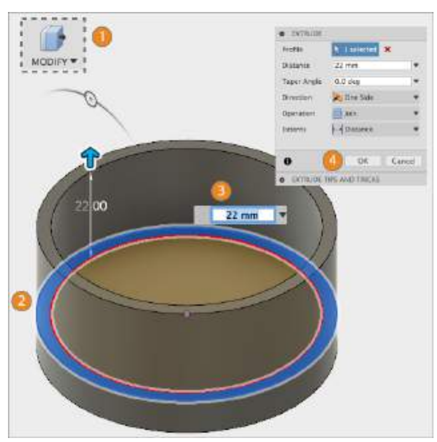
Note: We can also create the inner circle with the Sketch > Offset command. Click on the outer circle and offset it to 56 mm. If you type in a value when creating sketch geometry the dimension will be automatically added. Otherwise you can apply a dimension like in the previous step.



Step 9: – Extrude the inner circle

1. Select **Modify > Press Pull.**
2. Select the inner circle profile.
3. Drag the arrow up or type in a value of **22 mm** to set the depth.
4. Press **OK.**

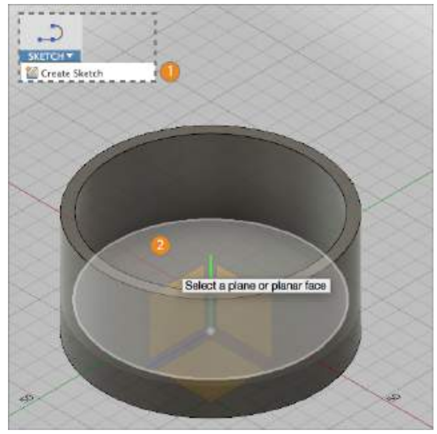
Note: One sketch may contain multiple profiles. In this case there is a profile inside the circle you drew and a profile between the outer edge and the inner circle. You can select multiple profiles for an extrude feature. In this case you will only select the outer profile.



Step 10: – Create another sketch

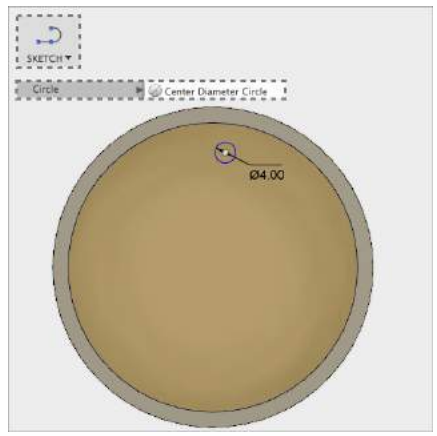
We're now going to start creating smaller circle profiles for the holes so we can pattern them later.

1. Select **Sketch > Create Sketch.**
2. Select the top of the cylinder.



Step 11: – Create a circle

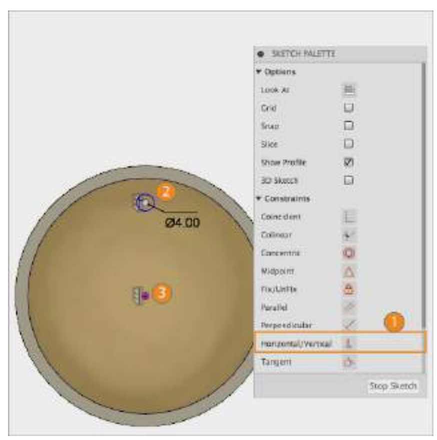
1. Select **Sketch > Circle > Center Diameter Circle.**
2. Select a point in the top part of the inner circle.
3. Start dragging the circle.
4. Type a value of **4 mm.**
5. Press **Enter** twice to accept the value.



Step 12: – Constrain the Circle

1. Select **Horizontal/Vertical** constraint from the Sketch Palette.
2. Select the center point of the new circle and the center point of face.
3. Press **esc** to exit the command.

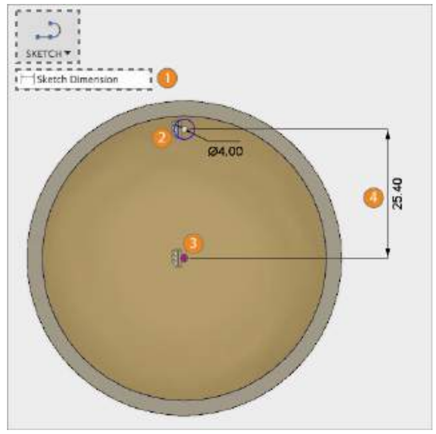
Note: Constraints create relationships in your sketch. By saying that these two points are "vertical" determines how the will be aligned in the sketch. These relationships are persistent, meaning that if the center of the face moves, the sketch will move also.



Step 13: – Dimension the circle

1. Select **Sketch > Sketch Dimension.**
2. Select the center of the circle.
3. Select the center of the face.
4. Click to place the dimension.
5. Type in a value of 25.4 mm.
6. Press **Enter** to accept the value.

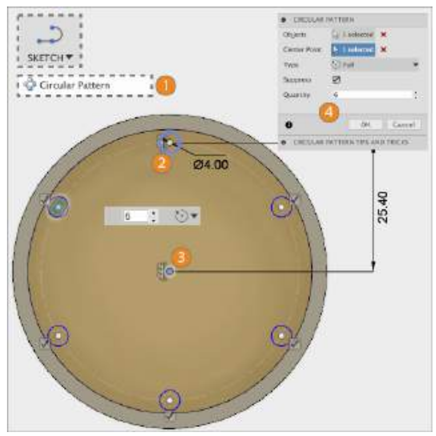
Note: This now has "fully" constrained the circle. This means that the geometry is fully locked down.



Step 14: – Pattern the Circle

1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the circle.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch** to end the sketch mode and return back into the modeling environment**.**

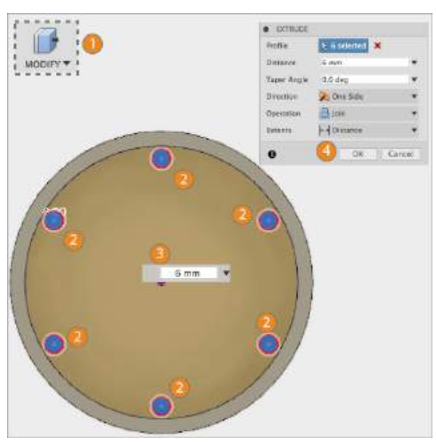
Note: Make sure you initiate what you want to select by click on the "no selection" box before trying to pick the center point for the pattern. Each box should then say "1 Selected." You can press the red "X" next to the box if you want to clear the current selections.



Step 15: – Extrude the Circles.

5. Select **Modify > Press Pull.**
6. Select the 6 Profiles.
7. Drag the arrow up or type in a value of **6 mm** to set the depth.
8. Press **OK.**

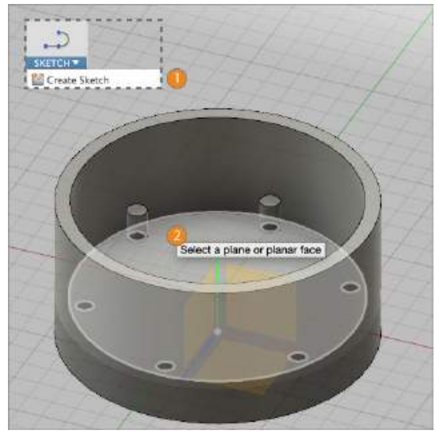
Note: It may help to zoom in a little bit to make sure you are grabbing the correct profiles. Make sure you have six profiles selected. If you need to add or remove profiles later hold ctrl (for Windows) or command (for Mac) and select the profile.



Step 16: – Create another sketch

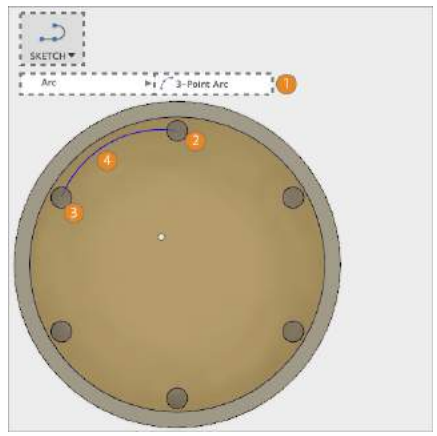
We’re almost done with our design. We need to create the pockets for the round gear teeth.

3. Select **Sketch > Create Sketch.**
1. Select the top of the cylinder again.



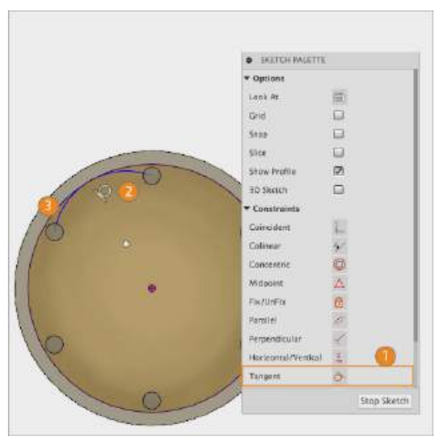
Step 17: – Create an arc

1. Select **Sketch > Arc > 3 Point Arc.**
2. Select the center of one circle.
3. Select an adjacent circle center point.
4. Click the third point near the edge.
5. Press **esc** to exit the command.



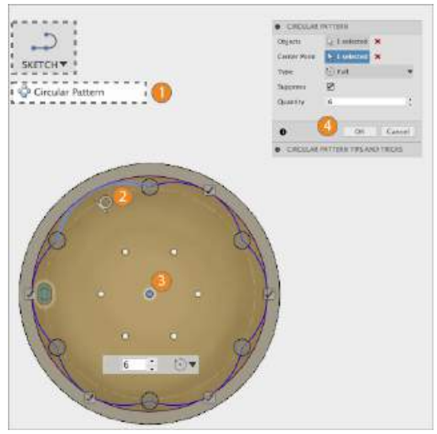
Step 18: – Constrain the Arc

1. Select the **Tangent** constraint from the Sketch Palette.
2. Select the new Arc and the inner edge so that they are constrained.
3. Press **esc** to exit the command.
4. Select **Stop Sketch** to return back to the modeling environment.



Step 19: – Pattern the Arc

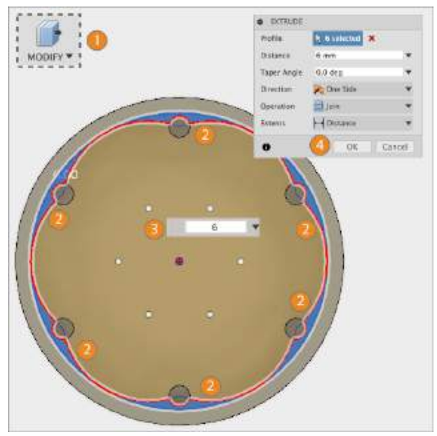
1. Select **Sketch > Circular Pattern.**
2. In **Objects,** select the arc.
3. In **Center Point**, select the face center.
4. Change the **Quantity** to **6.**
5. Press **OK.**
6. Select **Stop Sketch.**



Step 20: – Extrude the Arcs

1. Select **Modify > Press Pull.**
2. Select the 6 Profiles.
3. Drag the arrow up or type in a value of **6 mm** to set the depth.
4. Press **OK.**

Note: Notice that by simply creating the arcs, the profiles are automatically created. The geometry we are extruding is the space between the existing geometry and the new arcs.



Step 21: – All Done

1. Congratulations you have finished this lesson and have learned the basics of how to create sketch geometry, dimension sketches, constrain sketches, as well as creating solid models based on sketches. Next we will look at more ways to define geometry in a sketch.

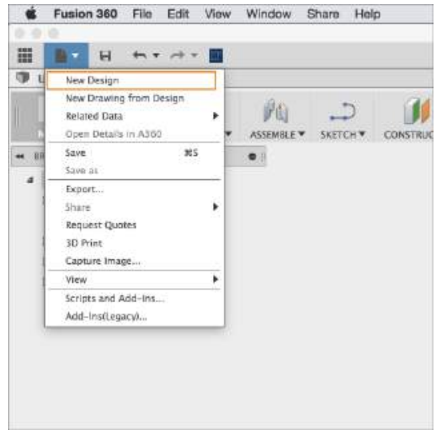
Lesson 2: Creating a sketch

Learning Objectives

1. Create a 2D sketch
2. Use construction geometry
3. More advanced use of constraints

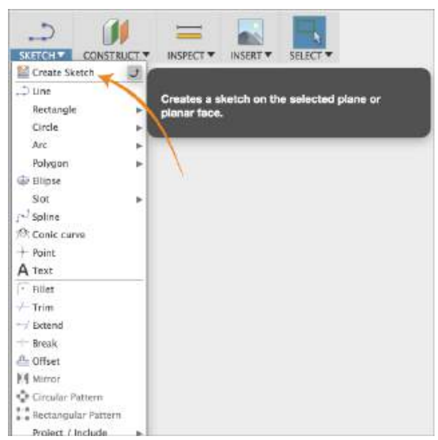
Step 1: Create new Design - Let's start with creating a new design. We're going to use this to create a new part.

1. Launch Fusion 360.
2. Start a new design.



Step 2: – Start the **Sketch** command

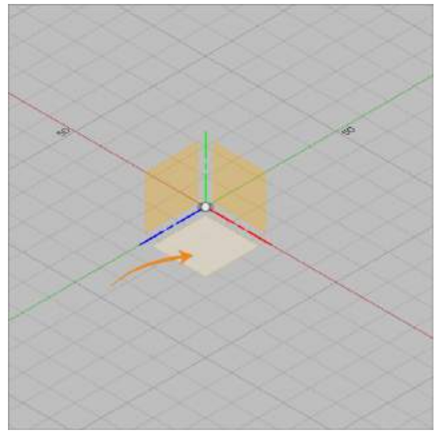
1. Select **Sketch** > **Create Sketch**.



Step 3: – Select the sketch plane

1. You are now prompted to select a “plane” to sketch on.
2. Select the “Front” (XY) plane.

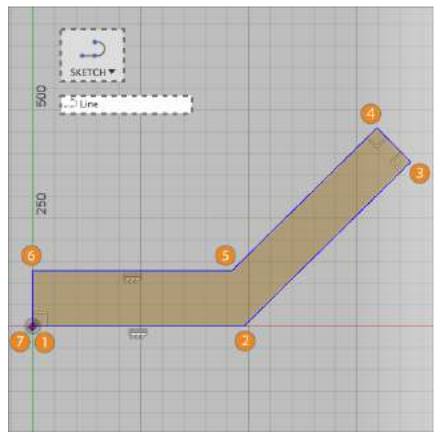
Note: Aside from the origin planes, you can create sketches on one of the 3 default planes, on a custom construction plane or on an existing model face, more on this later.



Step 4: – Create Lines

1. Select **Sketch > Line.**
2. Select the sketch origin.
3. Click to end the line
4. Continue sketching lines as follows
5. Press **esc** to exit the command

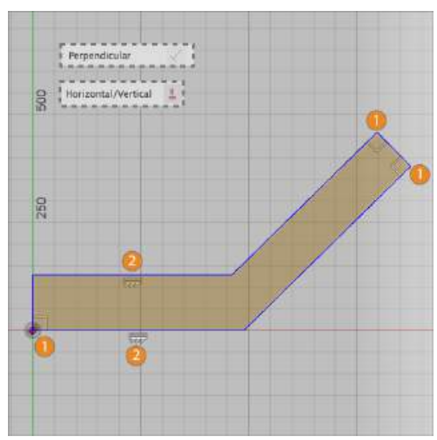
Note: Note as you place the lines some constraints are “automatically” created. If you do not get exactly the same ones don’t worry. Also try to make sure your first line is roughly 500mm it is good practice to sketch shapes “close” to the correct size.



Step 5: – Create Constraints

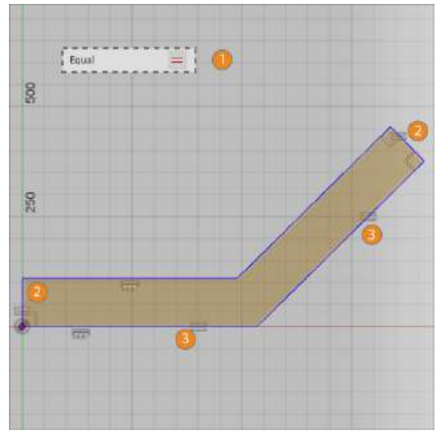
1. Select the **Perpendicular** constraint
2. Select two lines
3. Repeat 3 times (1)
4. Select **Horizontal/Vertical**
5. Select the two lower lines (2)
6. Press **esc** to exit the command

Note: The constraint commands are in the Sketch Palette on the right side of the screen. Some of these constraints may already be created. If they are don’t bother recreating them.



Step 6: – Create Equal Constraint

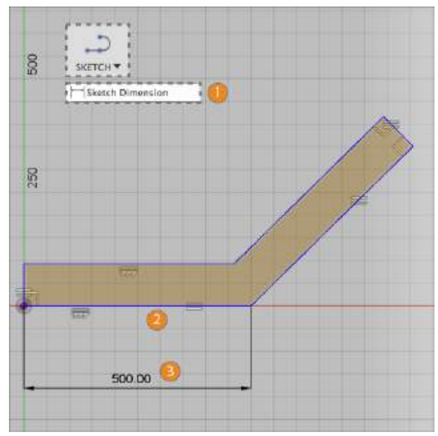
1. Select **Equal constraint**
2. Select the two pairs of lines shown
3. Press **esc** to exit the command



Step 7: – Create Dimension

1. Select **Sketch > Sketch Dimension.**
2. Select the line
3. Click again to place the dimension.
4. Type in a value of 500 mm.
5. Press **Enter** to accept the value.

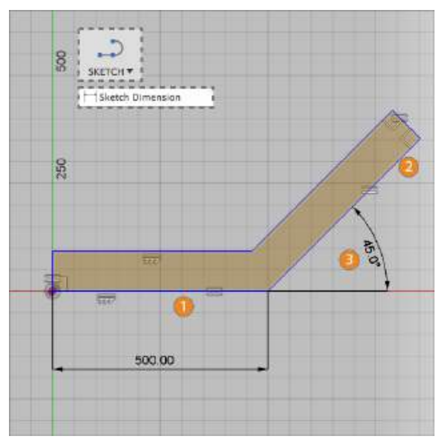
Note: The dimension command is still active and you can go right into placing the next dimensions.



Step 8: – Create Angle Dimension

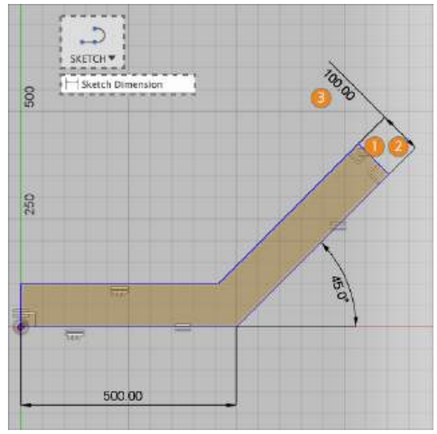
1. Select **Sketch > Sketch Dimension.**
2. Select the bottom line
3. Select the angled line
4. Place the dimension
5. Type 45
6. Press **Enter** to accept the value

Note: The dimension command is still active and you can go right into placing the next dimensions.



Step 9: – Aligned Dimension

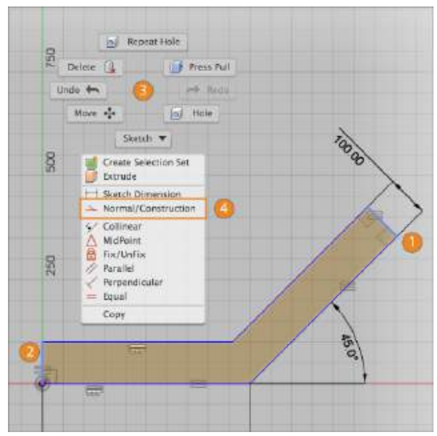
1. Select **Sketch > Sketch Dimension.**
2. Select the upper right line
3. Move just away from the line
4. Notice a small icon on the cursor
5. Select in space again to begin an aligned dimension
6. Type 100
7. Press **Enter** to accept the value
8. Press **esc** to exit the command



Step 10: – Construction Geometry

1. Select Upper line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**

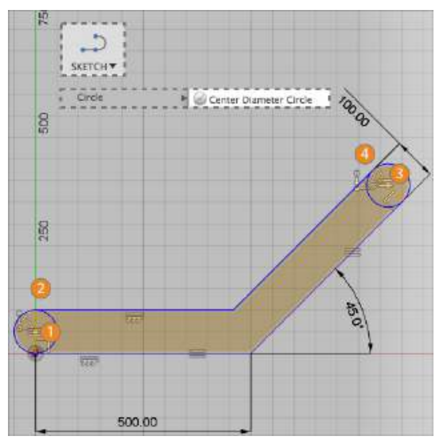
Note: Construction geometry is not considered when looking for profiles. Use construction geometry for reference when creating sketches. It will show as dashed lines to indicate it is construction geometry.



Step 11: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the bottom line midpoint
3. Select the intersection shown
4. Repeat at the top edge
5. Press **esc** to exit the command

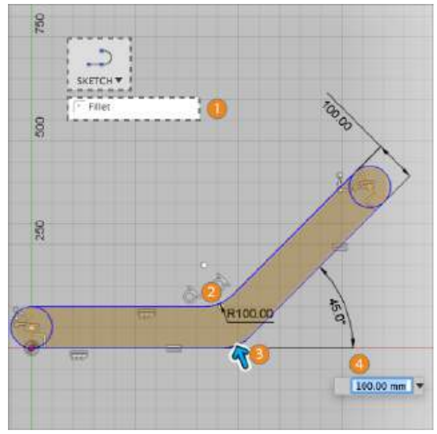
Note: Make sure you "snap" the geometry in place. You should see a midpoint (triangle) constraint created. Try dragging the circles, if they move you missed a snap.



Step 12: – Sketch Fillet

1. Select **Sketch > Fillet**
2. Select the intersection point
3. Select the other intersection point
4. Enter a value of 100 mm
5. Press **Enter** to confirm

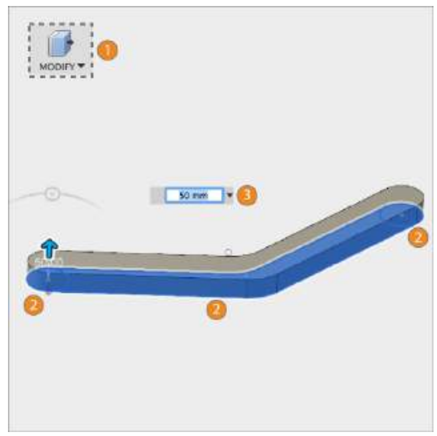
Note: All fillets created at the same time will have an equal radius. Create them separately to have different radius values. Note in many cases creating a fillet in a sketch isn't the best choice. As you see here it deletes the 500 mm dimension.



Step 13: – Extrude the profile

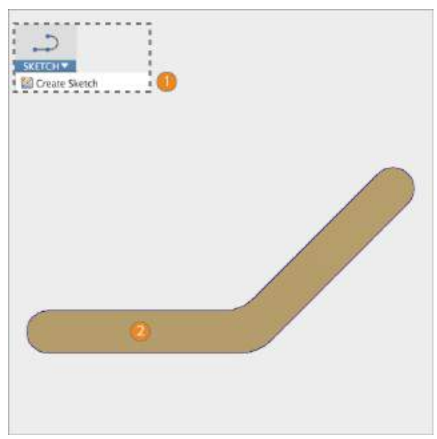
1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow up or type in a value of **50 mm** to set the depth.
4. Press **OK.**

Note: One sketch may contain multiple profiles. Here we limited the number of profiles by using construction geometry.



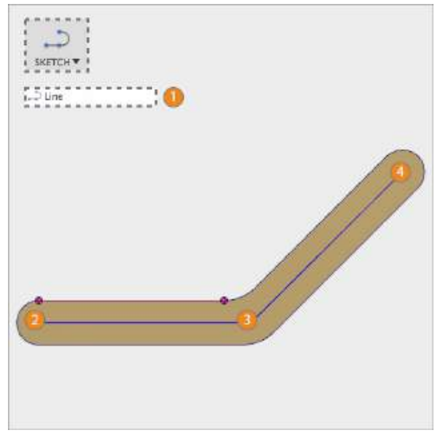
Step 14: – Create another sketch

1. Select **Sketch > Create Sketch.**
2. Select the top of the bracket.



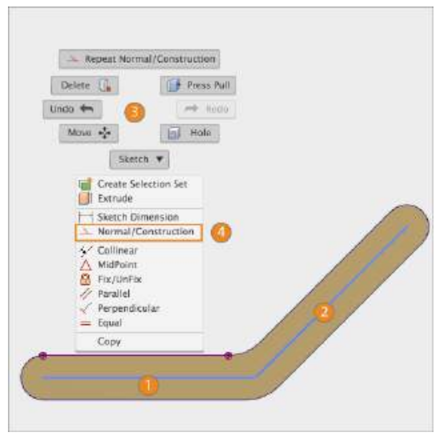
Step 15: – Sketch Lines

6. Select **Sketch > Line**
7. Select the left center point
8. Select near the middle
9. Select the upper center point
10. Press **esc** to exit the command



Step 16: – Construction Geometry

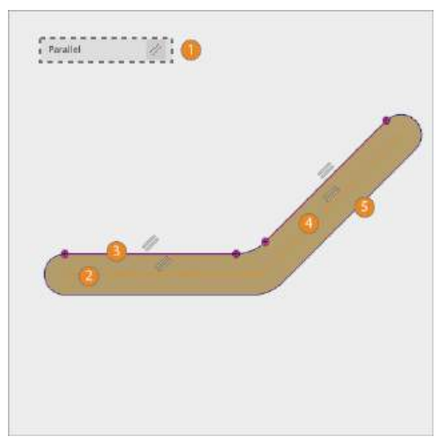
1. Select one line
2. Hold **Shift,** select second line
3. Press **Right Mouse Button**
4. Select **Normal/Construction**



Step 17: – Create Parallel Constraints

1. Select **Parallel**
2. Select lower line
3. Select lower edge
4. Select angled line
5. Select Angled edge
6. Press **esc** to exit the command

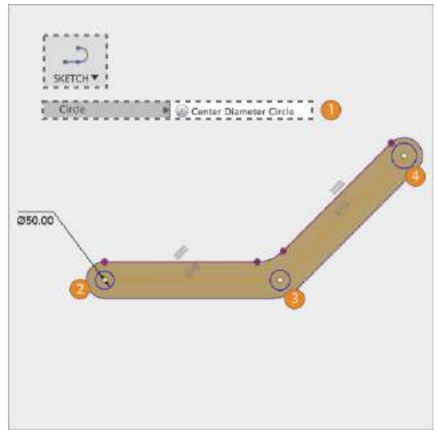
Note: This actually fully constrains these lines. It is worthwhile to learn the different strategies you can take to fully define the sketch lines.



Step 18: – Create 3 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower left center point
3. Type **50mm**
4. Press **Enter**
5. Draw 2 more circles

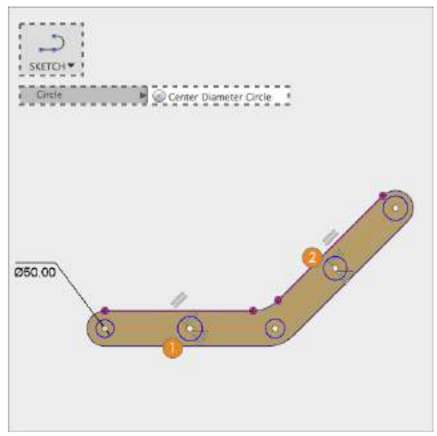
Note: Make sure to “snap” to the end points of the lines when placing the circle center points.



Step 19: – Create 2 Circles

1. Select **Sketch > Circle > Center Diameter Circle**
2. Select the lower line midpoint
3. Click to place the circle
4. Draw 1 more circle at the midpoint of the angled line

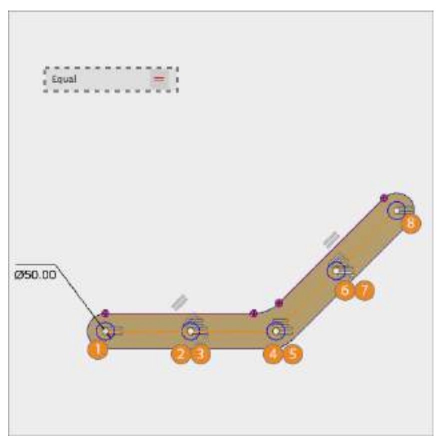
Note: Make sure to “snap” to the midpoints of the construction lines. You should see a small triangle appear to indicate your circle is locked to the midpoint.



Step 20: – Create Equal Constraints

1. Select **Equal**
2. Select 2 circles
3. Repeat until you have all circles equal
4. Press **esc** to exit the command

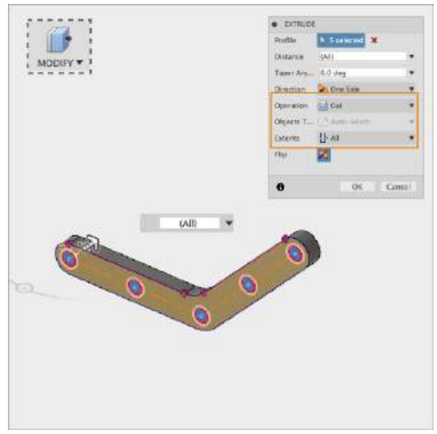
Note: You have to select the circles in pairs. One equal constraint is always applied to two circles. So for each constraint you need to select two circles.



Step 21: – Extrude Circles

1. Select **Modify > Press Pull.**
2. Select the 3 profiles
3. Drag the arrow into the part.
4. Make sure **Cut** is selected
5. Select **All** from the menu
6. Press **OK.**

Note: One sketch may contain multiple profiles. Make sure to select the 5 circles. You can add or remove profiles from a selection later by using the **CMD** key on mac and **CTRL** key on windows.



Step 22: – Lesson complete!

Congratulations you have finished this lesson and have learned more ways to create relationships in a sketch using constraints and construction geometry. In the next lesson, you'll go through the fundamentals of how to further develop 3D models using various modeling tools.

3.1: Sculpting

Sculpting in Fusion 360 allows for the intuitive freeform creation of organic solid bodies and surfaces by leveraging the T-Splines technology. In the Sculpt Workspace, you can rapidly explore forms by simply pressing and pulling on subdivided surfaces. This “hands-on” approach to 3D modeling allows for fast iteration and early stage conceptualization within Fusion 360. Sculpted forms are easily converted to solid bodies, and can be used in conjunction with Fusion 360’s solid modeling commands. Modeling with T-Splines is unlike any other subdivision-modeling tool. One of the main advantages of T-Splines is the ability to add detail only where necessary - a single T-Spline surface can be incredibly smooth, while still having areas of high detail and remaining easy to manipulate.

Lesson 1: Creating T-Spline Forms

Learning Objectives

1. Create a T-Spline Primitive Form
2. Create a T-Spline Revolve Form
3. Create a T-Spline Sweep Form
4. Create a T-Spline Loft Form

Datasets Required

In Samples section of your Data Panel, browse to:

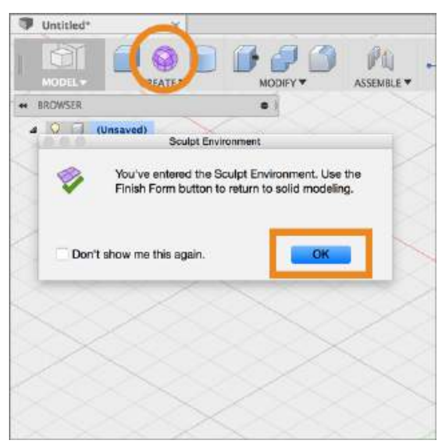
Fusion 101 Training > 03 – Sculpt > **03_Sculpting_Introduction.f3d**

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

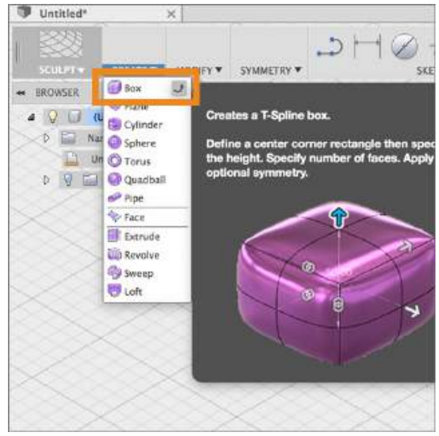
Step 1: Go to the Sculpt workspace – Let’s go to the Sculpt workspace to access the Sculpt tools.

1. In the Model workspace select **Create Form** to enter the Sculpt workspace
2. A dialog box appears, telling you to click **Finish Form** to return to the model workspace when you are finished sculpting.
3. Select **OK**



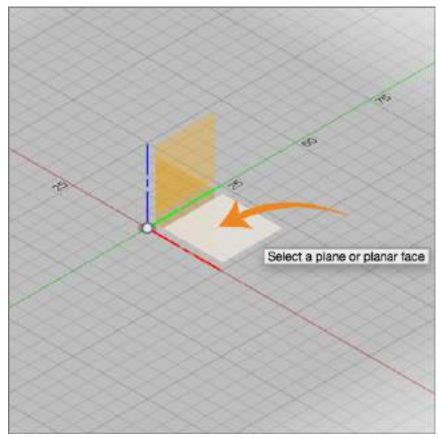
Step 2: – Select the T-Spline Box primitive

1. Select the drop-down arrow under **Create** to expand the list of creation commands.
2. Select **Box** to create a T-Spline box.



Step 3: Position the Box – When you create a new primitive you first need to indicate which plane you want to build on and then enter the dimensions of the primitive.

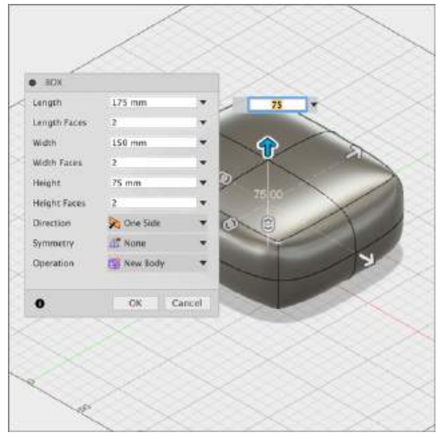
1. Select the bottom plane
2. Select the origin to specify the center point of the box 2D profile
3. Drag the mouse and click on the plane again to specify the initial size of the rectangle.



Step 4: Set the dimensions for the box

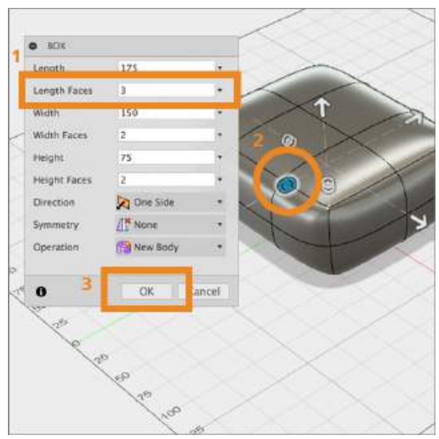
1. In the Box dialog window set the length to **175mm**, and the width to **150mm** respectively by entering these numbers.
2. Drag the arrow pointing up from the box, and set the height equal to **75mm**.

Note: When dragging a manipulator in Fusion 360 the increments for the move are tide to how close or far away the camera is to the manipulator. The closer you are the smaller the move increments, the farther you are the larger the move increments. If you find that the move is changing at too large of increments, simply zoom in to reduce the size of each step.



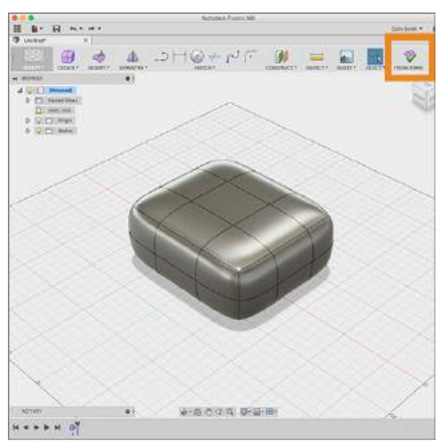
Step 5: Increase the number of faces to the box in Length and Width.

1. In the Box dialog window set the **Length Faces** to **3**
2. To set the width faces, drag the **double-headed arrow manipulator** on the box primitive up to increase the number of width faces until it equals **3**.
3. Select OK to complete the **Box** primitive setup.



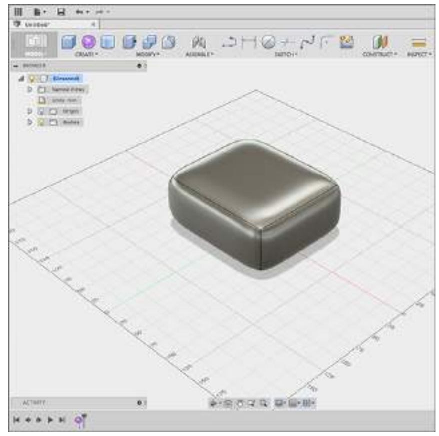
Step 6: Finish the form. You have just created your first T-Spline body. To include this form as part of your solid model you need to indicate that you are done creating T-Spline forms for the moment and that you want to go back to the Model workspace.

1. Select **Finish Form** at the end of the Sculpt workspace toolbar.
2. The T-Spline form is automatically converted to a solid body and you are brought back to the **Model**



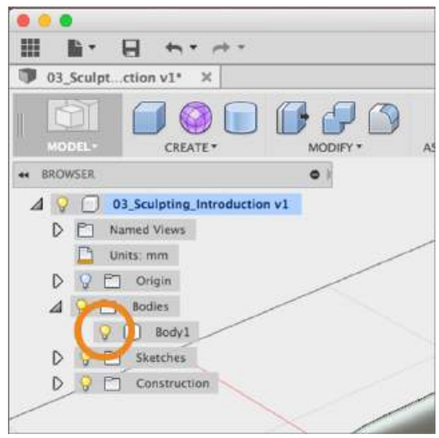
workspace.

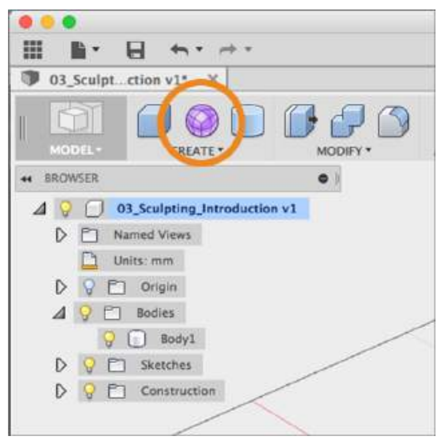
Note: If you create a closed T-Spline form it will be converted to a solid body when you select **Finish Form**. If you create an open T-Spline form, for example a T-Spline Plane, it will be converted to a surface body when you select **Finish Form**.



Step 7: Create a T-Spline Revolve Form. Let's create a T-Spline form by revolving a sketch curve. The Revolve command creates a form by rotation 2D geometry about a fixed axis

1. In the **Browser**, select the drop-down arrow next to your **Bodies** folder
2. Click the light bulb next to **Body1** to turn off the visibility.
3. Select **Create > Create Form**.

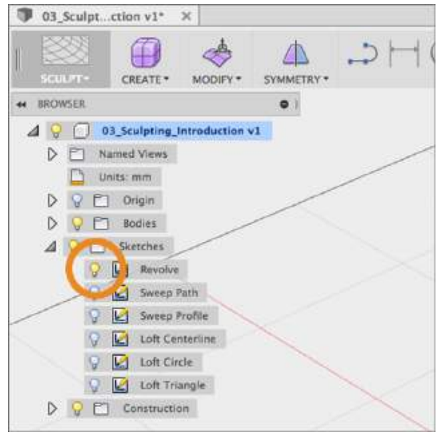




Step 8: – Turn on the visibility of the sketch curve that you will be revolving.

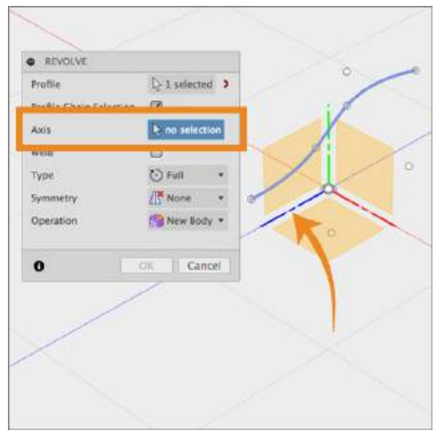
1. In the **Browser**, select the drop-down arrow next to your **Sketches** folder.

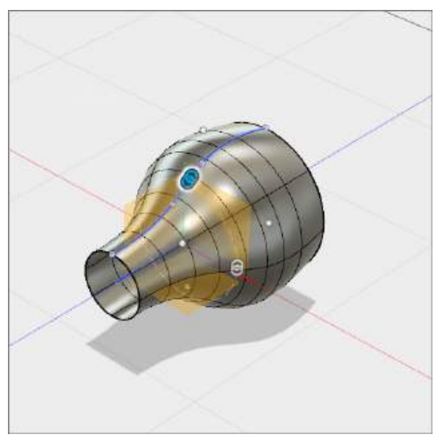
Select the **light bulb icon** next to **Revolve** to turn on the visibility of the **Revolve** sketch.



Step 9: Revolve a T-Spline using the sketch curve as input.

1. Select **Create > Revolve.**
2. Select on the **sketch curve** to identify it as the Profile curve to be revolved.
3. In the dialog window select **no selection** next to **Axis** to show the axis selector in the workspace
4. Select the **Blue** axis. A T-Spline form is revolved 360 degrees around the Blue axis.

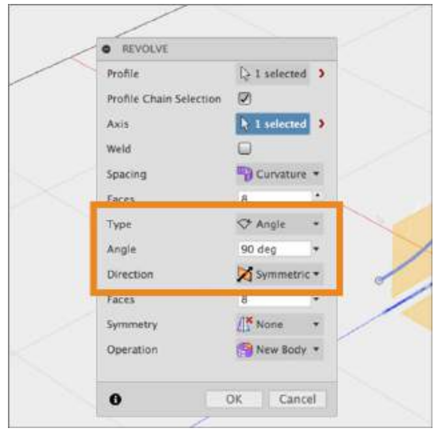


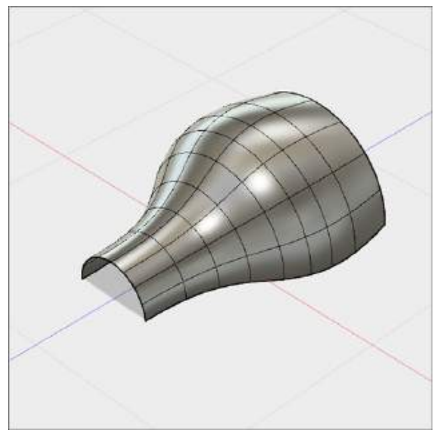


Step 10: Change the Revolve settings. Let's change the settings in the Revolve settings to change the **Angle** the direction that the curve is revolved.

1. In the **Revolve** dialog window, change the **Type** from **Full** to **Angle**.
2. Enter **90 degrees** for the **Angle** field.
3. Change the Direction from **One Side** to **Symmetric**.
4. Select **OK**.
5. Select **Finish Form** to return to the **Model** workspace.

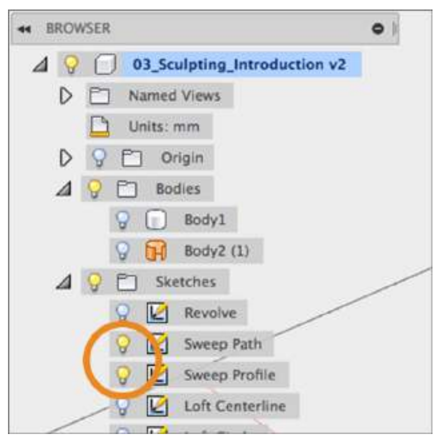
Note: The **Revolve** manipulator can also be used to adjust the number of faces in longitude and latitude of the revolved shape as well as the degree of the **Angle**.





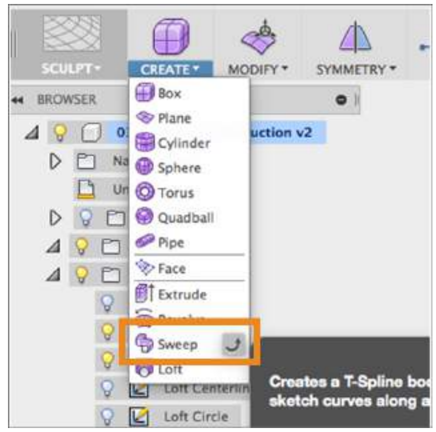
Step 11: Create a T-Spline Form using **Sweep**. The **Sweep** command uses two sketch curves to define a shape. One curve is selected as the **Profile**, which is swept along a **Path** curve to create the shape.

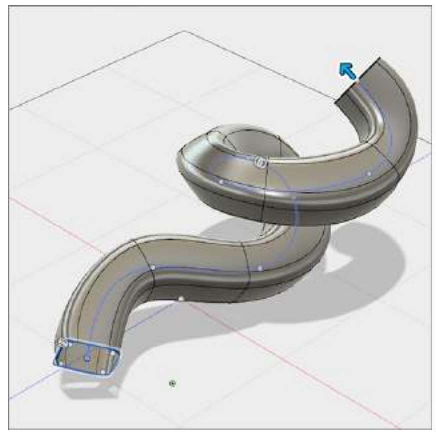
1. In the **Model** workspace hide the previously created **Body 2 (1)**
2. In the Browser in the **Sketches** section select the light bulb icon next to Revolve Sketch to turn off the sketch visibility.
3. Turn on the visibility for the **Sweep Path** and **Sweep Profile** sketches.



Step 12: Start the Sweep command.

1. Select **Create Form** in the **Model workspace** to change to the **Sculpt Workspace.**
2. Select **Create > Sweep** to launch the Sweep dialog window.
3. Select the **Sweep Profile** sketch to identify this sketch as the **Profile**
4. Select **no selection** next to **Path**.
5. Select the **Sweep Path** sketch to identify this sketch as the **Path**. The swept T-Spline Form is generated.

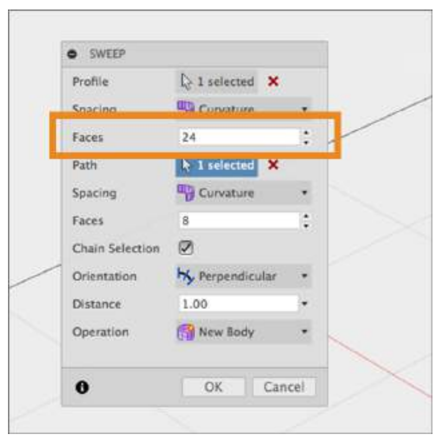




Step 13: Add faces to the swept surface. In order to match our rounded-square profile, we need to increase the number of faces for the profile.

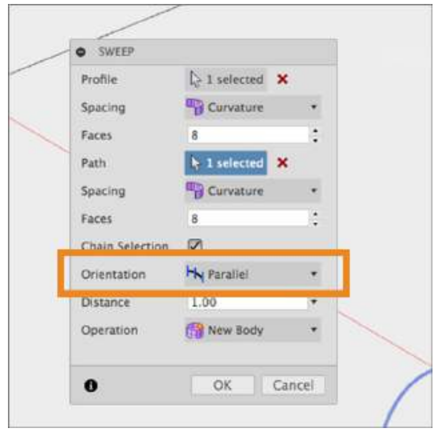
1. In the dialog window, set the number of faces for the **Profile** equal to **24**.

Note: The greater number of faces, the closer the body matches the path.



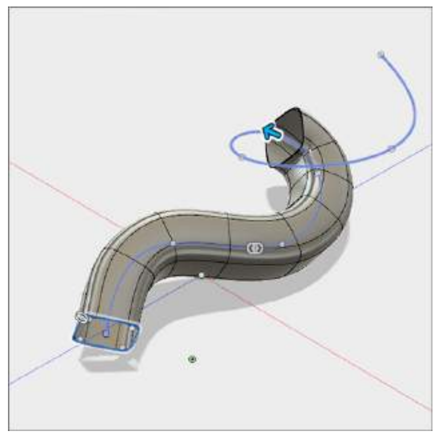
Step 14: Change the sweep orientation. There may be instances where the orientation of the profile as is moves along the path will give you a more desirable result. Use the **Orientation** option to change the sweep behavior.

1. In the dialog window, change the **Orientation** from **Perpendicular** to **Parallel**. Observe how the sweep behavior drastically changes.
2. Change the Orientation **back** to **Perpendicular**.



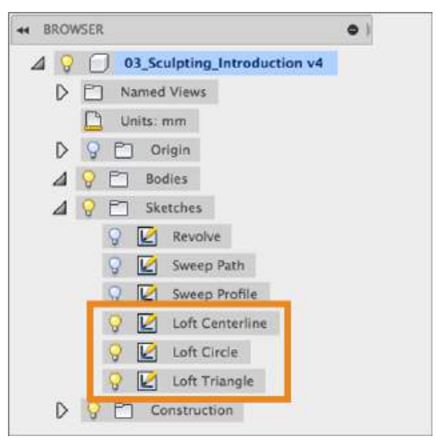
Step 15: Change the sweep distance. The Sweep command allows you to alter the amount of the path curve that is used to create the T-Spline form.

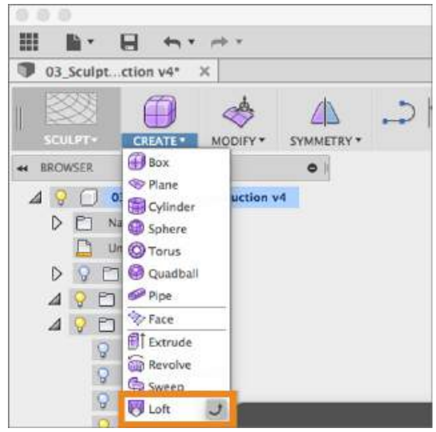
1. Drag the arrow at the end of the **Path** to alter the sweep distance. In addition to using this arrow, we can set the distance in the dialog window.
2. In the dialog window, set the distance equal to **0.5**.
3. The sweep now travels half the length of our Path curve.
6. Select **OK**.
7. Select **Finish Form**.



Step 16: Create a T-Spline Form with **Loft** using 3 sketch profiles as input to create a lofted shape.

1. Turn off visibility of **Body 3 (1)**
2. Turn off visibility of the two **Sweep** sketches.
3. Turn on visibility for sketches labeled **Loft Centerline**, **Circle**, and **Triangle**.
4. Select **Create > Create Form** to change to the Sculpt workspace

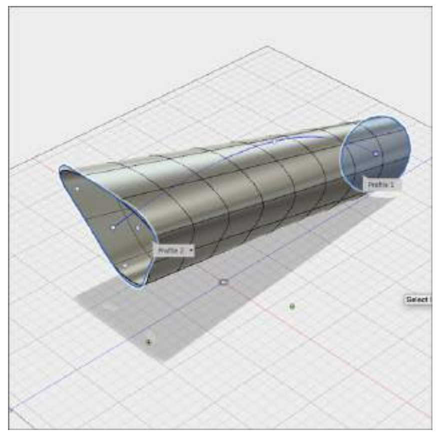




Step 17: Select the profiles to create the loft form.

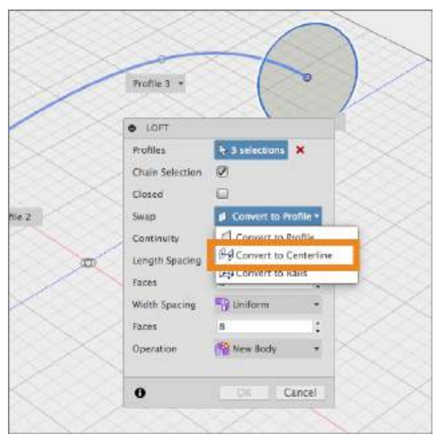
1. Select Create **> Loft.**
2. Click the **Triangular** profile in the canvas.
3. Click the **Circular** profile in the canvas. This creates a straight lofted shape transitioning between the triangle and the circle.

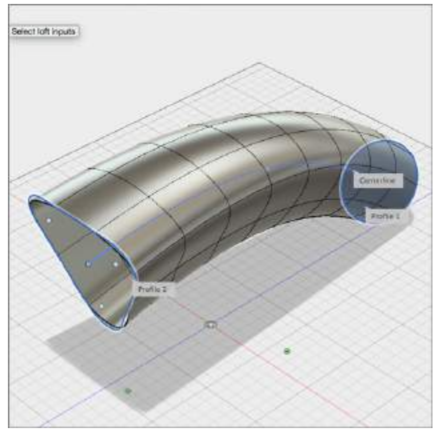
Note: You can have multiple profile shapes to loft between.



Step 18: Change the Loft shape by defining a centerline. By default the loft will always create straight transitional surfaces between profiles. To control the direction of the surface between the profiles you can add a sketch curve a centerline.

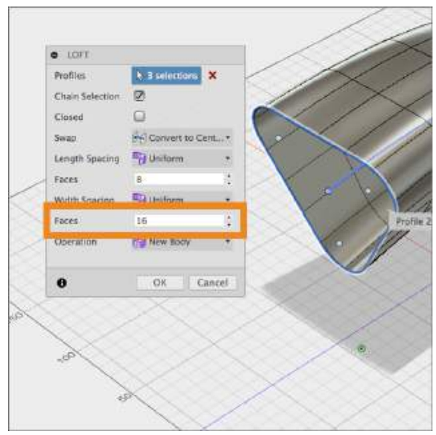
1. Click the **centerline** curve
2. You receive an error because initially the software assumes that you are adding another profile for the loft.
3. To specify this curve as a centerline select **Convert to Centerline** under **Swap** in the dialog window.





Step 19: The T-Spline surface is not matching well to the triangle profile. Let's increase the number of faces to improve the match.

1. In the dialog window, set the number of faces for the **Width** equal to **16**.
2. Click **OK**.
3. Select **Finish Form.**



Lesson 2: Modify a T-Spline Form

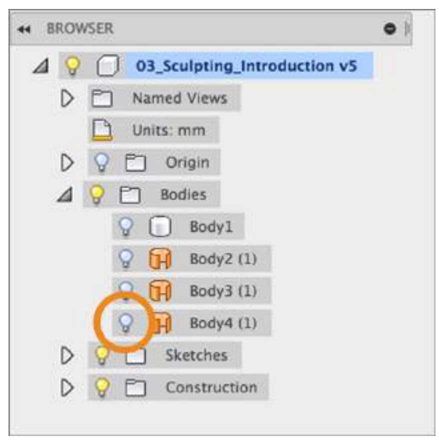
The real power of T-Splines is that it allows for freeform shape manipulation of shapes by moving, rotating and scaling vertices, edges and faces of a T-Spline surface. The most common command you will use to do this is Edit Form. You can also add and remove faces in your forms to get detail where you need it without making the entire form overly complex.

Learning Objectives

1. Move, Rotate and Scale T-Spline geometry with Edit Form
2. Add geometry to a T-Spline body with Edit Form
3. Change the display mode
4. Insert Edges

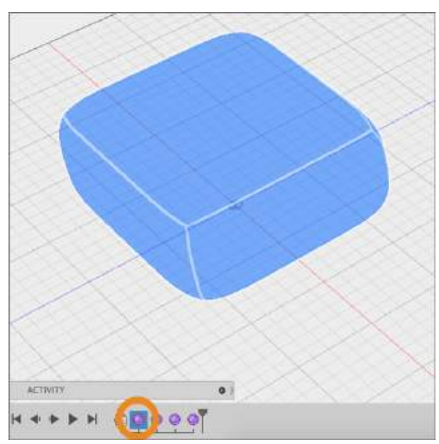
Step 1: Turn off the visibility of the Loft body

1. Turn off visibility of the previously created **Body 4 (1)** from the last lesson.
2. Turn off visibility for any visible sketches.

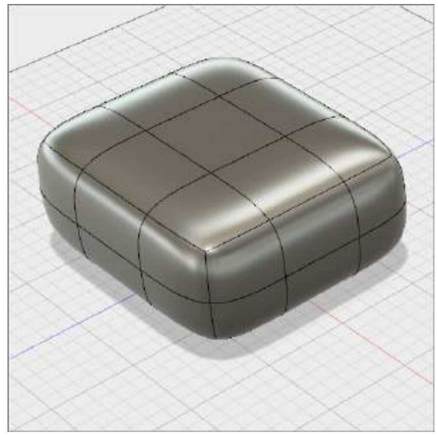


Step 2: Edit the T-Spline shape in the Timeline

1. In the **parametric timeline** at the bottom of the Model window, located the 4 T-Spline form icons.
2. Hover your pointer over the first form icon and you will notice that the primitive box is highlighted on the screen.
3. Right-click on the form icon in the timeline and select **Edit**.



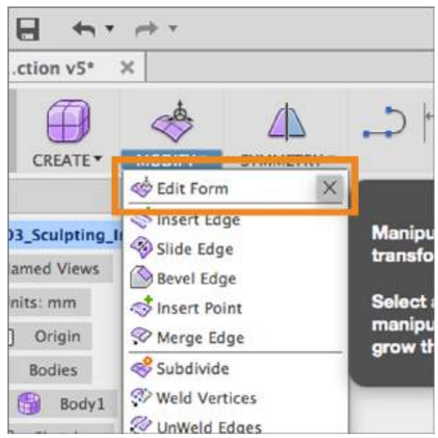
Note: When using Fusion 360 with history turned on, T-Spline operations do not have history in the same way that the operations in the Model and Patch workspaces do. However you can go back in history and make adjustments to your T-Spline shapes and those changes will be recomputed in the history of your model when you exit the Sculpt workspace.

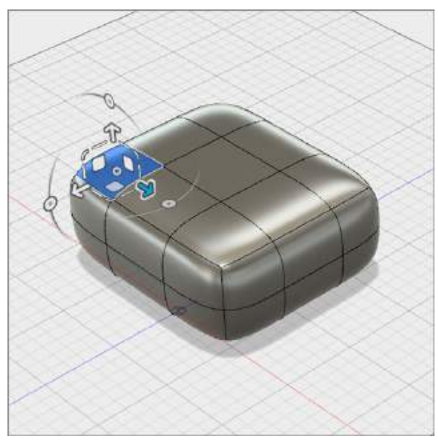


Step 3: Start the Edit Form command

1. Click **Modify > Edit Form**. The Edit Form command can be used to directly manipulate **Faces**, **Edges**, and **Vertices**.
2. Click on a top face located at one of the corners of the box.
3. The manipulator displays on the face with an assortment of tools.

Note: Directly manipulating a face creates the greatest changes in your form, while manipulating a single vertex will result in smaller changes.

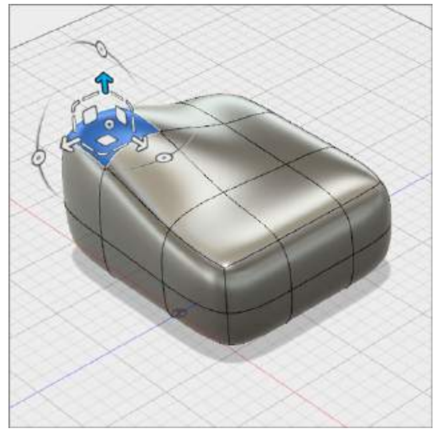




Step 4: Move a face in a single axis direction.

1. Click and drag on the arrow pointing up to translate (move) the selected face up by 30 mm.

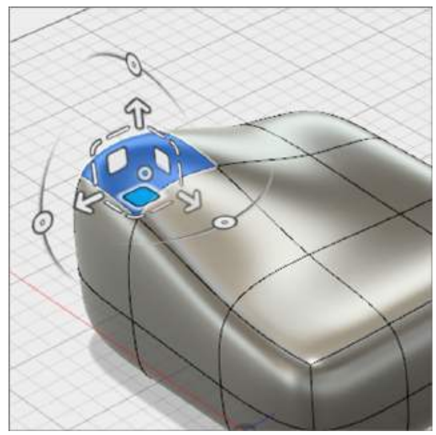
Note: When you move a face on a surface the surrounding faces move to maintain continuity.



Step 5: Move a face in a planar direction.

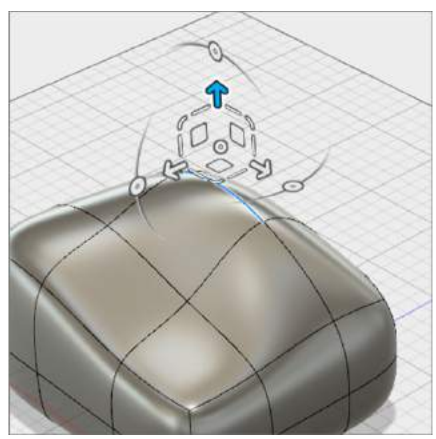
1. Click and drag on one of the **white squares** to translate the selected face on a plane parallel to a given plane in world space.

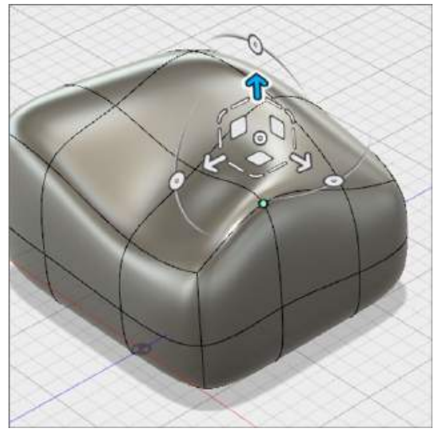
Note: By default the **Edit Form** manipulator uses the World Space coordinates, you can also use View space (based on the camera view) and Local space (based on the normal direction of the surface). These can be changed in the **Coordinate** options in **Edit Form**.



Step 6: Move a T-Spline edge and vertex.

1. Select a single **edge**.
2. Use any of the translate manipulators to compare the effect.
3. Select a single **vertex**.
4. Translate this vertex to see how this creates more subtle changes

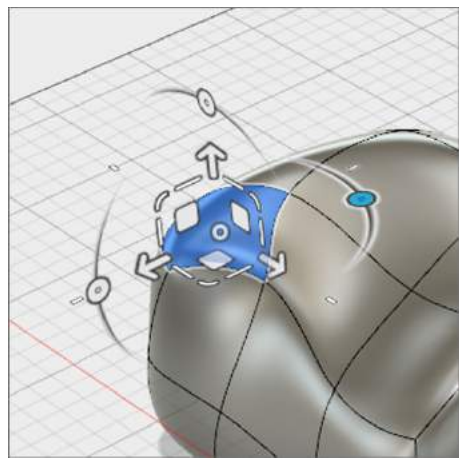




Step 6: Rotation with Edit Form

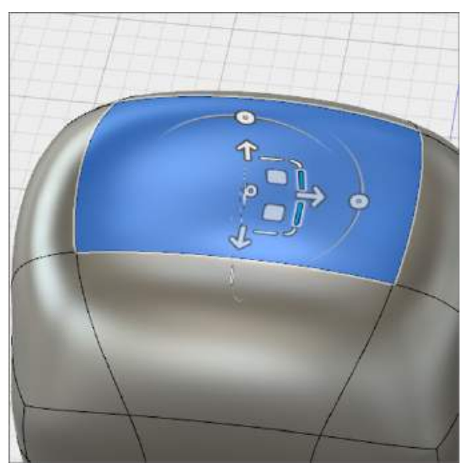
1. Make sure that at least one **face** is selected.
2. Click and drag one of the **circular arcs** to rotate the selected geometry about a single axis.

Note: Be careful not to rotate geometry too far, as self-intersecting faces, or geometry that twists through itself will result in not being able to convert it in to a solid body.



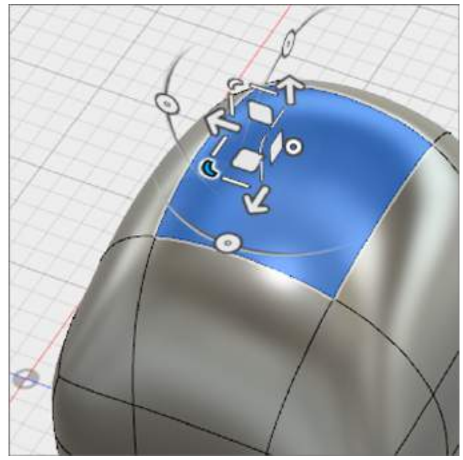
Step 7: Single direction scaling

1. Make sure that at least one **face** is selected.
2. Click and drag one of the **straight lines** to scale the selected face in one direction.



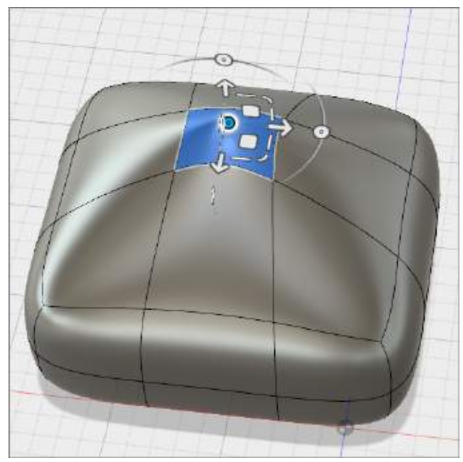
Step 8: Planar scaling

1. Make sure that at least one **face** is selected.
2. Click and drag one of the small **corner manipulators** to scale the selected face along a plane – or in two directions.



Step 9: Universal scaling

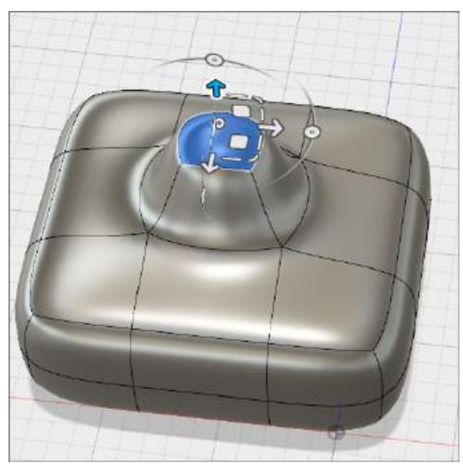
1. Make sure that at least one **face** is selected.
2. Click and drag the **circular manipulator** at the center of the manipulator.
3. Dragging **left** or **right** will scale the geometry in all directions **up** or **down**.



Step 10: Extrude a face to add geometry

1. Select a **single face** of your T-Spline form.
2. With the Edit Form command still active, hold the **alt-option** on Mac or **alt** key on Windows.
3. Click and drag the **single arrow** to add geometry outward from the selected face.
4. Let go of the **alt-option/alt** key as well as the left-mouse button.
5. Select OK to close **Edit Form**

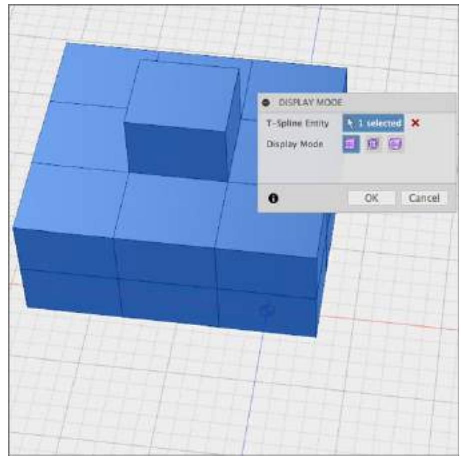
Note: You can use this hotkey function to extrude any edge(s) or face(s) that are selected on a T-Spline form.

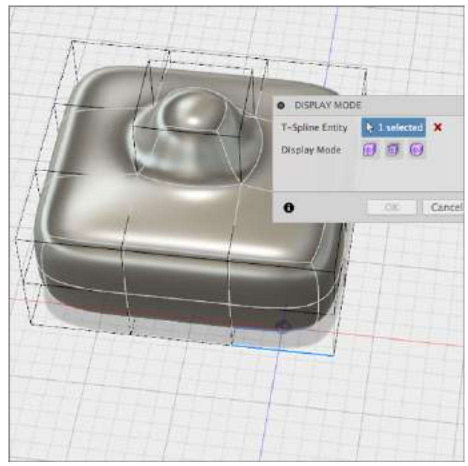


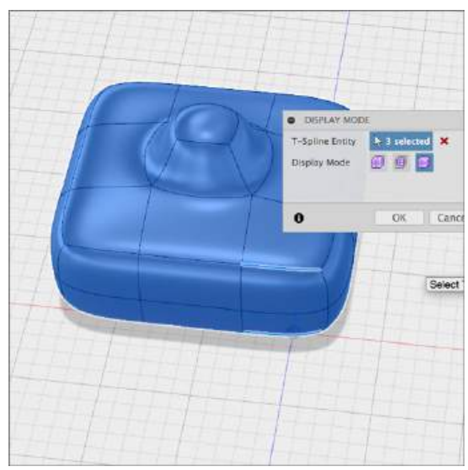
Step 11: Change display mode. When modeling with T-Splines you have 3 different display modes to choose from. Changing the display mode can help you find problem areas on your model and increase performance by not having to constantly smooth the T-Spline shape. For this example we will switch the display to Box Mode – a polygon version of the T-Spline form.

1. Select **Modify > Performance > Display** Mode to bring up the Display Mode window
2. In the Display Mode window select **Display Mode > Box Display** to show a unsmoothed version of the T-Spline form.
3. Select **Display Mode > Control Frame Display** to see a combination of the unsmoothed and smoothed versions of the T-Spline.
4. Select Display Mode > Smooth Display to go back to the smoothed version.
5. Select **Display Mode > Box Display**
6. Select **OK** to close the window

Note: Display mode can also be switch by using the combination of **Control + 1 for Box**, **Control +2 for Control Frame** and **Control + 3 for Smooth** on a Mac. Or **Alt +1**, **Alt +2**, **Alt +3** respectively on a PC. You can also find these controls in the **Selection Options** section of the **Edit Form** window.

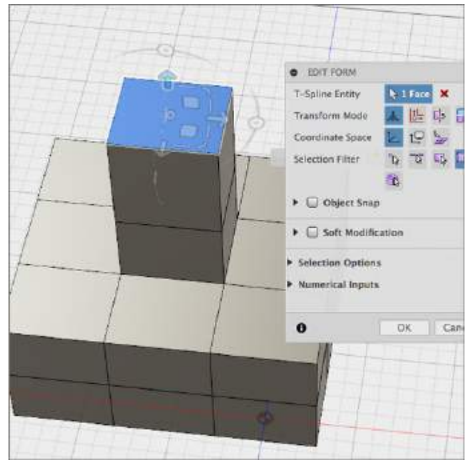






Step 12: Working in Box Display Mode.

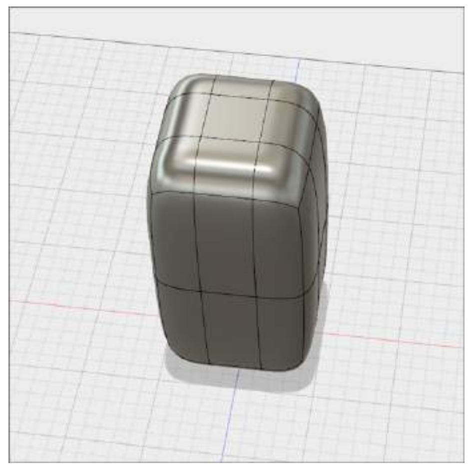
1. Hold down the **right mouse button** to bring up the radial menu.
2. Select **Edit Form** in the radial menu
3. Select a face and hold **alt-option/alt** while dragging the blue arrow up to extrude the face. You will notice that the performance is faster in Box mode that in Smooth mode.
4. Hold **control/alt +3** to return to smooth mode.
5. Hold down the **right mouse button** and select **OK** from the radial menu to close the **Edit Form** window.
6. Select **Finish Form** from the menu bar to go back to the Model workspace.

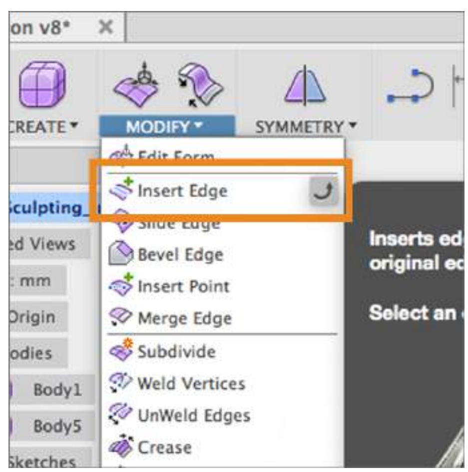


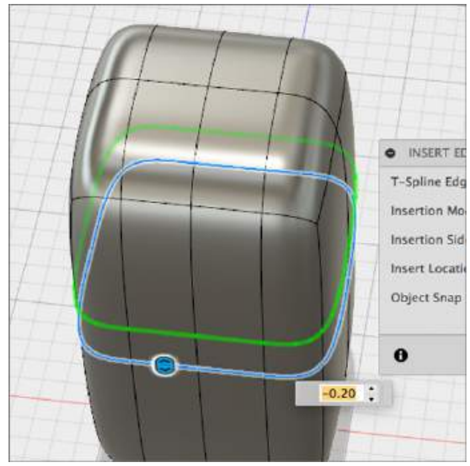
Step 13: Add edges to a T-Spline form

7. Enter the Sculpt workspace by selecting **Create > Create Form**.
8. Create another **Box** primitive whose length, width, and height are **100mm, 100mm**, and **200mm.**
9. Set the number of length, width, and height faces equal to **4, 2,** and **2.**
10. Click **OK**.
11. Click **Modify** > **Insert Edge**.
12. **Double-click** on one of the middle edges to select the entire middle loop.

Note: Double clicking on an edge will select the complete edge loop. Selecting a single edge inserts an edge to a face on either side of the selected edge. Selecting an edge loop adds a second loop.



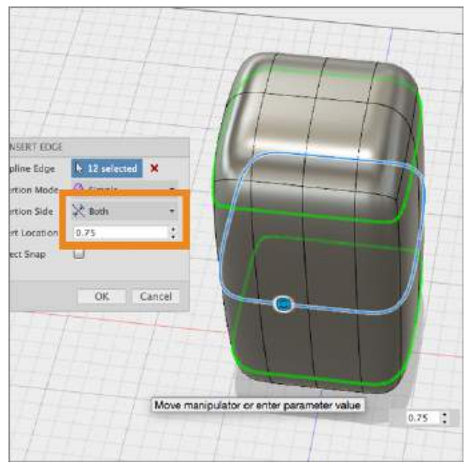




Step 14: Adjust the insert location

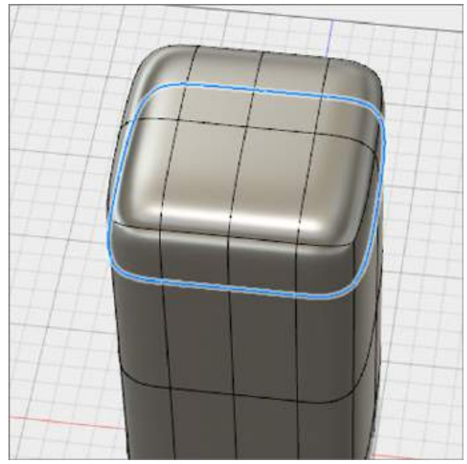
1. Click and drag the double arrow to adjust the position of the inserted edge.
2. By hand, or with the text field, set the **Insert Location** equal to **0.75**.
3. Change the **Insert Side** from Single to **Both**.
4. Click **OK**.

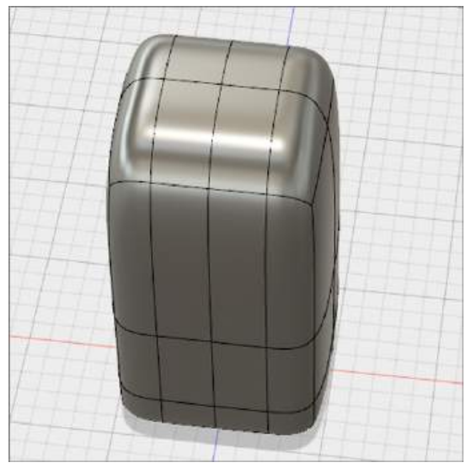
Note: You probably noticed that the shape of the T-Spline form changed after you inserted additional edges. The top and the bottom of the box became sharper by adding more edges near the existing top and bottom edges. If you want to insert an edge(s) and not change the shape of the T-Spline form, change the **Insertion Mode** to **Exact**.



Step 15: Delete and edge

1. Select the recently added the **upper edge loop**.
2. Select **Modify > Delete** or press the **Delete** key on your keyboard.
3. Select **Finish Form**.





Lesson 3: Create a T-Spline Form using a reference image

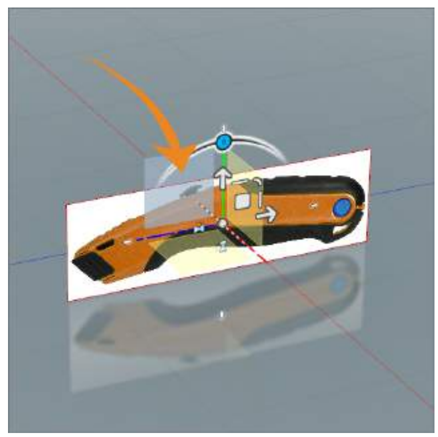
When sculpting a shape with T-Splines it is helpful to have a reference image to guide you. Reference images can be plan view sketches or photographs that are set in the background of the workspace to model from. In Fusion 360 you can attach an image to a work plane and then calibrate the image so that you are modeling in the correct scale. In this lesson we will use a side view photograph of a utility knife as reference to sculpt a T-Spline body that will be the outer shape body of the knife.

Learning Objectives

1. Insert and image in to the workspace using **Attach Canvas**
2. Use **Calibrate** to set the proper scale for the reference image
3. Invoke **symmetry** when modeling a T-Spline box
4. Use **Insert Point** to draw edges on a T-Spline face(s)

Step 1: Attach a canvas

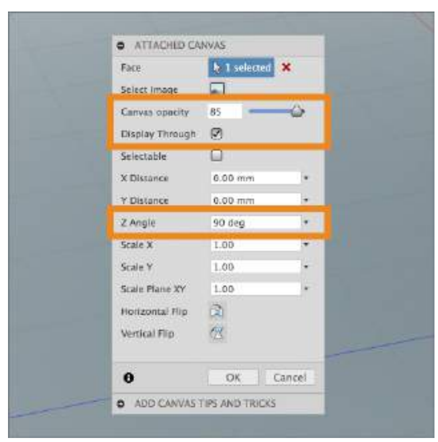
1. Select **Insert > Attached Canvas**.
2. Select the **YZ Plane** (between the green and blue axis) to indicate which plane the image should be attached to.
3. In the dialog window, click the **Select Image** button and navigate to the **03_UtilityKnife.jpg** file.



Step 2: Adjust canvas settings

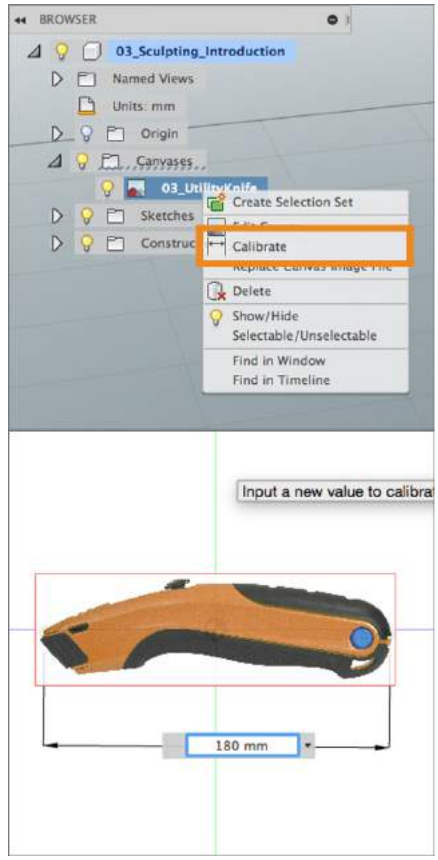
1. If necessary, **rotate** the canvas **90 degrees** to orient it properly
2. Lower the opacity to **85**.
3. Check the box for **Display Through** to ensure that the canvas can be seen through your T-Spline form.
4. Click **OK**.

Note: You don't need to be concerned about the size and scale of the image at this point. We will adjust the scale using the **Calibrate** tool. Calibrating the image ensures that you are modeling in the correct scale in the workspace.



Step 3: Calibrate the image.

1. In the Browser, click the drop-down arrow next to the **Canvases** folder.
2. Right-click on **UtilityKnife** and select **Calibrate**.
3. Click **Right** on the ViewCube to view the utility knife from the side.
4. Click once at the **front** of the utility knife.
5. Click once at the **back** of the utility knife.
6. Enter **180 mm** in the length field and hit enter.
7. The canvas will scale up accordingly.

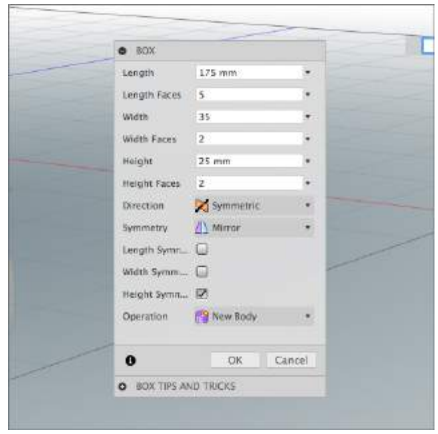


Step 3: Create a box primitive

1. Click the **Create > Create Form** icon to enter the Sculpt workspace.
2. Click **Create > Box**.
3. Select the same side plane (YZ) as the canvas to specify the plane that the Box is placed on.
4. Click once at the **origin** to specify the Box's center point.
5. Move the mouse and **click** again at the outer edge of the reference image to draw its 2D profile.

Step 4: Set box dimensions and add symmetry

1. Set the Box's **Length, Width,** and **Height** equal to **175, 35,** and **25 mm**, respectively.
2. Set the number of Length Face equal to **5**, and the width and height faces equal to **2**.
3. In the dialog window, change the **Symmetry** from None to **Mirror**.
4. Check the box for **Height Symmetry**.
5. Select **OK**.



Step 5: move faces to the top of the knife image.

1. If you are not in the right side view still, click on the **Right** side of the **View Cube.**
2. Select **Modify > Edit Form**.
3. Select the middle set of faces by holding the **left mouse button** and dragging to the lower right over top of the faces you want to select.
4. Using the **Planar Translation** manipulator, move the selected faces to align the top of the T-spline body with the top of the utility knife image.

Step 6: move faces to the top of the knife image.

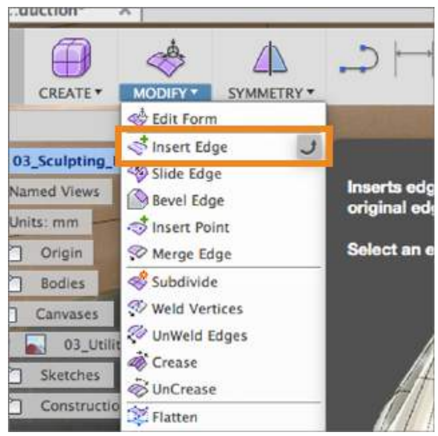
1. To align the bottom in this section, select the **bottom face** and use the **planar translation** and **rotate manipulators**.

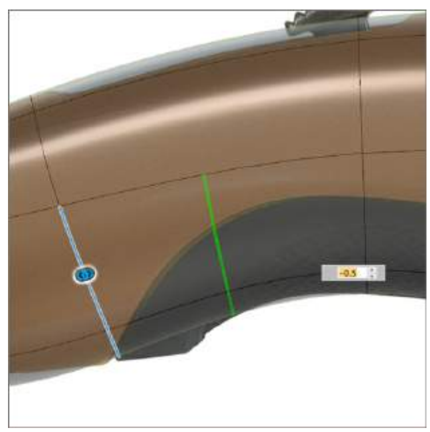
5. Repeat the previous 2 steps for the rest of the T-spline form. It will also be helpful to use the **single-direction scale** manipulator in some cases.
6. For more controlled editing, try modifying individual edges.

Step 6: Insert edges to get closer to the knife shape. Our T-Spline form is starting to resemble our reference image but there aren't enough edges in the T-Spline to capture all the detail of the Knife.

1. Hold **Shift** then select the edges shown at the front of the knife.
2. Click **Modify > Insert Edge**.
3. Drag the direct manipulator to the right to position the new edges at an **Insert Location** around **-0.5**.
4. Click **OK**.

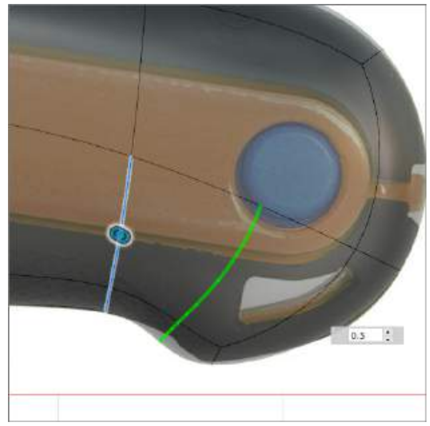
Note: With Symmetry enabled you only need to select the edges one side of the symmetry plane, the matching ones on the other side will be automatically selected also and will be displayed in yellow.





Step 7: Insert additional edges with **Insert Edge**.

1. Holding **Shift** then select the edges shown at the back of the knife.
2. Select **Modify > Insert Edge**.
3. Drag the direct manipulator to the right to position the new edges at an **Insert Location** around **-0.5**.
4. Click **OK**.



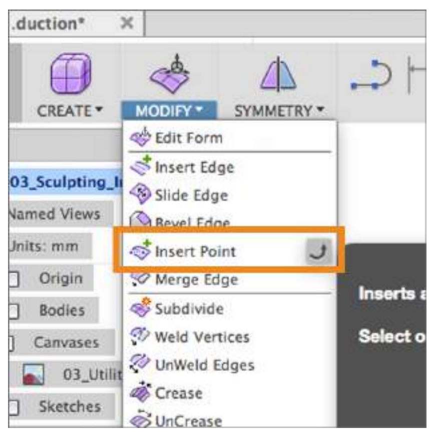
Step 8: Move the inserted edges with **Edit Form**.

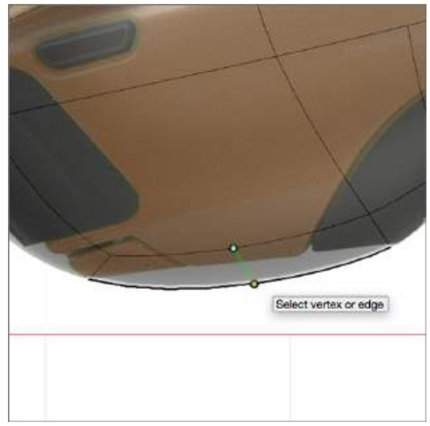
1. Use the **Edit Form** command to manipulate the recently inserted edges to achieve the result shown in the image.

Note: The planar translation manipulator will be extremely useful.

Step 9: Use **Insert Point** to draw edges on a face. To insert the final two edges we need, we'll use the **Insert Point** command. Slightly different from Insert Edge, the **Insert Point** command will easily insert an edge by connecting two points together.

1. Click **Modify > Insert Point**.
2. Hover over the middle of the top edge shown until a **red circle** appears – this indicates the midpoint
3. **Click** and repeat for the edge directly beneath, located along the line of symmetry.
4. Ensure the **Insert Mode** set to **Simple**.
5. Click **OK**.





Step 10: Use **Edit Form** to move faces and edges until you have matched the T-Spline body to the profile of the Knife image.

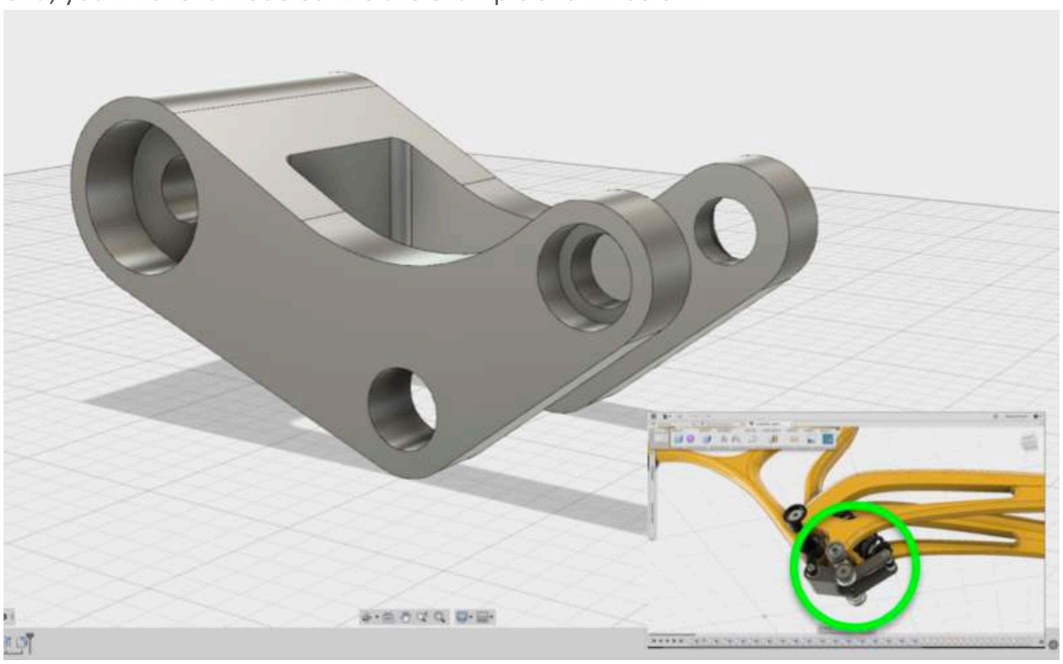
1. Use the **Edit Form** command to manipulate the recently inserted edges (as well as the surrounding geometry) to get the T-Spline primitive to match as closely as possible.
2. When you are satisfied with the shape of the T-Spline body select Finish **Form** from the Sculpt Menu bar.

4.1: Modeling

3D Modeling is a key process of getting your ideas from a concept to a read-for-manufacture state, making it core foundation of the product development process. In Fusion 360, there are a couple different ways you can start a design. Chapter 02 and 03 showed you how to start with sketches and with sculpted bodies. This lesson will continue where the previous chapters left off and walk you through the fundamental modeling techniques based sketches, based on a sculpted body, explore these different design approaches, and learn tips and tricks along the way.

Lesson 1: Modeling based on Sketches

We'll be using a sketch of a mountain bike rocker arm to go through this lesson. At the end, you'll have it modeled like the example shown below.



Learning Objectives

1. Creating geometry based on sketches
2. Using sketch lines as reference
3. Using sketches to drive changes in geometry

Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 04 – Modeling > **04_Model_from_sketch**

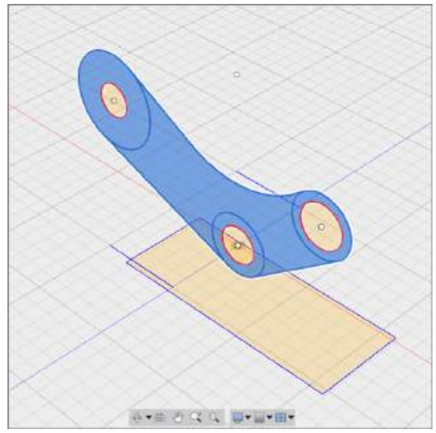
Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

Step 1: Select profiles - Let's start with this sketch of the rocker arm. We're going to use this to create a solid body.

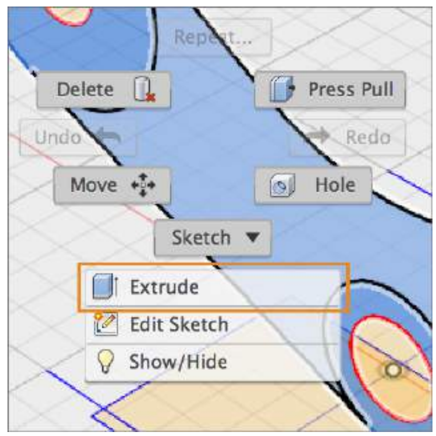
1. Hold down **Shift** and select the profiles shown in the image. Make sure that the 3 center holes are the only profiles not selected.

Note: If you are having trouble selecting certain profiles, use your mouse wheel and zoom in closer; this should make it easier to select.



Step 2: – Start the **Extrude** command

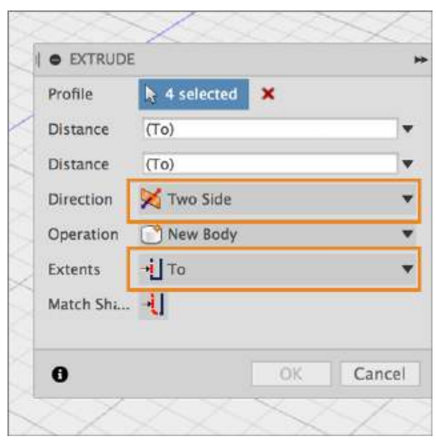
1. Right click on a selected area of the sketch and select **Extrude**. We're going to extrude the selected profiles.



Step 3: – Define the extrude options in the Extrude dialog box

1. Set Direction to **Two Side**.
2. Set Extents to **To**.

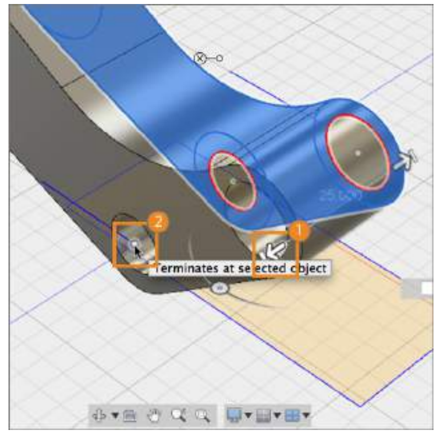
You should now see 2 arrows appear on your selected sketch profiles. We're going to use these arrows to define where we want the extrusion to go. This is especially useful when you have set geometry you can use as reference, much like our sketch here.



Step 4: – Set the distance for the left side

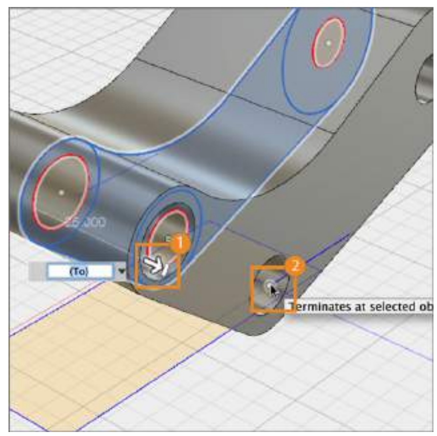
1. Click once on the **left arrow manipulator.**
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.

Note: When Extrude extent is set to "To", make sure you select the line sketch and not the rectangle sketch. When you've done this, the extrusion will automatically terminate at that point. This is selection really useful when you have reference geometry you want to use to create new geometry.



Step 5: – Set the distance for the right side

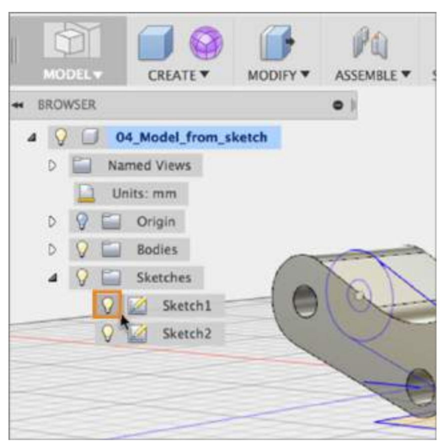
1. Repeat the same thing on the right side. Click once on the **Right arrow manipulator.**
2. Now hover over the **line sketch** on the left side and click on the **end point** as the extent you want to extrude to.
3. Click **OK** (or hit **ENTER**) to finish the command.



Step 6: – Turn sketch visibility back on

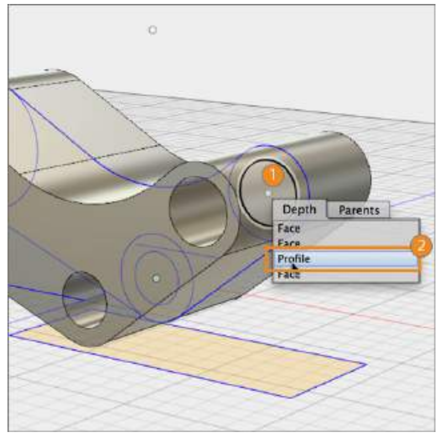
1. Let's go to the browser and within the Sketches folder, click the light bulb icon next to **Sketch1** to turn the visibility of that sketch back on.

The Visibility of a sketch is automatically turned off after a modeling action has been committed based on that specific sketch.



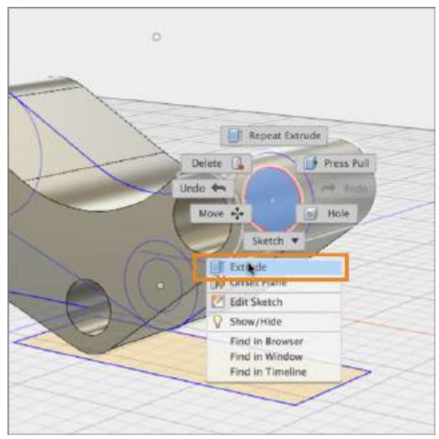
Step 7: – Select a sketch profile behind an obstruction

1. Now we need to select the circle profile to make an extrusion. If you find yourself in this situation where it is hard to select a specific geometry because it is being obstructed, then **hover** over the profile, **click** and **hold**. After a few seconds, you'll see a dialog menu show up, letting you choose what exact entity you'd like to select.
2. Select **Profile**. You should now see the circle profile selected.



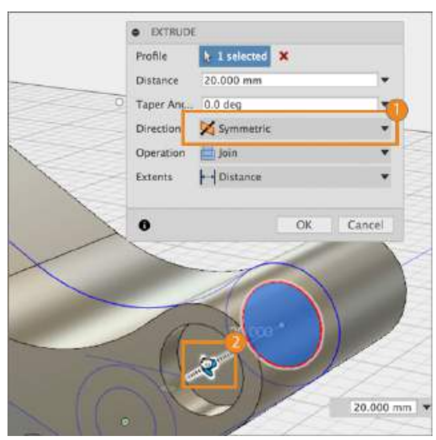
Step 8: – Extrude the circle profile

1. Right click on the selected circle profile and pick **Extrude**. We're going to create this command again to create new geometry.



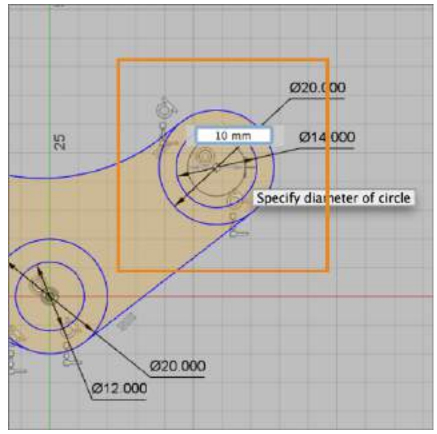
Step 9: – Join the new extruded body

1. Set the Direction to **Symmetric.** Leave Operation as Join.
2. Drag the arrow to **20 mm.** Click OK to finish.



Step 10: – Sketch a new circle profile

1. Right click on **Sketch1** in your browser and select **Edit Sketch.** This will take you back into the first sketch and create more geometry. Notice that the timeline reflects us going back to this sketch item as well.
2. Go to the Sketch drop-down menu and select **Circle** > **Center Diameter Circle**
3. Create a circle snapped the center with a diameter of **10 mm.** Click **Enter** twice to commit. Click **Stop Sketch** to exit out of sketch mode.

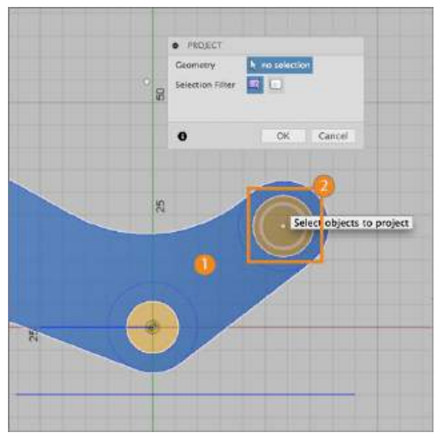


Step 11: – Project the circle onto a new face

Go to the Sketch drop-down menu and select **Project / Include > Project**.

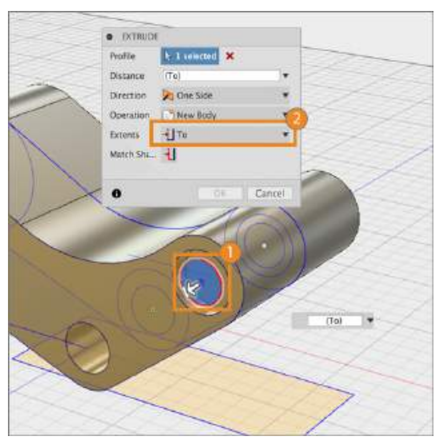
1. First select the outer face,
2. Then select the new circle sketch we just created.
   Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

You should now see that the circle is now project onto the outer face of the model.

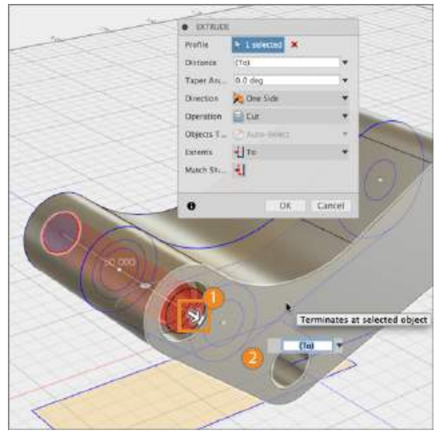


Step 12: – Extrude the circle as a cut

1. Let's select the project circle profile, right-click and choose **Extrude**.
3. In the command dialog, change the Extents to **To**.



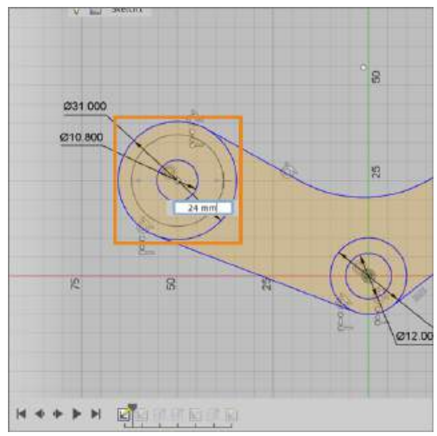
Step 13: – Make the cut



1. Click on the **Arrow Manipulator** to activate the extrusion.
2. Rotate the model to the other side so that we see the other face we want to extrude to. Click on that face and click OK to finish.

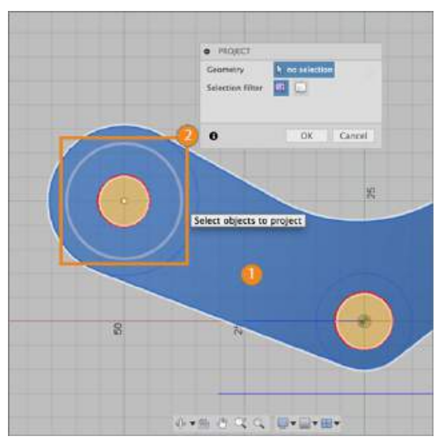
You should now see a cut made through the entire width of the model. This cut is now tied to the original circle sketch, thus allowing us to easily make dimension changes moving forward.

Step 14: – Sketch a new circle profile



1. We're going to move to the other side of the rocker model. Right click on **Sketch1** in your browser and select **Edit Sketch.**
2. Go to the Sketch drop-down menu and select **Circle** > **Center Diameter Circle**
3. Create a circle snapped the center with a diameter of **24 mm.** Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

Step 15: – Repeat Project sketch workflow



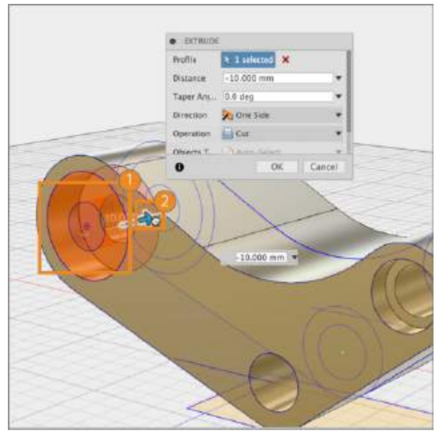
Go to the Sketch drop-down menu and select **Project / Include > Project**.

1. First select the outer face,
2. Then select the new circle sketch we just created.
   Click OK to finish, and **Stop Sketch** to exit out of the sketch mode.

You should now see that the circle is now project onto the outer face of the model.

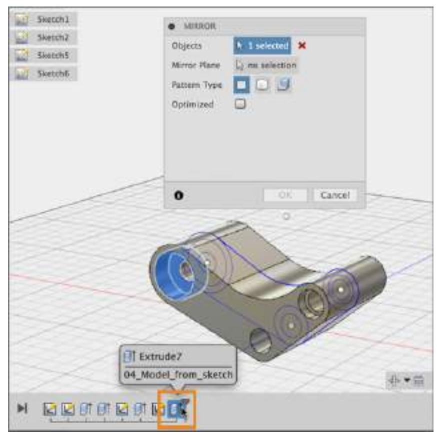
Step 16: – Extrude the circle as a cut

1. Select the area between the project circle and the smaller circle, right-click and select **Extrude**.
2. Drag the **Arrow Manipulator** to **-10 mm**. Click OK to finish.



Step 17: – Mirror the cut on the other side

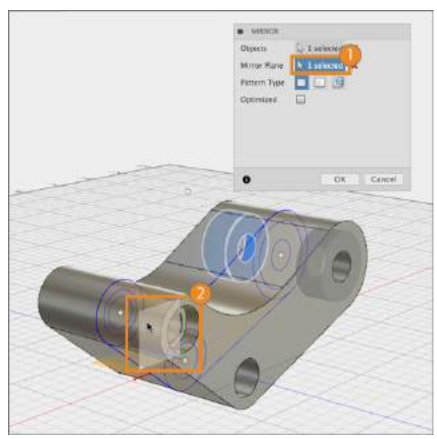
1. Now that we've made this cut, let's mirror it on the other side. Go to the Create drop-down menu and select **Mirror**.
2. Go to timeline and select the last extrusion as the object we want to mirror.



Step 18: – Mirror the cut on the other side

1. Click the **Mirror Plane** option to activate which mirror plane to use.
2. Select the origin plane that is in the middle of the model. Click OK to finish.

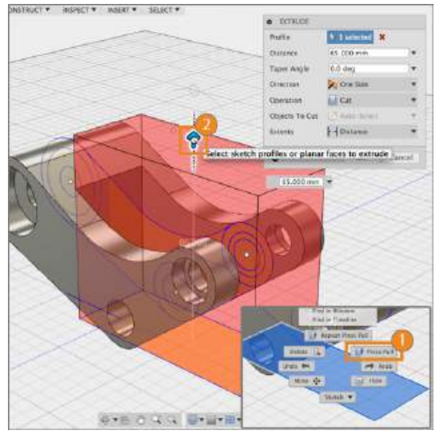
Note: If you have having trouble selecting the origin plane, remember to zoom out or click and hold to get the option to choose what you'd like to select.



Step 19: – Use Press-Pull to cut

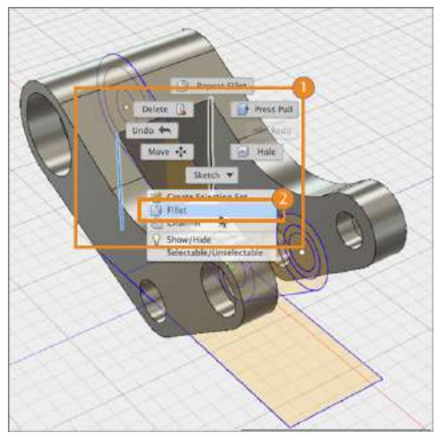
1. Now let's select the rectangle sketch at the bottom, right-click and use **Press-Pull** (on the right of your cursor)**.**
2. Drag the **Arrow Manipulator** through the model so the cut goes all the way through.

Notice that Press-Pull automatically turned into an Extrude command. This is the nature of Press-pull – it adapts to what the action is gives you a predictable outcome. If you had selected an edge and decided to use Press-Pull, it'll automatically turn into a Fillet.



Step 20: – Add a couple fillets

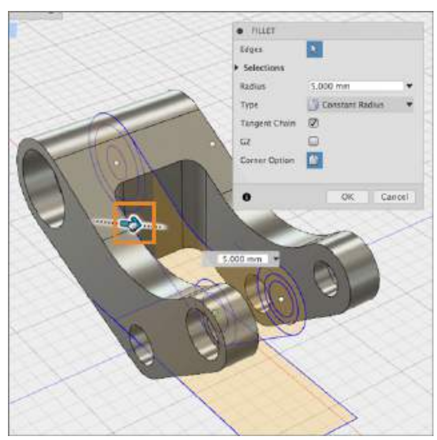
1. Let's finish the model by adding a couple of fillets on the inside edges. Select them by holding **Shift**.
2. Right-click and select **Fillet**.



Step 21: – Add a couple fillets

1. Drag the **Arrow Manipulator** to **5 mm**

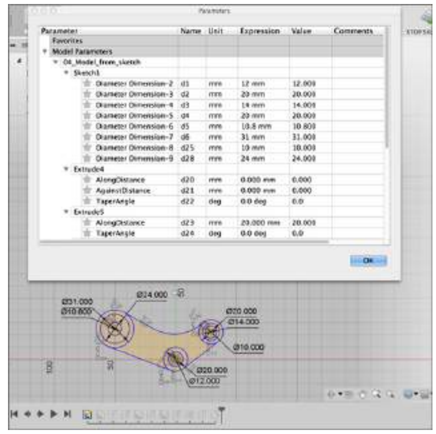
   Click OK to finish.



Step 22: – Making changes to your model

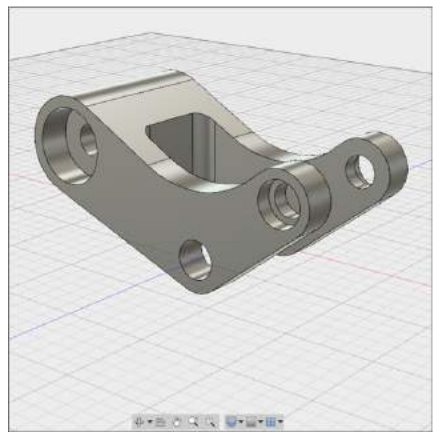
Since all the extrusions, mirror, and fillets are based on the original sketches, we can go back to **Sketch1** and **Sketch2** and any time and make dimension changes without needing to change each downstream feature or worry about any of them failing.

You can also go to the Modify drop-down menu and select **Change Parameters**. This will allow you to change any dimension in a chart form, assign custom names, set values or functions, and see the changes update instantly.



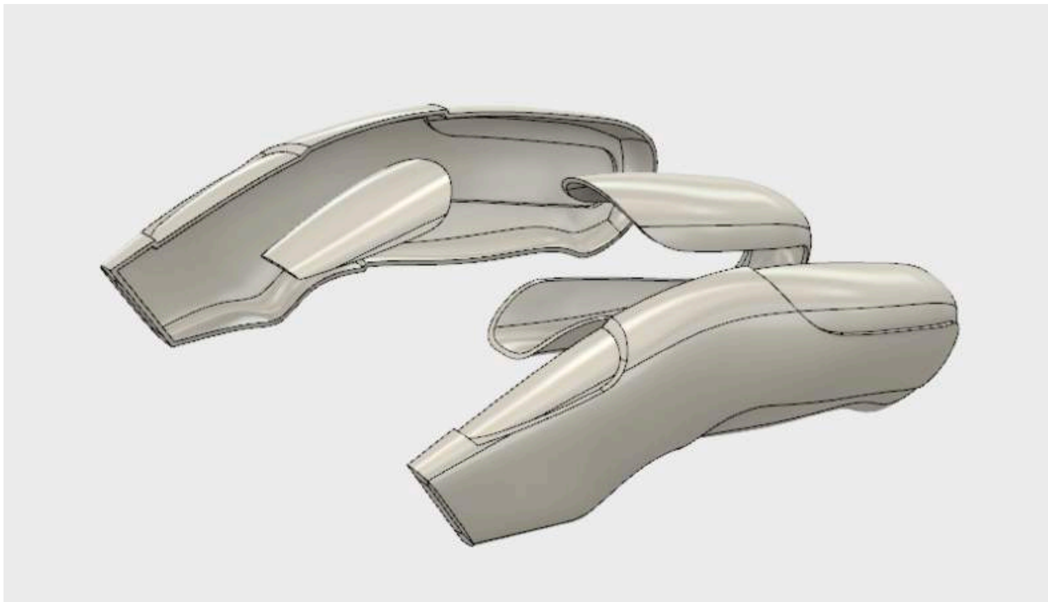
Step 23: – Model complete!

Congratulations, you have completed this lesson on how to model based on sketches! You're ready to move on to the next lesson.



Lesson 2: Modeling based a sculpted body

Now that you've seen how to use model based on sketches, we'll take it one step further and go through how to take advantage of sketches and model geometry based on an existing sculpted body. We'll be using a sculpted utility knife model. At the end of this lesson, you'll have gone from a single model to 4 separate pieces like the example shown below.



Learning Objectives

1. Using sketches and planes to split bodies
2. Using bodies to join and cut other bodies

Datasets Required

In Samples section of your Data Panel, browse to:

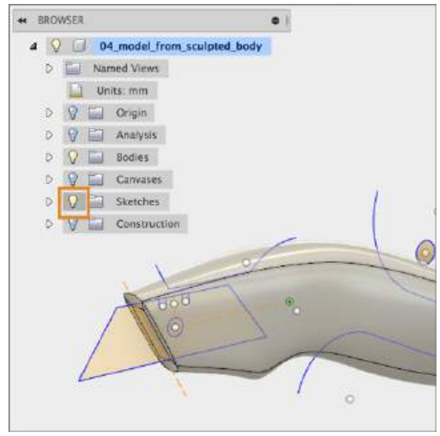
Fusion 101 Training > 04 – Modeling > **04_Model_from_sculpted_body**

Open the design and follow the step-by-step guide below to get started with the lesson.

Step-by-step Guides

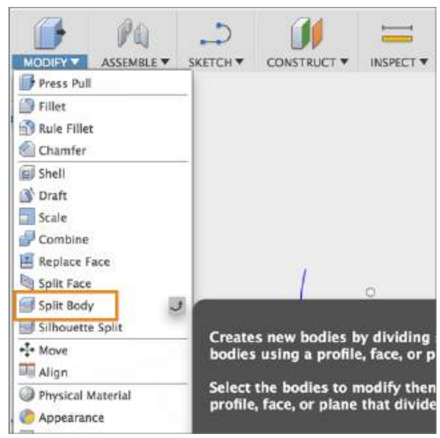
Step 1: – Turn on Sketch visibility

1. Let's start by going to the browser and locating the **Sketch** folder. Click the light bulb to turn on sketch visibility. You should now see a number of sketch lines and profiles appear on your model.



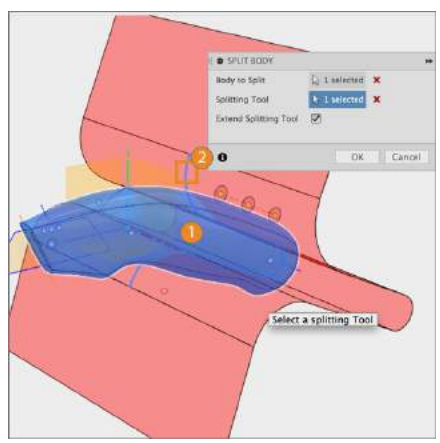
Step 2: – Split the model into 2 bodies

1. In order to create the handle grips, we're going to use a couple of the sketch lines to split the model into 3 separate bodies. Go to the Modify drop-down menu and select **Split Body**.



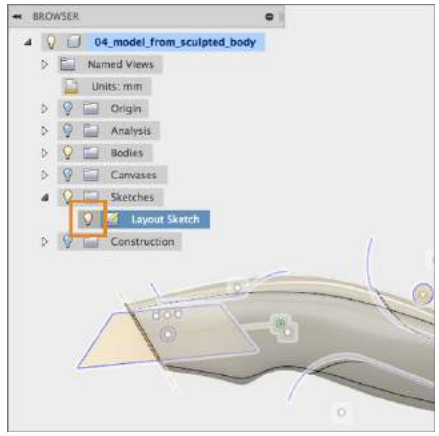
Step 3: – Make the split

1. Select the **body** as the **Body to Split.**
2. Click on **Splitting Tool** to activate the selection. Select the **long grip line sketch** as the **Splitting Tool**. Click OK to finish the split.



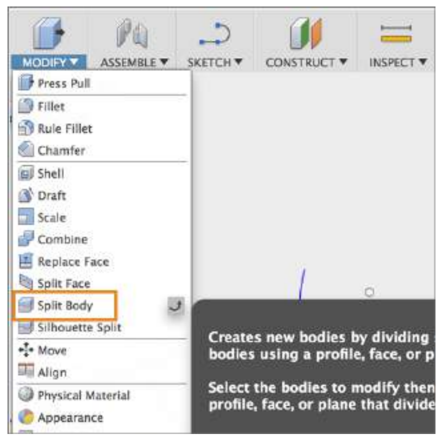
Step 4: – Repeat previous step

2. Let's repeat the previous step and make another split – this time it'll be for the grip at the top of the knife model. Go back to the Sketch folder and turn on **Layout Sketch** visibility.



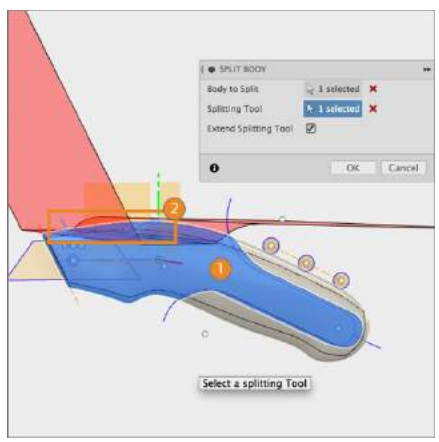
Step 5: – Split the model into 3 bodies

2. Go to the Modify drop-down menu and select **Split Body**.



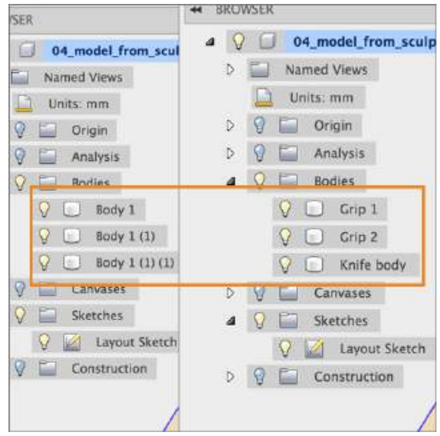
Step 6: – Make the split

3. Select the **body** as the **Body to Split.**
4. Click on **Splitting Tool** to activate the selection. Select the **short grip line sketch** as the **Splitting Tool**. Click OK to finish the split.



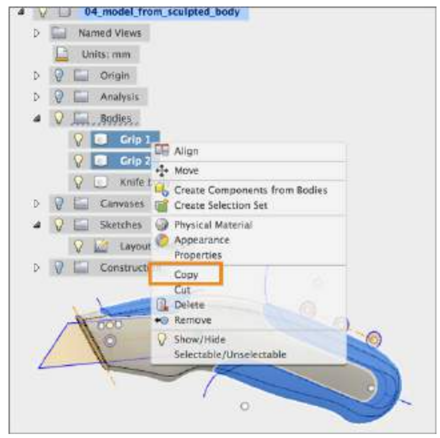
Step 7: – Rename bodies

3. In the Bodies folder, you'll notice that there are now 3 bodies. Let's rename them. **Double click** on the body and rename:
   Body 1 to **Grip 1**
   Body 1 (1) to **Grip 2**
   Body 1 (1) (1) to **Knife body**



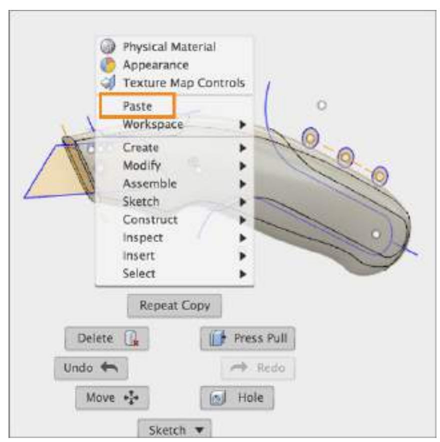
Step 9: – Copy and paste bodies

1. We're now going to perform a modeling technique. Select **Grip 1** and **Grip 2**, right-click and select **Copy.**



Step 10: – Copy and paste bodies

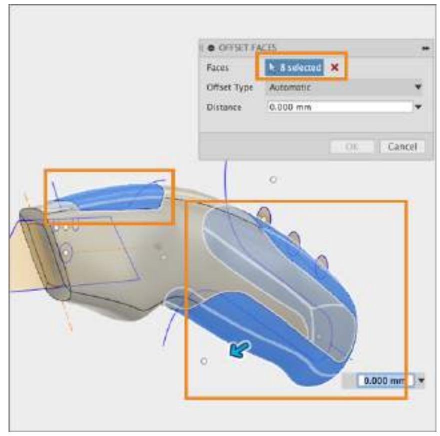
1. Click somewhere on the canvas, right-click to activate the marking menu, then select **Paste.**
2. You should now see 2 more bodies appear in your Bodies folder called **Grip 1 (1)** and **Grip 2 (1)**. Click OK to finish the paste action.



Step 11: – Offset bodies with Press-Pull

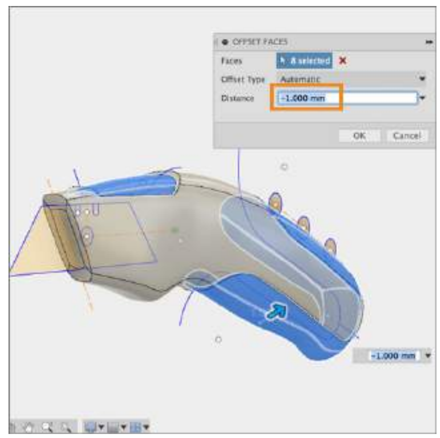
1. Turn off visibility of **Grip 1** and **Grip 2.** We're going to work on the 2 new grip bodies.

2. Right-click somewhere in canvas and select **Press-Pull**. Select all the faces of **Grip 1 (1)** and **Grip 2 (1)**. Make sure to rotate around and get all the faces. Hold **Shift** to add onto the selection. You should have a total of 8 faces selected.



Step 12: – Offset bodies with Press-Pull

1. Enter an Offset Distance of **-1 mm.** Click OK to finish. You should see that the faces have been successfully offset.

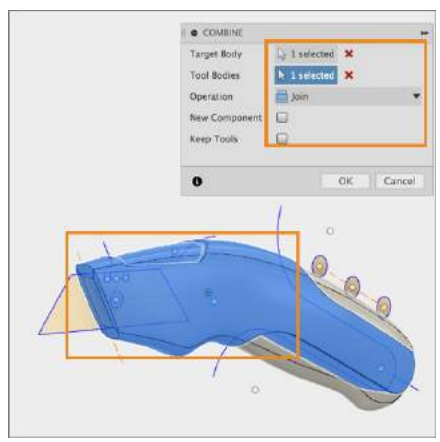


Step 13: – Join bodies with Combine tool

1. Let's now join these offset bodies to the knife body so that they are part of a whole body. Go to the Modify drop-down menu and select **Combine.**

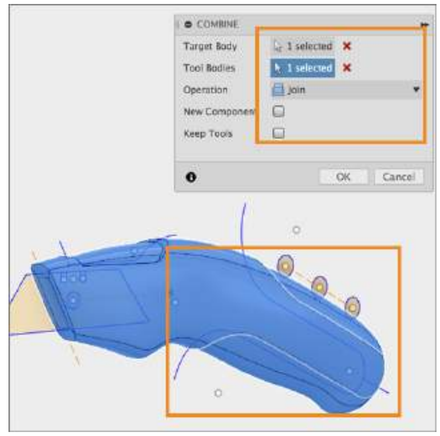
2. In the command dialog, set:
   Knife Body as the **Target Body**,
   Grip 2 (1) as the **Tool Body,**
   Join as the **Operation,**
   Uncheck Keep Tools,

3. Click OK to finish the operation.



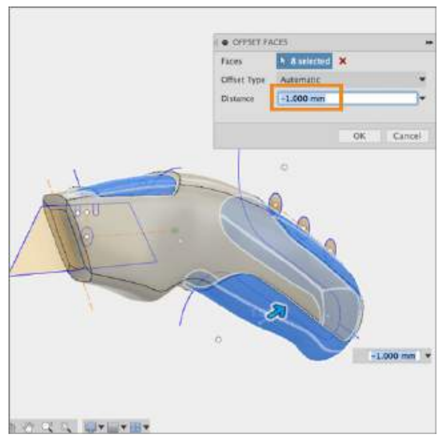
Step 14: – Repeat Combine Join

1. Repeat the previous step, but this time; join **Grip 1 (1)** to **Knife Body**. Click OK to finish the operation.
2. You should now only see **Grip 1, Grip 2,** and **Knife Body** in your Bodies Folder in the browser.



Step 15: – Cut Bodies with Combine tool

1. Now let's use the new knife body to cut the original grips so that they fit exactly right. Turn on visibility of **Grip 1** and **Grip 2.**
2. We're going to focus on **Grip 1** first.

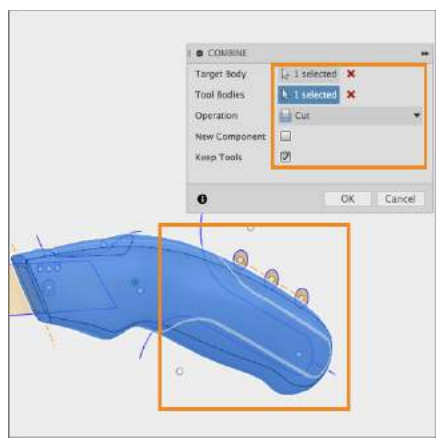


Step 16: – Cut Grip 1 with Combine tool

1. Go to the Modify drop-down menu and select **Combine.**

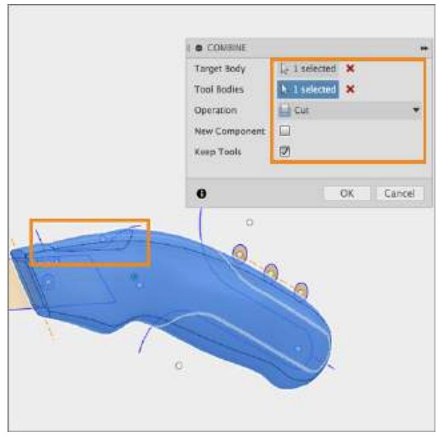
2. In the command dialog, set:
   Grip 1 as the **Target Body**,
   Knife Body as the **Tool Body,**
   Cut as the **Operation,**
   Check Keep Tools,

3. Click OK to finish the operation.



Step 17: – Repeat Combine Cut for Grip 2

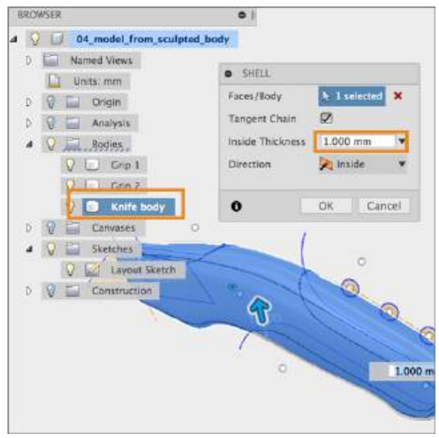
1. Repeat the previous step, but this time, cut **Grip 2** using **Knife Body**. Click OK to finish the operation.



Step 18: – Shell the knife body

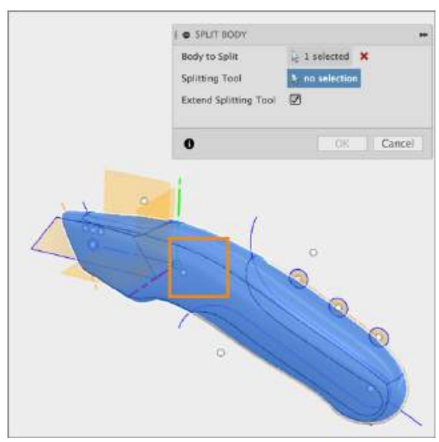
1. Now that we have our grips modeled, let's shell the inside of the knife body. Go to the Modify drop-down menu and select **Shell.**
2. Select **Knife Body** from the browser. Make sure that the shell thickness is **1 mm.** Click OK to finish.

Note: In the Shell command, selecting bodies from the browser will only shell the inside of those bodies. Selecting the face of a body will shell remove that face and shell the inside.



Step 19: – Split Knife Body into 2 pieces

1. Now that we have our knife body shelled, let's split it into 2 pieces. Go to the Modify drop-down menu select **Split Body.**
2. Select **Knife body** as the Body to Split.

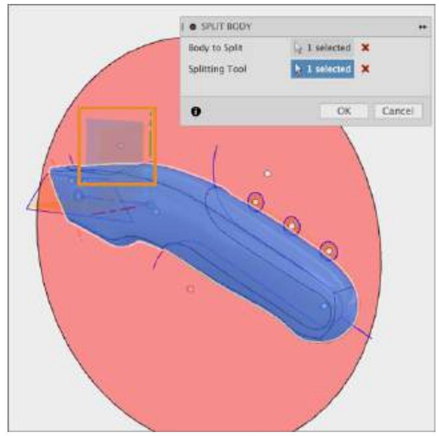


Step 20: – Split Knife Body into 2 pieces

1. Click on **Splitting Tool** to activate the selection

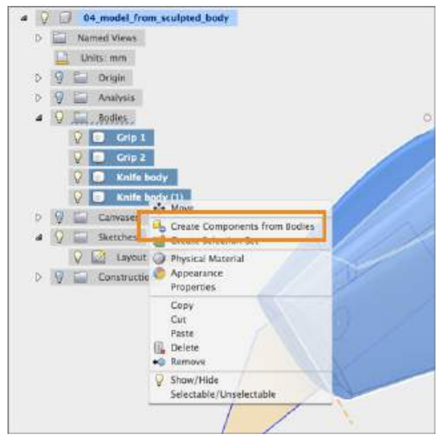
2. Select the plane that cuts down the middle of the **Knife body.** If you have trouble selecting the plane, zoom out until you can select it.

3. Click OK to finish the operation.



Step 21: – Convert Bodies to Components

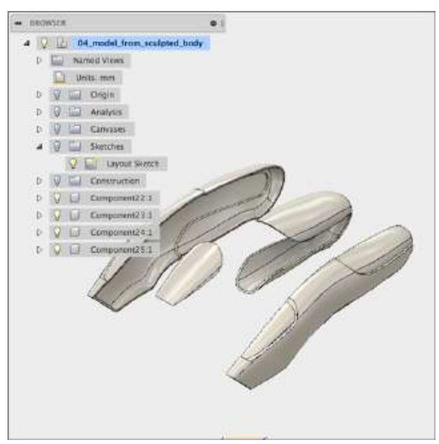
1. You should now see 2 knife bodies as well as the 2 grips in your Bodies folder. As the last step, let's convert these into components.
2. Select **all 4 bodies**, right-click and select **Create Components from Bodies.**



Step 22: – Lesson complete!

1. You have completed the lesson!

2. You can now drag the components apart and see all the work we did around the grips and the knife body.



Overview

Fusion 360 organizes and manages data using a centralized, cloud-based, collaboration platform. This enables designers and engineers to work more easily and efficiently together. Use this powerful and secure set of tools to dramatically improve the way you design, visualize, simulate, and share your work, on demand.

Learning Objectives

In this section you will learn how to:

- Create Fusion 360 designs and save versions.
- Create and manage Fusion 360 Group Projects.
- Add and remove users from Fusion 360 Group Projects.
- Find, view, and manage files within Fusion 360's collaborative web browser environment and the in-application dashboard.
- Access Fusion 360 data from a mobile device.
- Import and export files from Fusion 360.
- Publicly share data with external stakeholders.

If you like to watch the video to this tutorial, click here: **Launch Video**

Tips for this exercise:

- To complete the mobile section of the tutorial, install the Autodesk 360 App to your mobile device.
- Partner up with a friend who also has Fusion 360. There is an exercise where you have the option to grant access to your project.
- We recommend installing Google Chrome to best utilize the collaborative capabilities of Fusion 360 (the in-browser 3D viewer is not yet supported for IE, Firefox, and Safari).

Create and Manage Fusion 360 Group Projects: In this section you use data from an existing group project to create, setup, and manage a new group project in Fusion 360. You control who has access to this project. You create a new design, create multiple versions and determine where your new design will be stored. Once finished you then share a specific version publicly so that those not part of your project can access, inspect and review.

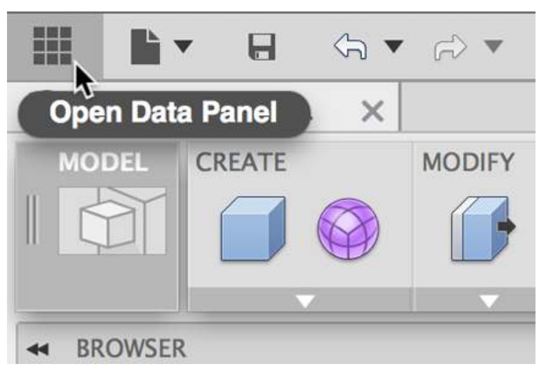
Fusion 360 allows you to securely access data from anywhere. You also control who else has access. The main way Fusion 360 controls access is by using "group projects." Group projects are the control mechanism for how you define who has access to specific sets of information. Group projects ensure that only the correct collaborators have access to your data.

Group projects are like folders in that they allow you to organize partitions of data, but they have the unique ability within Fusion 360 to explicitly define who may access data within it. Projects are unique locations where teams keep all related information in one shared place. People can share and access design data, discuss challenges and successes, and stay current with project activities. Each project has its own data, people, calendar, and wiki.

Common Configurations:

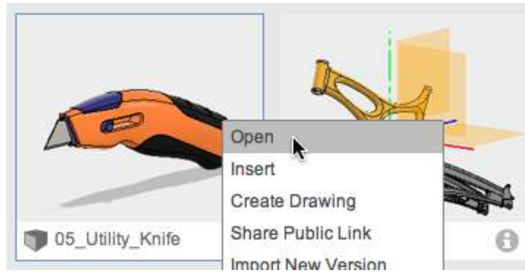
Some companies like to use group projects to separate between different jobs or work orders. Others prefer to use group projects to separate between different customers. Students commonly segment by specific assignments or by class. Whatever your configuration Fusion 360 offers flexibility to adjust and adapt as you evolve in your requirements.

Open Fusion 360 design file: In this section you will open the introductory design file.

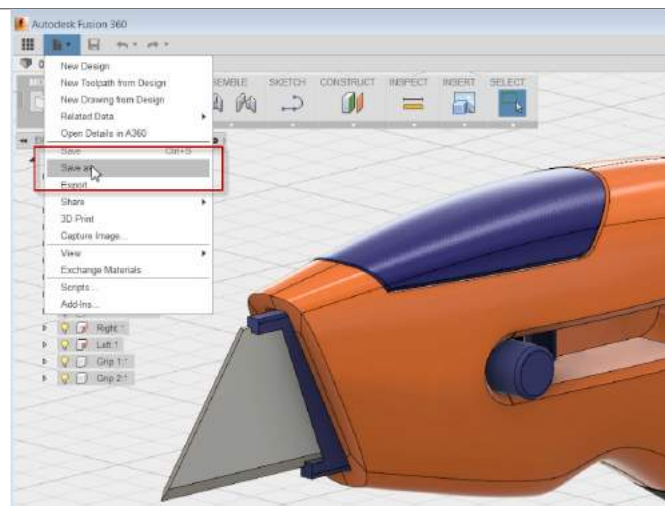
Step 1 – Open the Data Panel 1. Open the Data Panel by clicking on the icon located at the top left of the menu bar. 2. The Data Panel will slide open.	

<table>
<tr>
<td>
Step 2 – Open the design

In this module we will be using the 05_Utility_Knife.f3d file to complete the exercise. If you haven't set up a new project and uploaded the necessary designs, please follow the steps in the Introduction module.

1. At the top left of the Data Panel, select the project where you uploaded the 05_Utility_Knife.f3d file.
2. Navigate to this design and either double-click or right-click and select open.
3. When the design has opened in your modeling window, click on the icon to close the Data Panel.
</td>
<td>


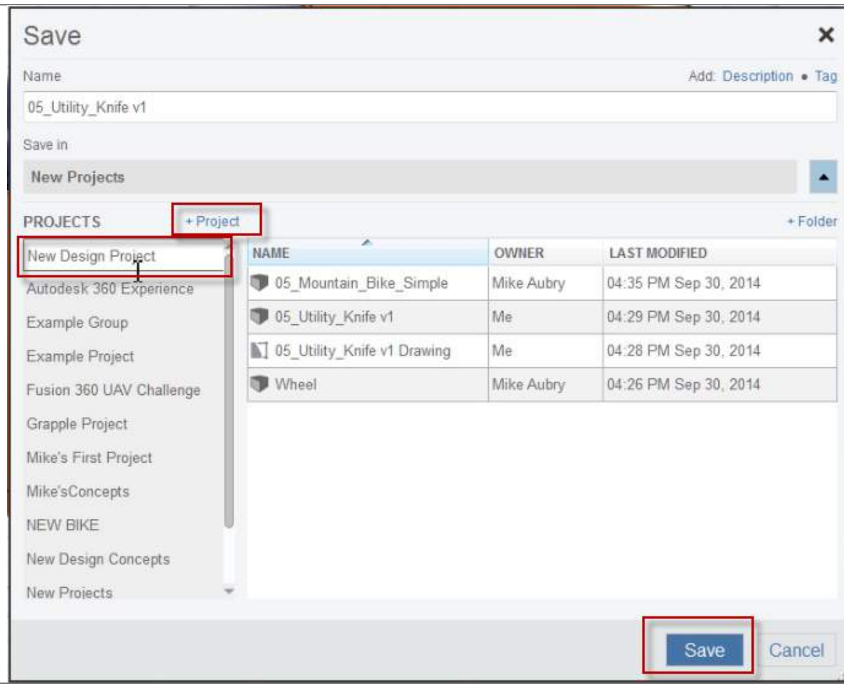
</td>
</tr>
<tr>
<td>
Step 3 – Start the Save As command

1. Select File > Save As.
</td>
<td>


</td>
</tr>
</table>

Step 4 – Create a new project

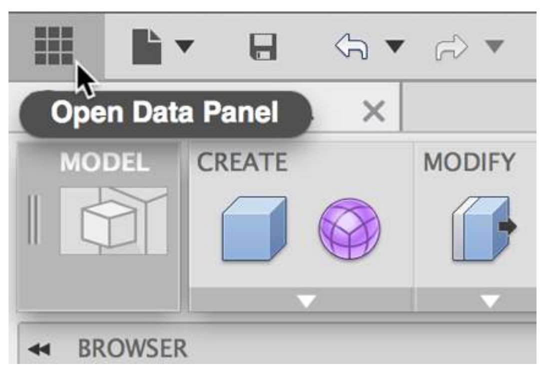
1. Click **+Project** to create a new project.
2. Name the project “**New Design Project**”.
3. Select **Save**.

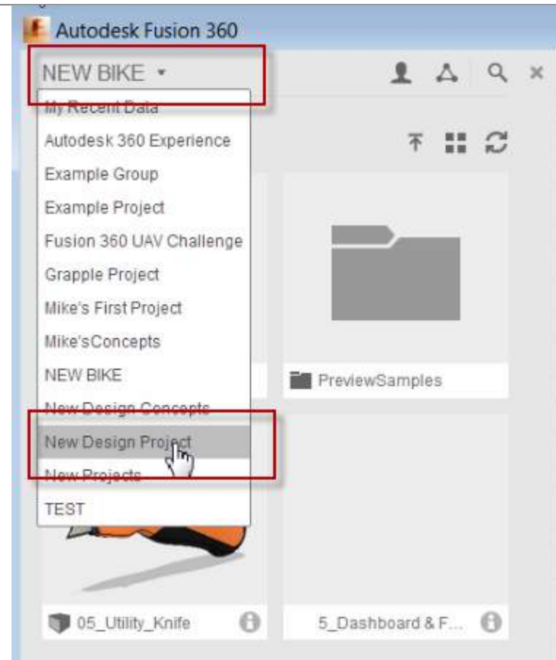
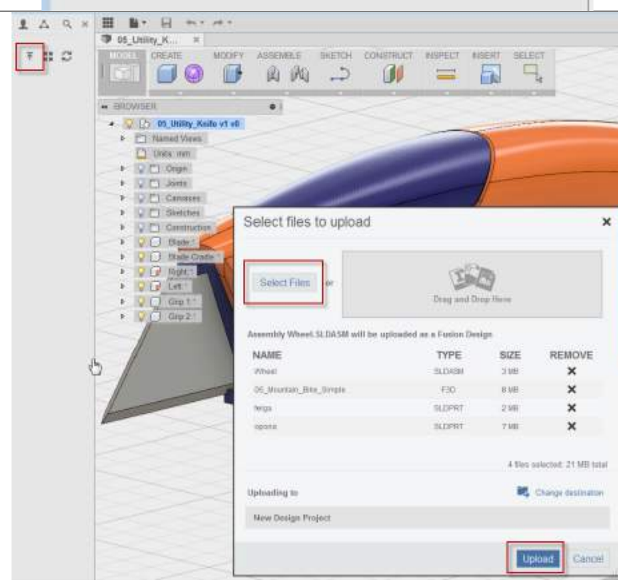


Upload data to a project: In this section, you upload data from your local drive to the project you just created.

Step 1 – Open the Data Panel

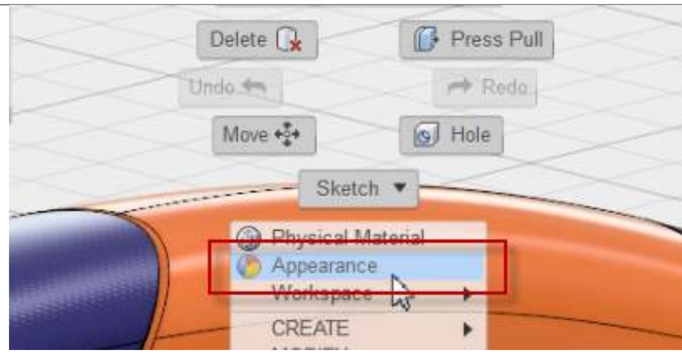
1. Open the Data Panel by clicking on the icon located at the top left of the menu bar.
2. The Data Panel will slide open.



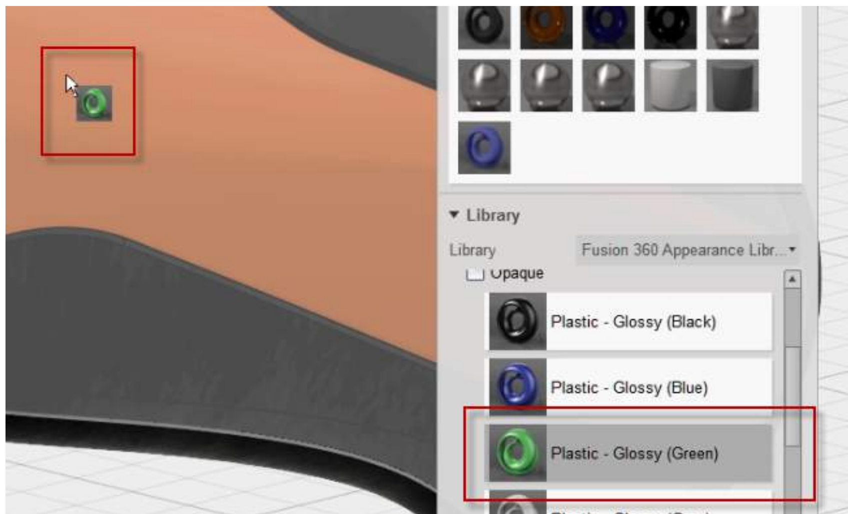
Step 2 – Set the active project 1. Click on the name of the active project. 2. Select **New Design Project** from the list. This is the project you created in the previous step.	
Step 3 – Upload data Fusion 360 translates data from many sources including Autodesk Inventor, Dassault SolidWorks, and PTC Creo. 1. Click **Upload**. 2. Click **Select Files** and navigate to the Chapter 5 dataset and upload: • 05_Mountain_Bike_Simple.f3d • Wheel.SLDASM • Flega.SLDPRT • Opona.SLDPRT 3. Click **Upload**. Uploading will continue in the background while you work.	

Create new versions: In this section, you make a change to a design then save it to create a new version.

<table>
<tr>
<td>
Step 1 – Start the Appearance command

1. Right-click and select **Appearance**.
</td>
<td>


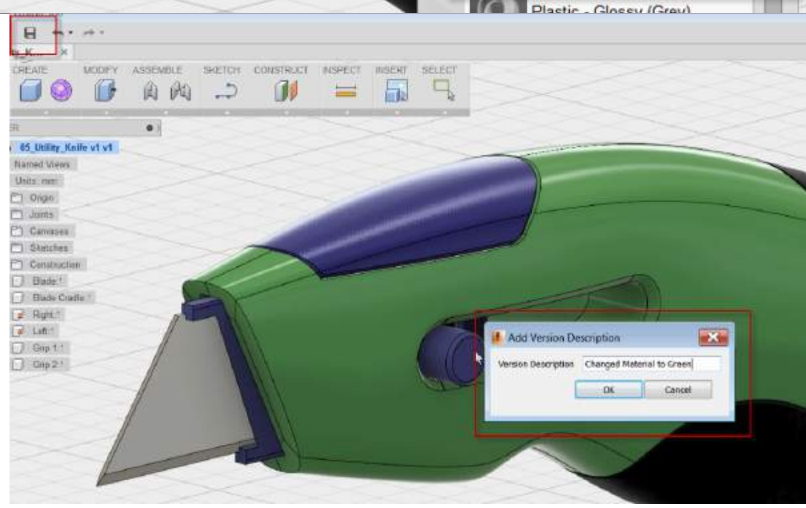
</td>
</tr>
<tr>
<td>
Step 2 – Change the appearance

1. Select **Plastic > Opaque > Plastic – Glossy (Green)**.
2. Drag and drop the appearance over the orange cover.
3. Click **Close** to dismiss the appearance command.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 3 – Create a version

Fusion 360 manages and stores each version you create. Make a change to an existing model and save it as a new version.

1. Click **Save**.
2. Enter the comment: “**Switched material from orange to green.**”
3. Click **OK**.

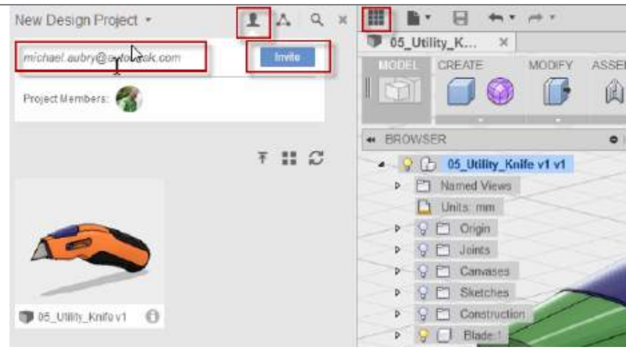
We will explore where this new version is stored shortly.
</td>
<td>


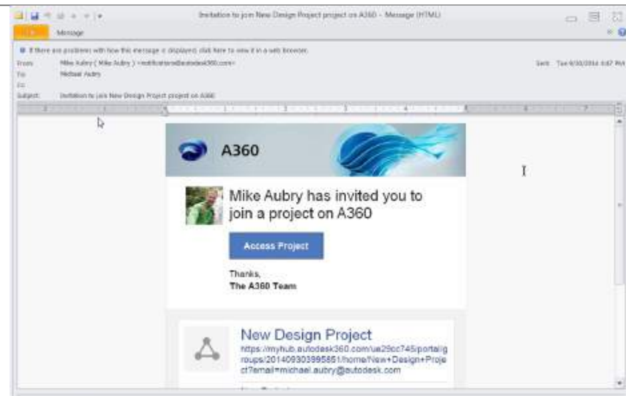
</td>
</tr>
</table>

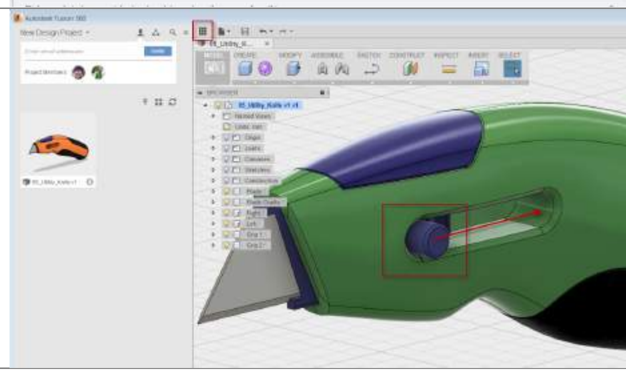
Add a user to your project: In this section you will open the introductory design file.

<table>
<tr><td>Step 1 – Add a project member

Fusion 360 allows you to control who has access to design and other data within projects. Invite another collaborator to access your project.

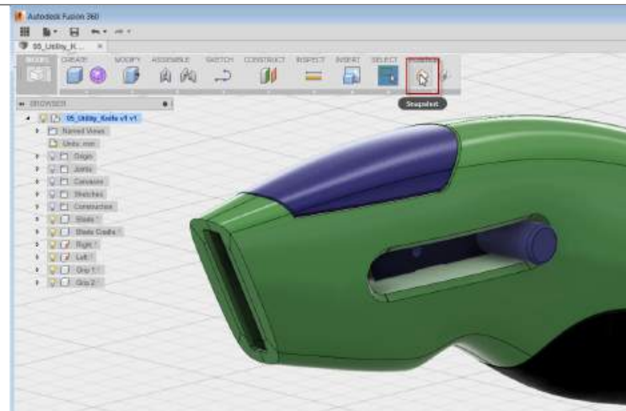
1. In the Data Panel, click the Project Members button (the head shaped icon).
2. Enter an email address of another Fusion user who you would like to have access. If you don’t have one feel free to use michael.aubry@autodesk.com.
3. Click Invite.</td><td>
</td></tr>
<tr><td>Step 2 – Confirmation email

1. Once added, the user you will receive an email notifying him or her of their addition.</td><td>
</td></tr>
<tr><td>Step 3 – Make a design change

1. Drag the slider to the right to retract the blade.</td><td></td></tr>
</table>

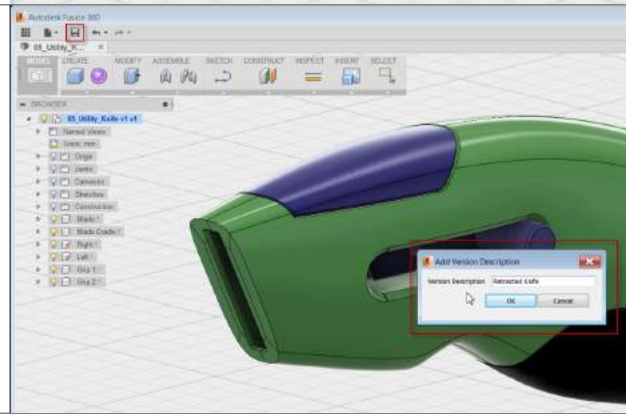
Step 4 – Create a snapshot

1. Click **Position > Snapshot** to create a snapshot to maintain the position.



Step 5 – Save the design

1. Click **Save** to save a new version.
2. Add a comment: “**Retracted knife**”.
3. Click **OK**.
4. **Close** the utility knife design.

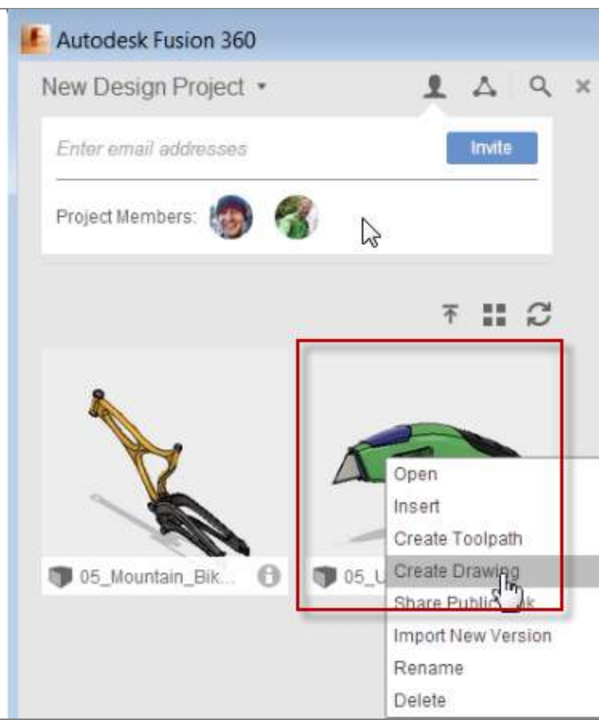


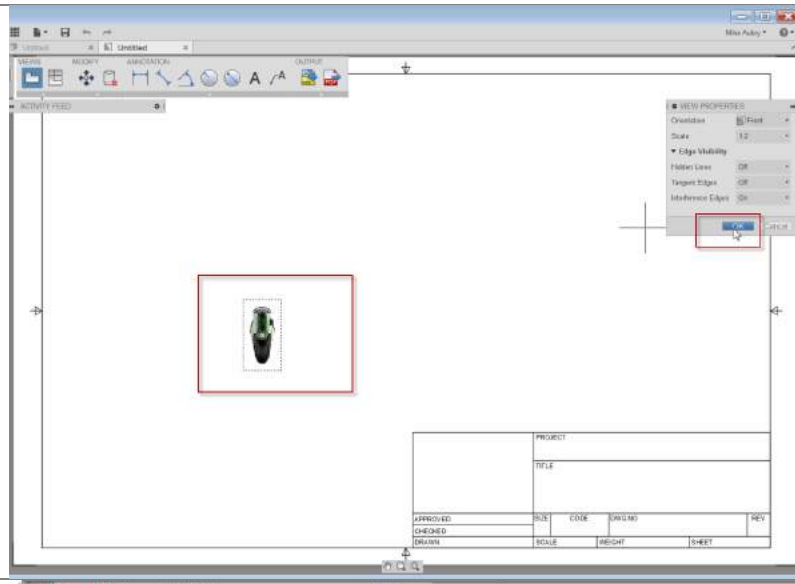
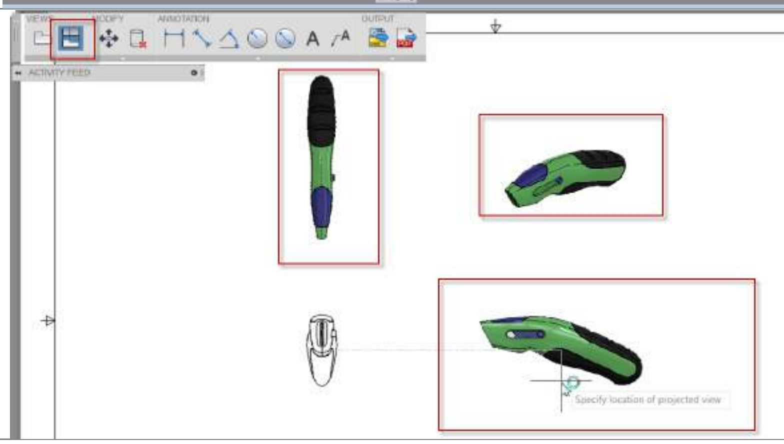
Create a referenced document: In this section, you create a drawing from the utility knife. Fusion maintains the relationship between the design and the drawing.

Step 1 – Create a drawing

Fusion 360 maintains associative relationships between drawing and design documents. What this means is if you change the design the downstream drawing will be updated as well.

1. In the Data Panel, right-click on the **05_Utility_Knife** design.
2. Click **Create Drawing.**

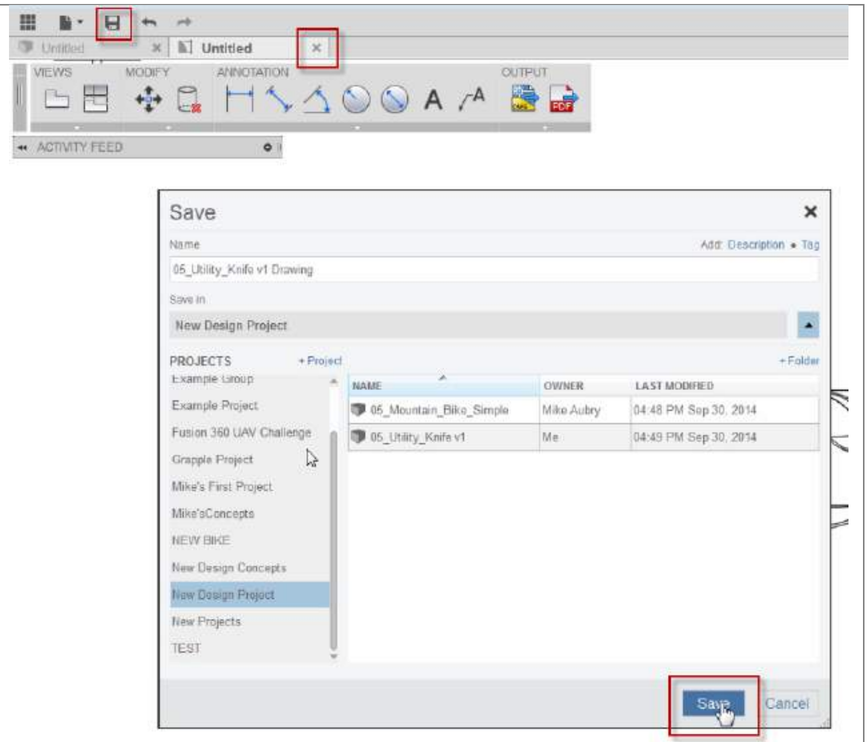


Step 2 – Place the base view 1. Left-click on the drawing sheet to place the base view. 2. Click **OK** to create the view.	
Step 3 – Add projected views 1. Click **View > Projected View**. 2. Click on the base view. 3. Click in the three areas shown to create top, right, and isometric views. 4. Right-click and click OK to create the views.	

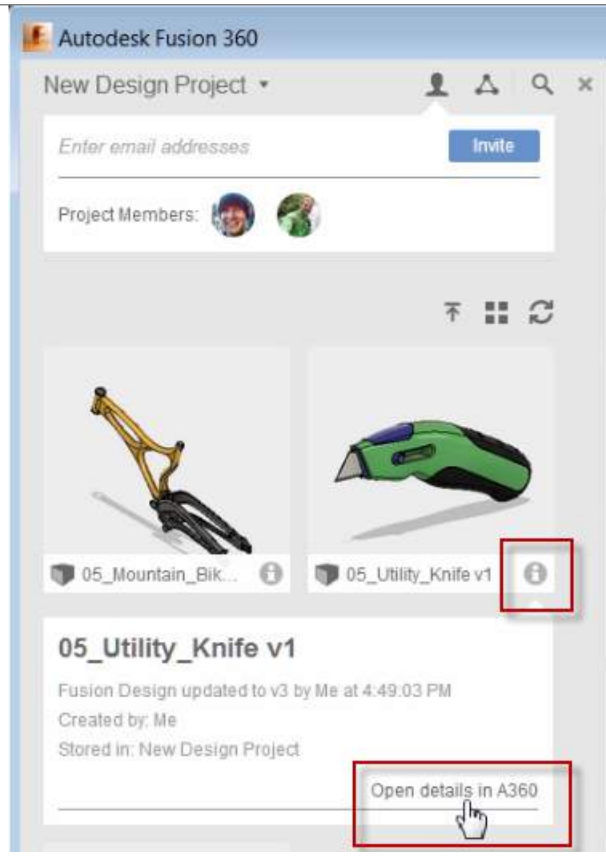
Step 4 – Save and close

1. Click the **Save** icon.
2. Click **Save** to accept the default name and project.
3. Close the drawing.

You will see the association in a later step.



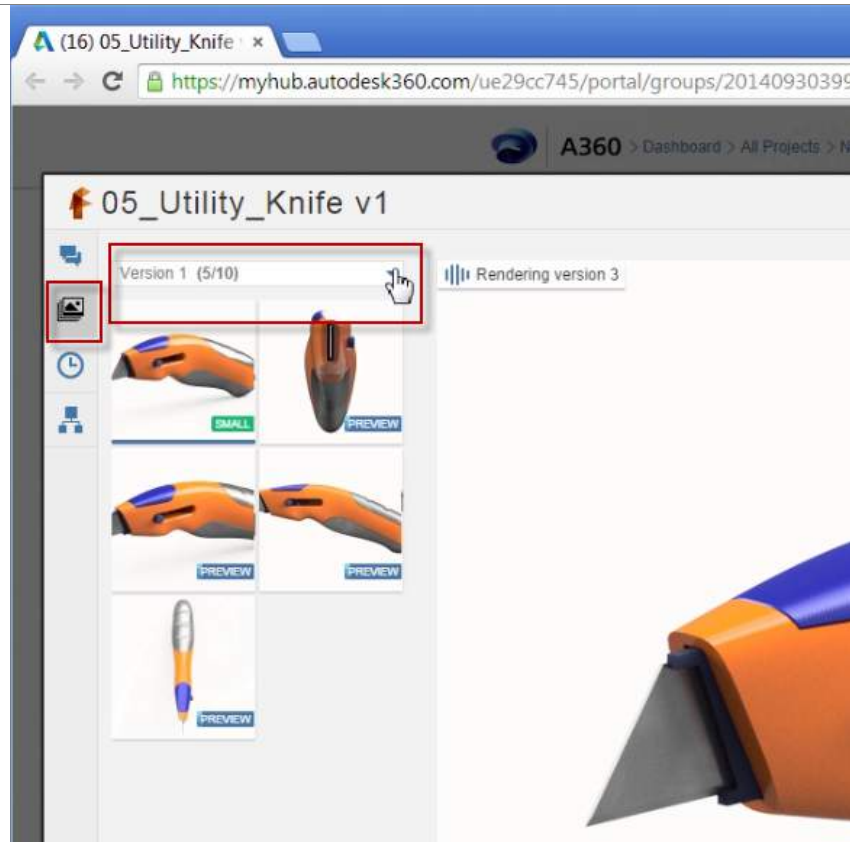
Access data in a web browser: Use Autodesk A360 to access your data in a web browser.

Step 1 – Access Autodesk A360 Fusion 360 allows you to manage data centrally through your web browser. Open your design in a web browser. 1. In the Data Panel, click the "**i**" icon on the 05_Utility_Knife to display details about the design. 2. Click **Open Details in A360**. Suggestion: Consider using Google Chrome for this exercise as your default browser. Some features may not yet be supported for Safari, Firefox and Internet Explorer.	

Display versions and use the viewer: In this section, you use the viewer to display the different versions of the utility knife. You also use the viewer to investigate the design.

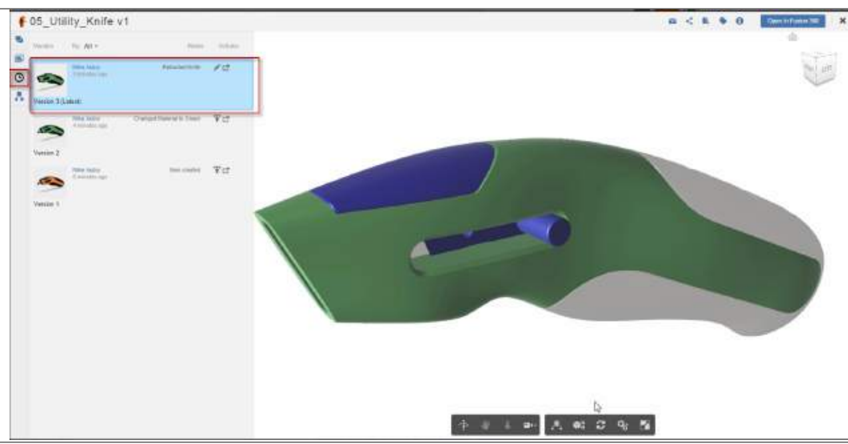
<table>
<tr>
<td>
Step 1 –View renderings
Each version that is saved is archived and stored by Fusion 360. View all three versions that were created for the Utility Knife.

Click the Rendering icon.
Select each version of the utility knife to display the renderings of previous versions. The images were created automatically when you saved a new version in previous steps.

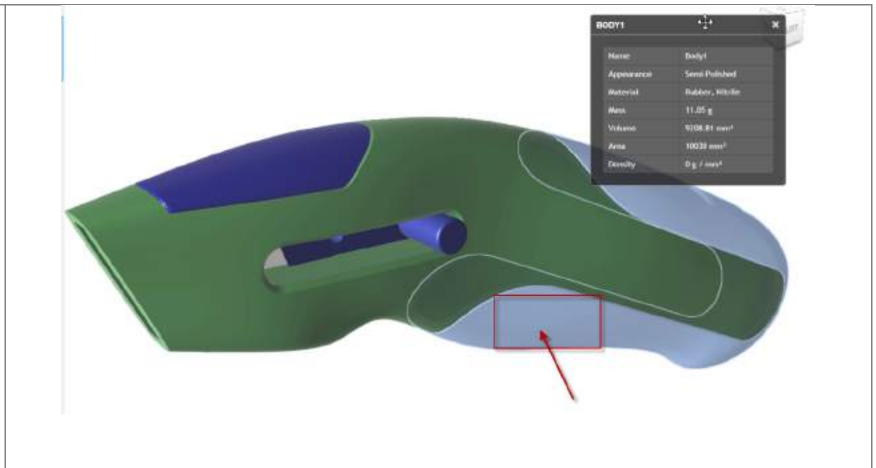
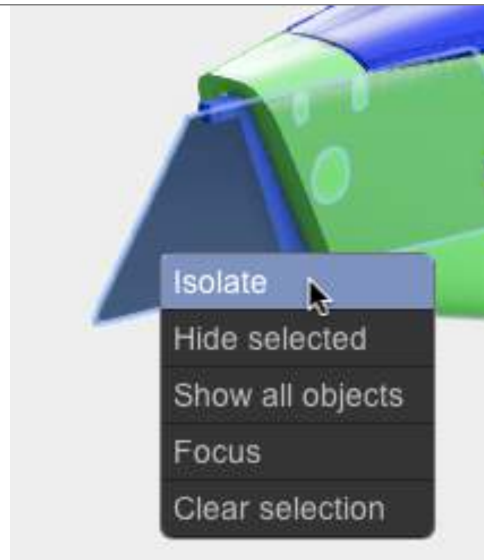
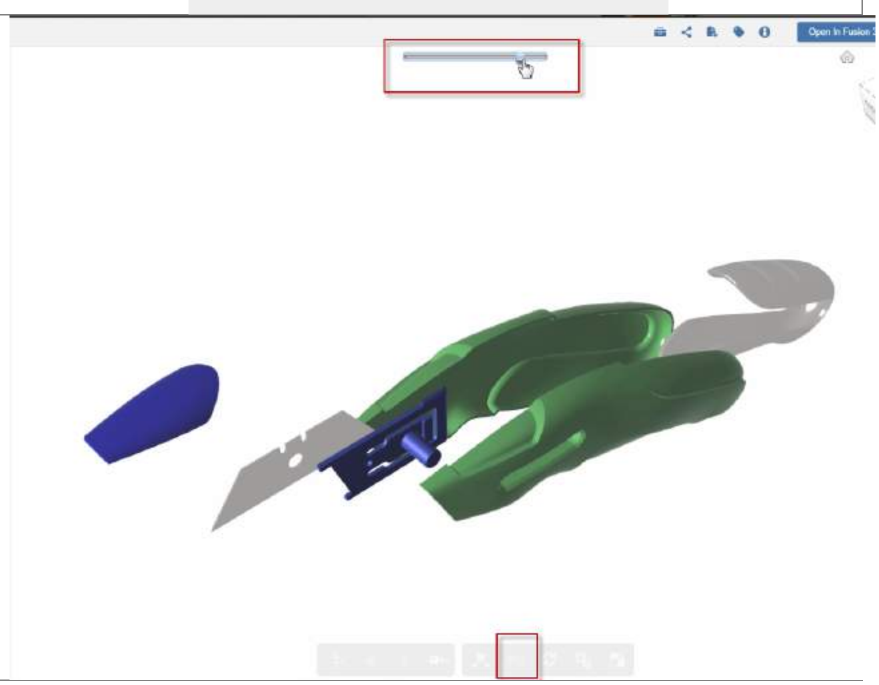
Cloud based rendering is discussed further in the Rendering chapter.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – View versions

Click the Versions icon (the watch shaped icon).
Click the different versions to view 3D previews. Notice the comments that were entered in previous steps are also available to view.

</td>
<td>


</td>
</tr>
</table>

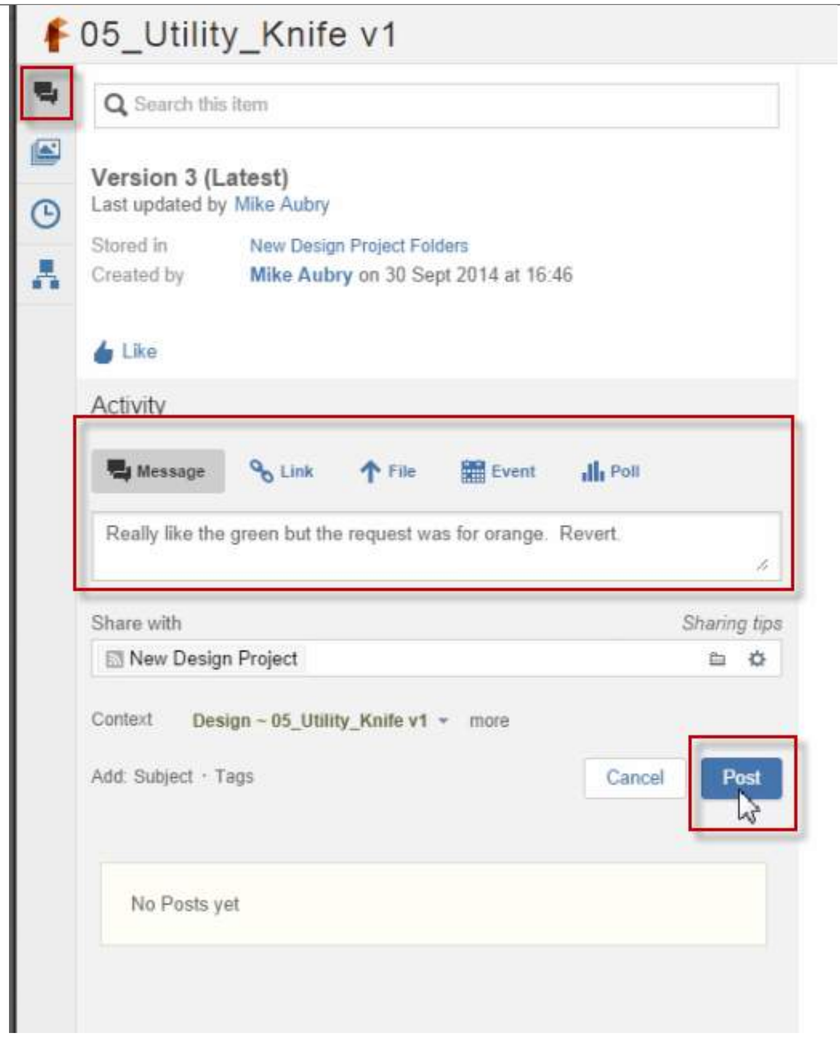
Step 3 – View the 3D design 1. Hold the left mouse button to rotate the 3D model in the viewer. 2. Left-click on an item to select and highlight specific bodies. Notice that information like the material and weight of the selected item is displayed.	
Step 4 – Isolate an item 1. Select an item. 2. Right-click and select **Isolate** to display only display the selected item. 3. Right-click and select **Show all objects** to display the entire design.	
Step 5 – Explode the design 1. Click the **Explode** button at the bottom of the screen. 2. Use the slider to expand and contract the assembly.	

Document design decisions: In this section, you add a message to the design in A360. Messages can be seen by anyone with access to the project.

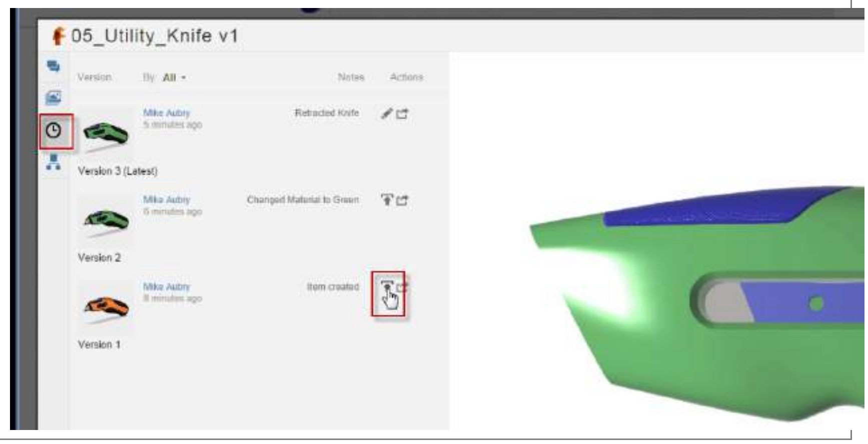
Step 1 – Document activity

Fusion 360 allows you to collaborate and articulate design decisions within your design. Create a message to describe to your team a design change you will make.

1. Click the **Preview** icon.
2. Click in the message field and enter: "Really like the green but the request was for orange. Revert."
3. Click **Post** to post the message.



Promote a previous version: In this section, you promote a previous version to be the current version. This let's you "rollback" your design.

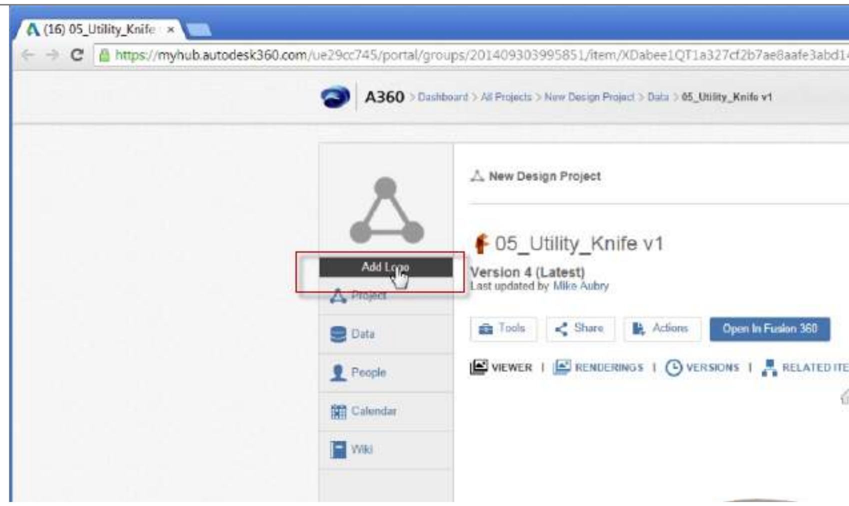
Step 1 – Promote previous versions Fusion 360 stores and allows you to promote previous versions. Any archived version can be promoted back to the current version. 1. Click the **Versions** icon (the watch shaped icon). 2. Click the **Promote** icon.	

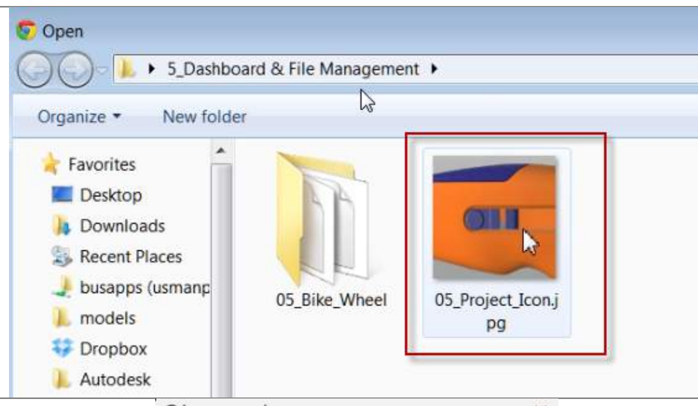
View associated data: In this section, you view a list of data associated to the utility knife design then close the view of the design.

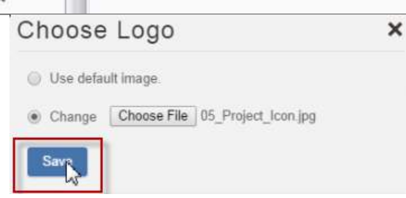
Step 1 – View associated documents Fusion 360 manages associativity between designs and drawings. View the drawing that you created in a previous step. 1. Select the **Related Items** icon. 2. The drawing is listed as an associated item.	
Step 2 – Close the viewer 1. Click the X to close the viewer.	

Customize your project: You can add a custom logo to your project. This makes it easier to differentiate between projects.

<table>
<tr>
<td>Step 12 –Add a logo to your project

1. Select Add Logo in the A360 web browser.
2. Select Choose File.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Select the image

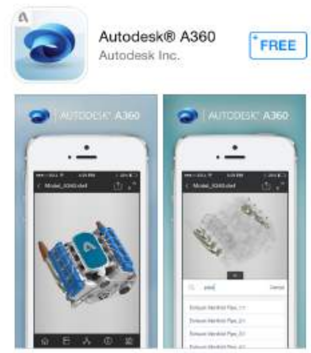
1. Navigate to the png image provided in the chapter 5 dataset or use any jpg or png under 200x200 pixels.
2. Select the image then click Choose.</td>
<td>
</td>
</tr>
<tr>
<td>Step 3 – Save the change

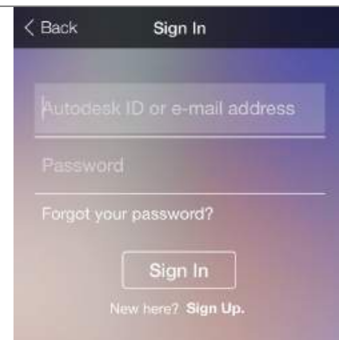
1. Click Save.</td>
<td>
</td>
</tr>
</table>

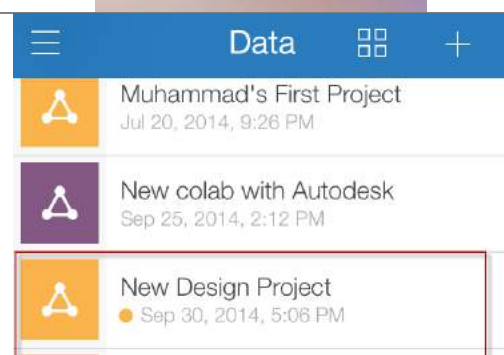
Access data from a mobile device: In this section, you access the project from a mobile device.

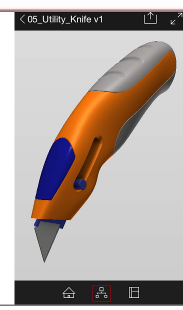
<table>
<tr>
<td>Step 1 – Access data from a mobile device

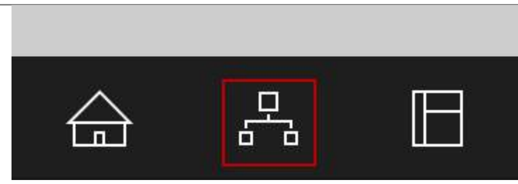
Fusion 360 is accessible through your mobile devices.

1. To access data from your apple mobile device install the free Autodesk A360 App via the Mac App Store or Google Play Store.</td>
<td>
</td>
</tr>
<tr>
<td>Step 2 – Sign in

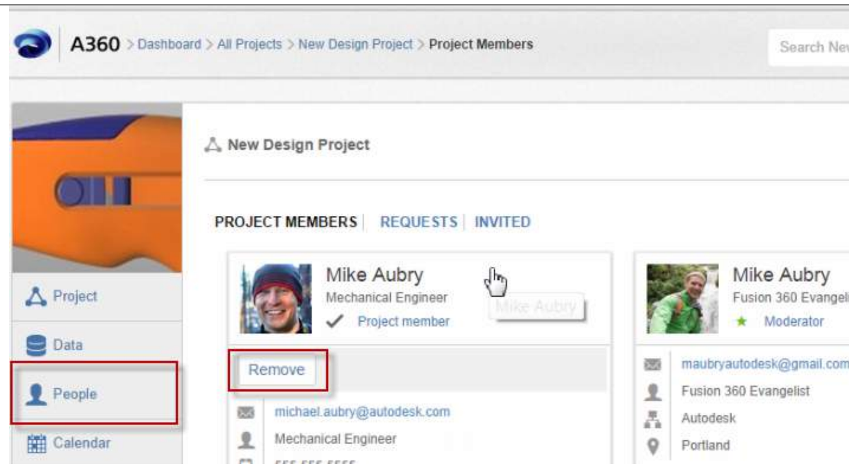
1. Sign in to the App using your Autodesk ID (it is the same account you use for Fusion 360).</td>
<td>
</td>
</tr>
<tr>
<td>Step 3 – Select the project

1. Scroll to the New Design Project and select it.</td>
<td>
</td>
</tr>
<tr>
<td>Step 4 – Select the design

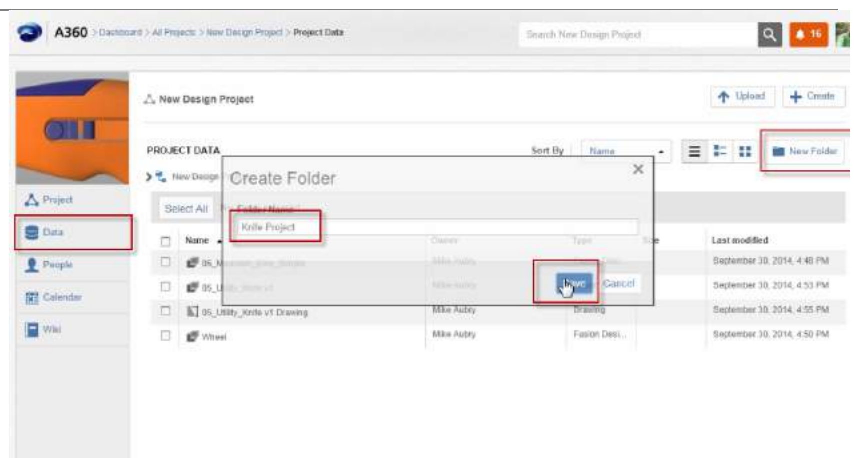
1. Open the Utility Knife.</td>
<td>
</td>
</tr>
</table>

Step 5 – Select the action 1. Select the **Isolate** icon.	
Step 6 – Select the object 1. Choose **Blade Cradle** to isolate the blade.	
Step 7 – View the object 1. Rotate the model by placing a finger and moving on the screen. 2. Double tap on the model to change the rotation pivot point, which will appear as a green sphere. 3. Exit the app and return to your laptop to complete the remaining exercises.	

Revoke access to the project: Fusion 360 allows the moderator of a project to add or remove access to that project. Revoke the access of a project member.

Step 1 – Revoke access from a project 1. Go to the **New Design Project** in the A360 web interface. 2. Select **People**. This shows all the members of the project. 3. Select **Remove**. This removes the access of the user you added earlier in an earlier step. The user no longer has access to the project from any device.	

Manage data: Familiarize yourself with common data management operations like move, rename and delete by moving the Utility Knife into a new folder.

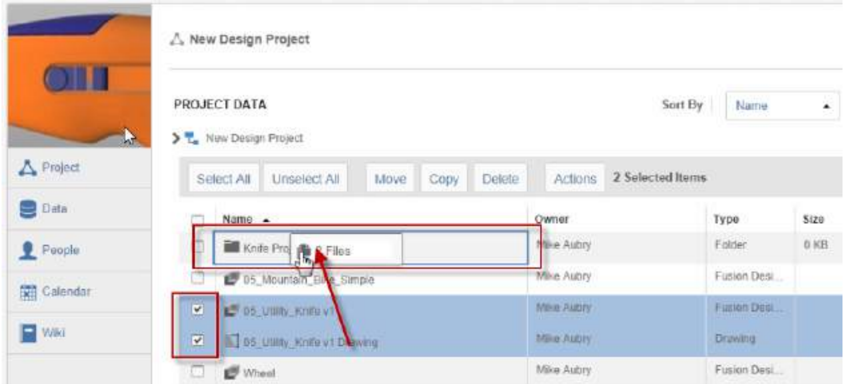
Step 1 – Create a new folder 1. Select **Data** to explore the folder hierarchy of the project. 2. Click **New Folder**. 3. Enter "**Knife Project**" then click **Save**.	

<table>
<tr>
<td>

Step 2 – Move data

1. Check the boxes next to the design and drawing.
2. Drag and drop the items into the folder to move them.
3. Open the **Knife Project** folder.

</td>
<td>

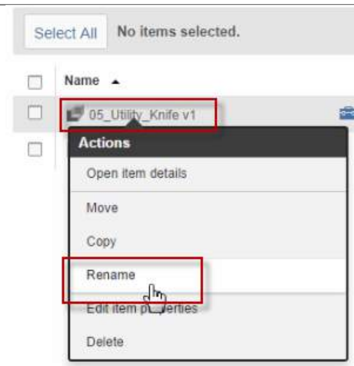


</td>
</tr>
<tr>
<td>

Step 3 – Rename a design

1. Right-click on the **05_Utility_Knife** design and select **Rename**.

</td>
<td>

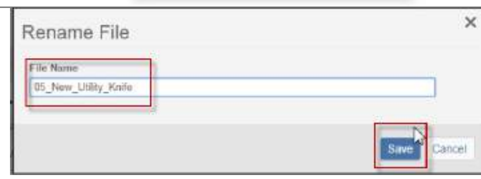


</td>
</tr>
<tr>
<td>

Step 4 – Enter new name

1. Rename to: “**05_New_Utility_Knife**”.
2. Select **Save**.

</td>
<td>

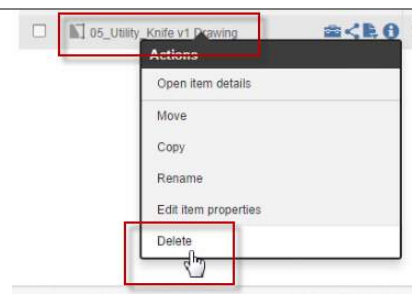


</td>
</tr>
<tr>
<td>

Step 5 – Delete an item

1. Right-click on the **05_Utility_Knife** drawing and select **Delete**.

</td>
<td>

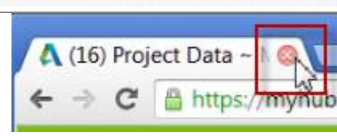


</td>
</tr>
<tr>
<td>

Step 6 – Close your browser

1. Close the web browser.

</td>
<td>

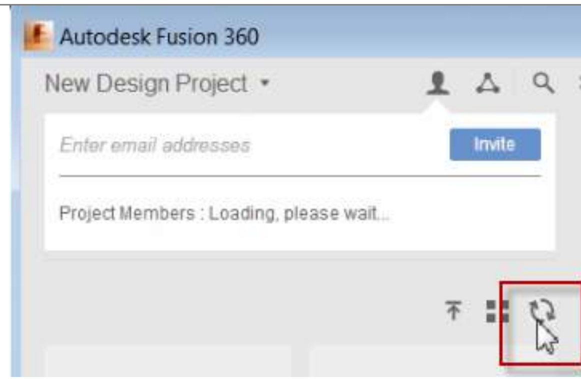


</td>
</tr>
</table>

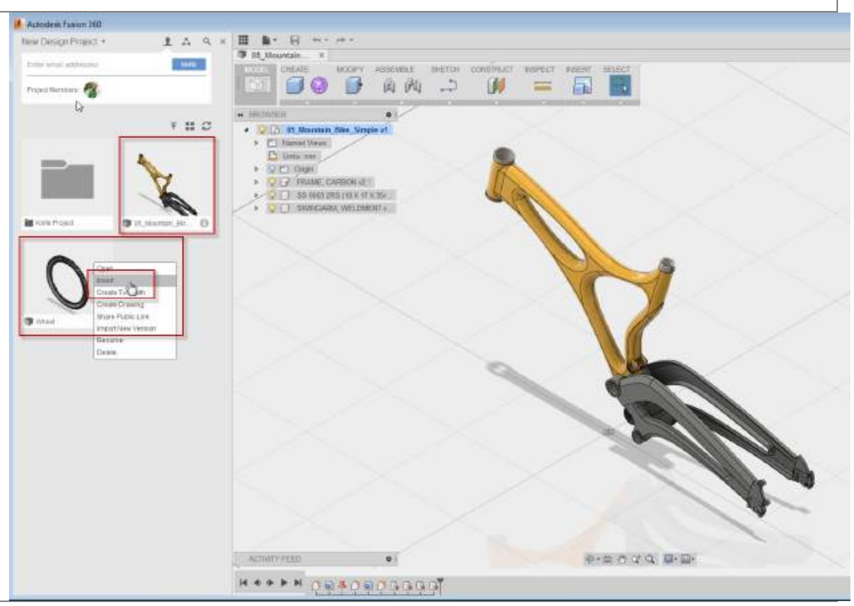
Insert designs into other designs: In this section, you insert a design into an assembly design.

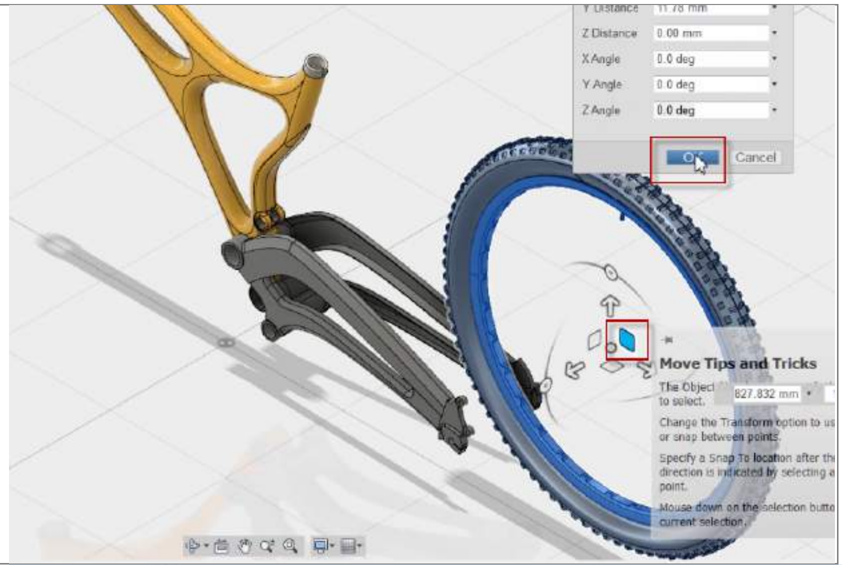
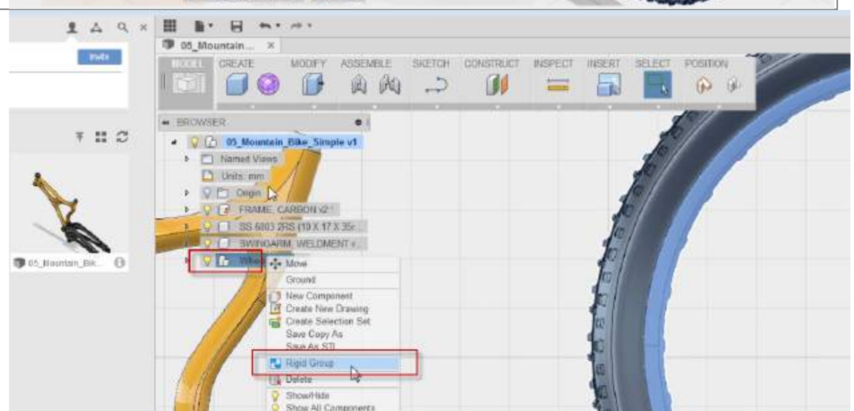
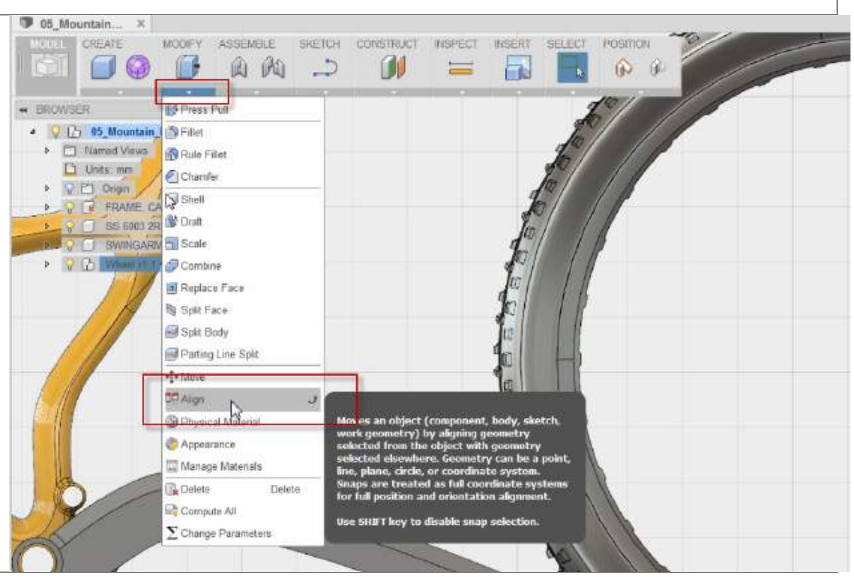
<table>
<tr>
<td>
Step 1 – Refresh data

1. Return to Fusion 360.
2. Select Refresh in the Data Panel. This will get the most current data and reflect the changes made in the web browser in the previous step.

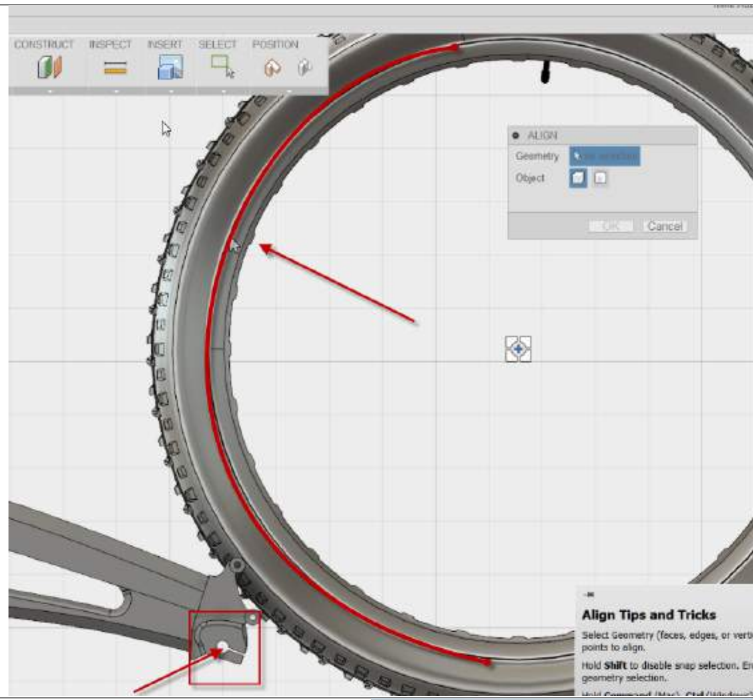
Notice that while you were completing the previous steps that Fusion 360 has fully uploaded the bike dataset and SolidWorks wheel dataset. The wheel is also combined into one single Fusion design.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Insert a design into an assembly

1. Double-click the Mountain Bike design to open it.
2. Right-click on the Wheel design and select Insert to insert the Wheel into the Bike Design.
</td>
<td>

</td>
</tr>
</table>

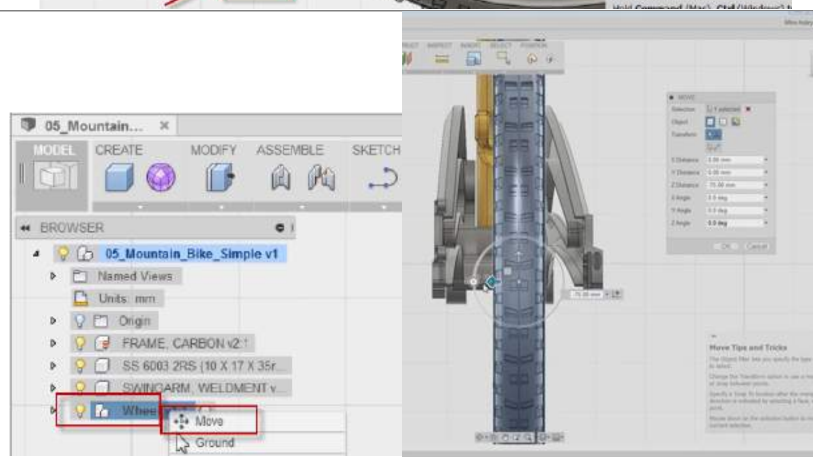
Step 3 – Place the wheel 1. Use the manipulator to position the wheel in the bike assembly. 2. Click **OK** when done.	
Step 4 – Lock the wheel component 1. Right-click **Wheel** in the browser and select **Rigid Group**. Rigid groups move as one object. This will make it easier to align to the wheel with the bike in the next step.	
Step 5 – Start the Align command 1. Click **Modify > Align**.	

<table>
<tr>
<td>
Step 6 – Select areas to align

1. Select the circumference of the wheel and select the circumference of the bike frame's dropout. The wheel will axially align with the bike dropout.
</td>
<td>


</td>
</tr>
<tr>
<td>
Step 7 – Center the wheel

1. Right-click Wheel in the browser and select **Move**.
2. Drag the manipulator to approximately center the wheel.
3. Click **OK**.

The Assemblies tutorial will show how to be more precise using joints.
</td>
<td>


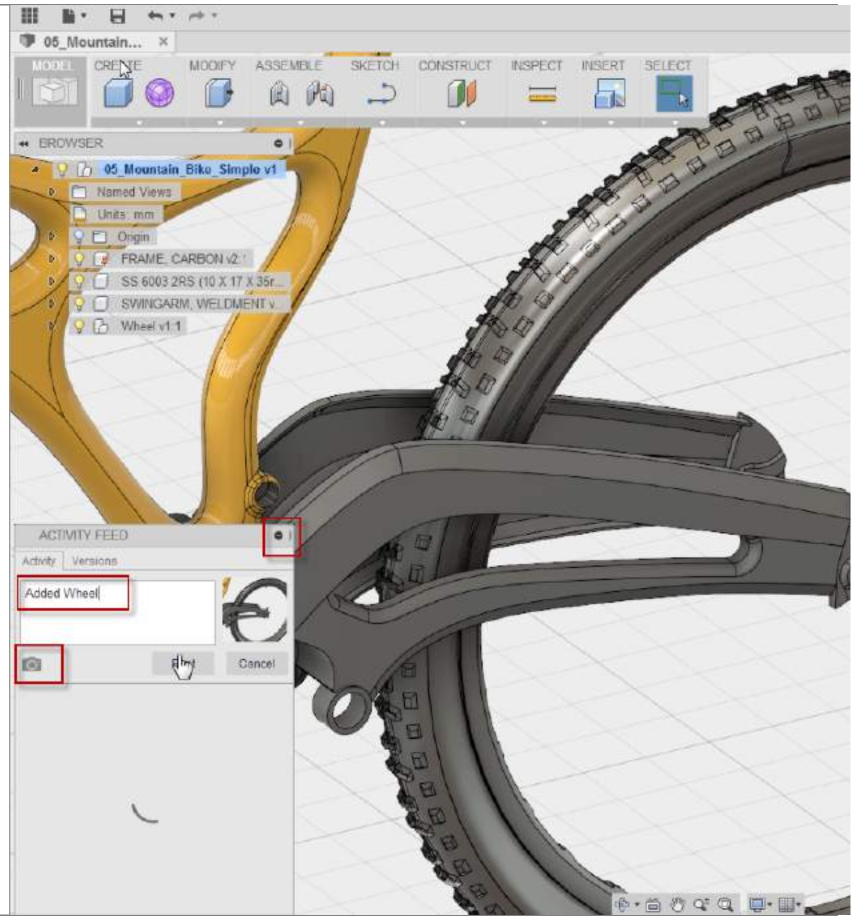
</td>
</tr>
</table>

Step 8 – Activity feed

1. Click the "**+**" to expand the Activity Feed.

The activity feed is where you see recent activity in the design.

2. Click the **Camera** icon to create a snapshot of the screen to show progress you have made on the design.
3. Enter "**Added Wheel**" in the comment field.
4. Click **Post**.

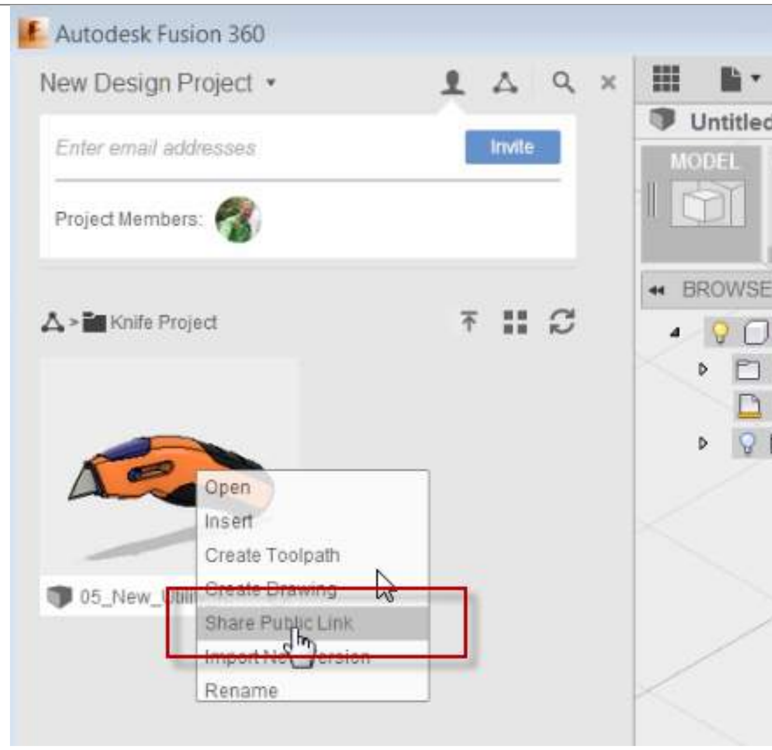


Share designs: In this section, you create a public link to share your design. Then paste the link into a browser to see what other stake holders see when using the link. Finally, you download the file from the public link.

<table>
<tr>
<td>
Step 1 – Share public link

Regardless of whether a user has an Autodesk ID, you are still able to share and make designs available for download. Share a design using a public web link.

Open the Data Panel and browse to the 05_Utility_Knife.
Right-click on the Utility Knife and choose Share Public Link.

</td>
<td>


</td>
</tr>
<tr>
<td>
Step 2 – Configure the link

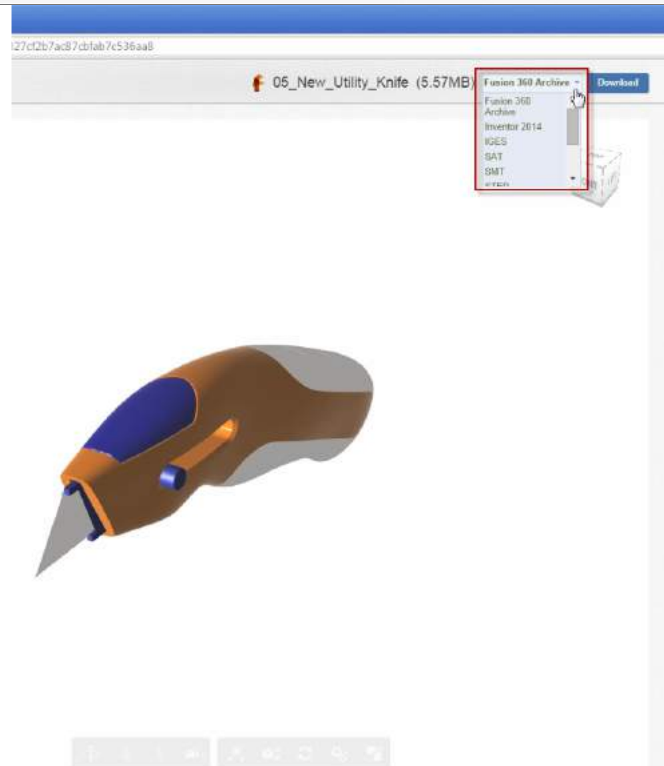
Check the following:

Share the latest version with anyone using this public link
Allow the item to be downloaded
Require a password

Enter “Fusion!” as your password.
Click Copy to copy the link to your clipboard.
Close the Share Public Link dialog box.

</td>
<td>


</td>
</tr>
</table>

Step 3 – Use the link 1. Open your web browser. 2. Paste in the copied URL. 3. Enter the password, "**Fusion!**".	
Step 4 – View and download 1. A 3D preview is displayed in the browser. 2. Select a CAD format to download. 3. Click **Download**.	

Congratulations! You completed the Manage and Collaborate module.

Intro to 2D Strategies | CAM for Fusion 360

CNC milling toolpaths are broadly classified as either 2D, 3D, 4-axis, and 5-axis, depending on the number of axes involved and how they move. The term, 2D, is a bit of a misnomer because all modern CNC machines control at least three axis and all three axes move at one time or another for every 2D machining operation. A more accurate term, 2-1/2D, is commonly used in CNC manufacturing. For more information, please refer to the Autodesk CNC Handbook.

2D vs. 3D Defined

2D (Prismatic) Parts

2-1/2D milling toolpaths machine only in the XY plane. The Z- axis is used only to position the tool at depth. The move to the cutting plane is a straight down feed, rapid, ramp or helical feed move.

The term, Prismatic, is a term commonly used in engineering to describe 2-1/2D parts. There are, however, prismatic parts that require 4th or 5-Axis machining, so the term is used in machining only to describe parts where all machined faces lie normal to the machine tool spindle. The XY axes are normal to the machine spindle and Z is used only to position the tool to depth (either in a feed or rapid motion).

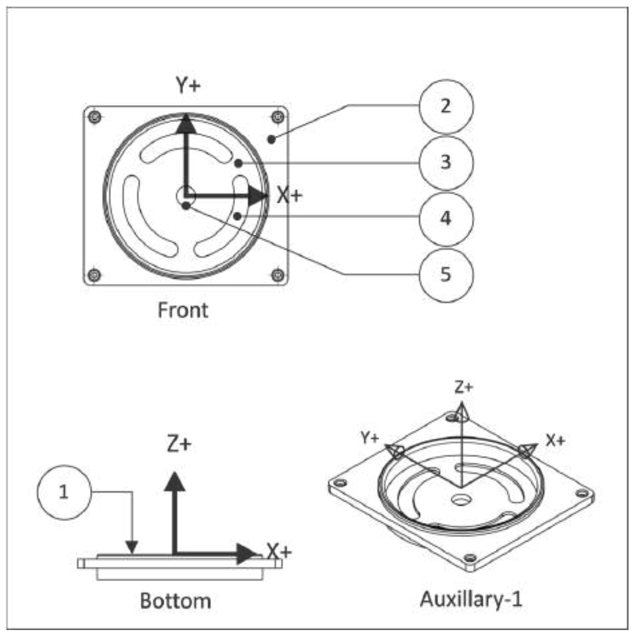


Figure 1: Prismatic Part (Orientation in CAD)

Figure 1 shows a prismatic part. All machined features lie parallel to the XY plane. Each Z-level can be machined by positioning the tool at a fixed Z-level and then moving the XY axes to remove material. Every feature can be reached with the tool approaching either from the Front or Bottom views. There are several cutting planes in this example, including the model top (1), top of the face where the holes start (2), the bottom of the pocket (3) where the slots begin, the bottom of the slots (4), and the bottom of the hole through the center (5).

Learning Objectives

Upon successful completion of this lesson, you will be able to:

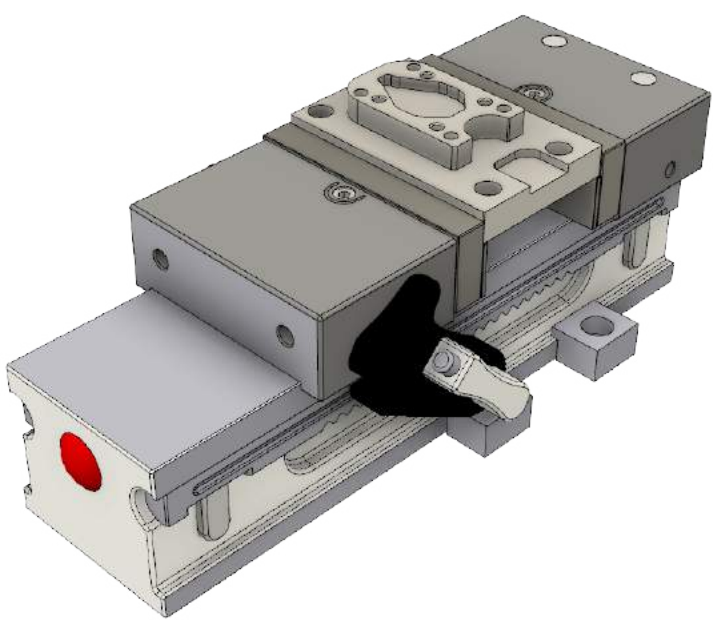
- Explain the difference between 2-1/2D and 3D machined parts.
- Explain the difference between common CAD and CAM graphics views
- Identify 2D machining features based on part geometry and your knowledge of tools and 2D toolpaths.
- Identify commonly used machining parameters for 2D tool path operations.
- Apply a Job Setup to a 2D Milled Part
- Apply a multitude of 2D Operations to a Milled Part
    - Facing Toolpaths
    - 2D Adaptive Toolpaths
    - 2D Contour Toolpaths
    - Chamfer Milling Toolpaths
    - Bore Toolpaths
- Produce Setup Sheets
- Simulate Toolpaths and Stock Material Removal
- Produce NC Code via Post Processing

Datasets Required

In Samples section of your Data Panel, browse to:

Fusion 101 Training > 09 – CAM > **09_2D_Strategies**

Open the design and follow the step-by-step guide below to get started with the lesson.



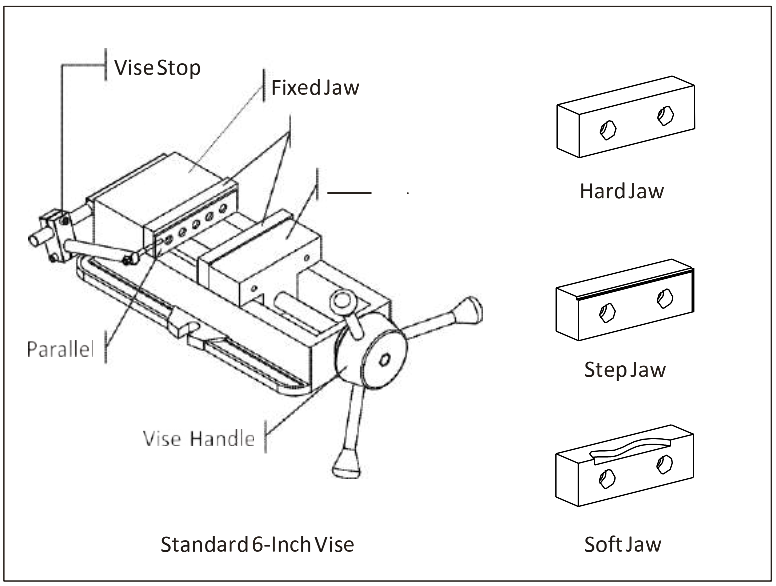
Lesson 1: Workholding & Job Setup

Fixture Component Terminology

Vise and Accessories

The CNC vise is precision engineered and manufactured with components ground flat and perpendicular to within .0002 inches. The most common is referred to as a six-inch (6”) vise, because the width of the jaws is six inches.

Once the vise is bolted to the table and aligned, parts are loaded into the vise and clamped by closing the jaws. The vise can exert tremendous force, so care is taken not to over-tighten the vise and deform fragile parts. Vise pressure must be appropriate to the part being held and expected cutting forces.

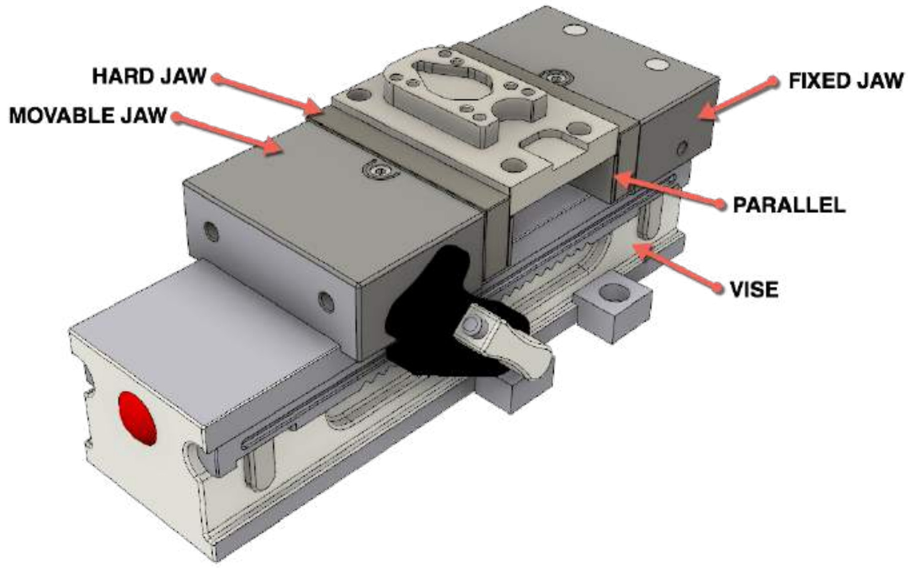


The **Fixed Jaw** remains stationary. The **Moving Jaw** opens when the **Vise Handle** is turned. It is a good practice to remove the vise handle after the jaws are closed and before running the program. This is done by simply sliding the handle off.

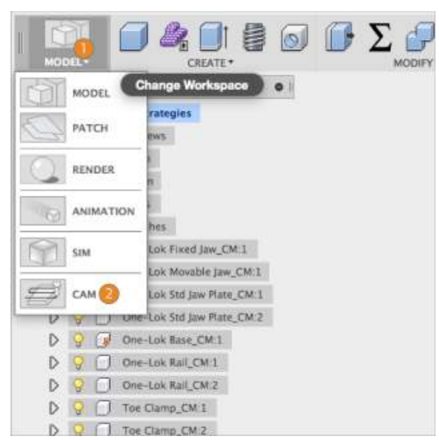
A **Vise Stop** is a device that allows the parts to be loaded into the vise precisely. This image shows a style of vise stop that is particularly useful because it is adjustable up-down and left-right.

Hard Jaws are made of hardened steel and precision ground on all sides. They are usually used along with parallels.

Parallels are thin steel plates, available in various widths, used to set the grip length of the vise jaws.



Step-by-step Guides:

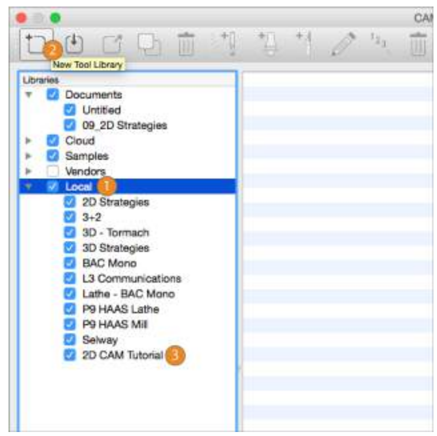
Step 1: Activate the CAM Workspace.	

Step 2: – Start the **TOOL LIBRARY** command

1. Click **TOOL LIBRARY**

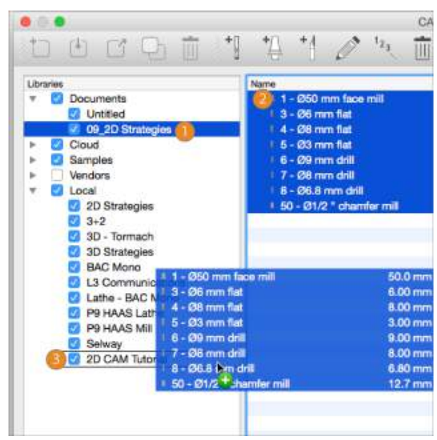
Step 3: – Create a **NEW TOOL LIBRARY**

1. Click on **Local**
2. Select the **NEW TOOL LIBRARY** Icon
3. Double Click on the **NEW TOOL LIBRARY** name and rename to **2D CAM Tutorial**.



Step 4: – Copy and Paste **TOOLS** into **NEW LIBRARY**

1. Click on **Documents** and select the Library **09_2D_Strategies**
2. Select **ALL** tools in the library, and drag and drop into your new library created.

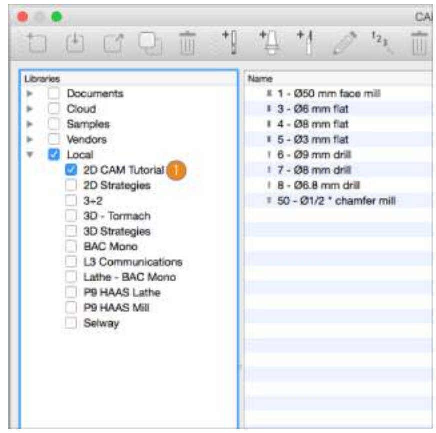


Step 5: – Turn off all other Libraries

1. Click off all other libraries and only show **2D CAM Tutorial.**

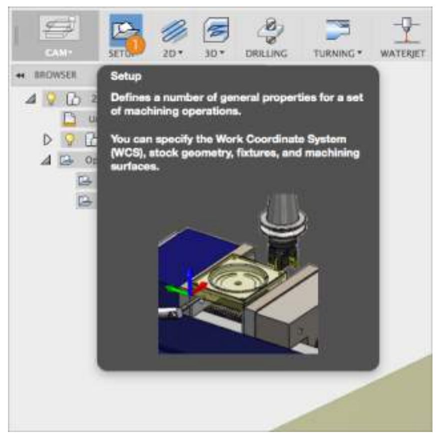
Then…

EXIT OUT OF TOOL LIBRARY



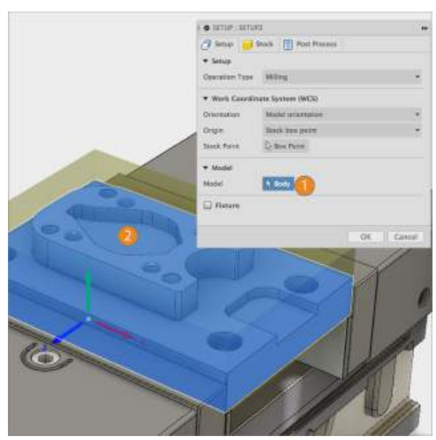
Step 6: – JOB SETUP

1. Click **SETUP**



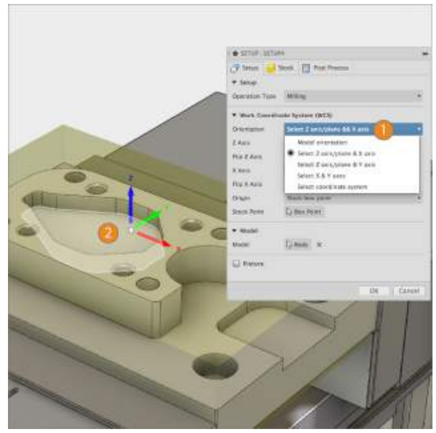
Step 7: – Select the **Part** you want to Machine

1. Under **MODEL**, active the **NOTHING** Icon
2. Select the **2D Strategies Part** in the Screen.



Step 8: – **Orientate** and **Locate** the Work Coordinate System (WCS) correctly.

1. Under Work Coordinate System (WCS), pick the Orientation drop down and select '**SELECT Z axis/plan & X axis**.'
2. Pick the **Highlighted top face**, and the WCS will orientate in the top/center of the part with **'Z' facing north**.

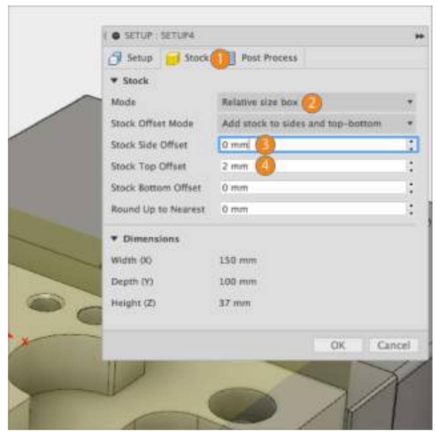


Step 9: – Change **STOCK** Options

1. Click on the **'STOCK'** Tab
2. Under **MODE**, select **Relative Size Box**
3. Under **STOCK SIDE OFFSET**, change to 0 mm
4. Under **TOP SIDE OFFSET**, change to 2 mm

THEN…

CLICK **OK** TO ACCEPT

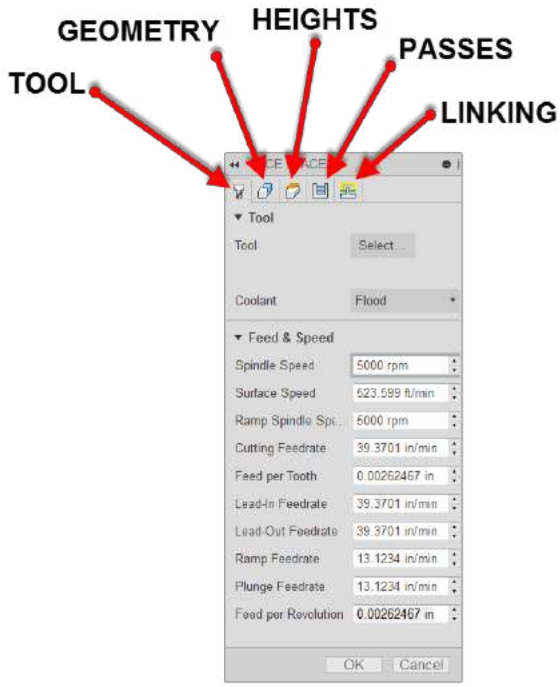


Lesson 2: Toolpath Operations

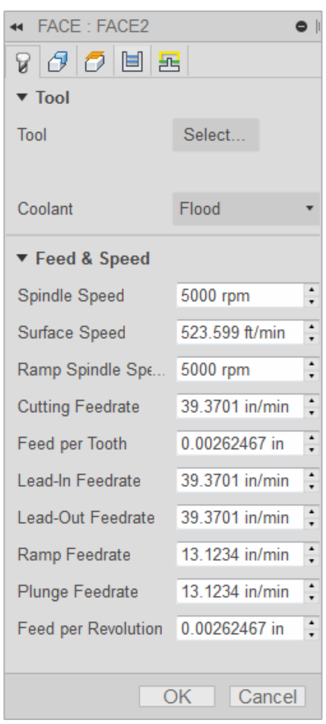
Understanding Toolpaths by Type and Use

Before going further, it is helpful to understand how 2D toolpaths are classified in most CAM software. Please refer to the Autodesk CNC Handbook for more elaborate detail.

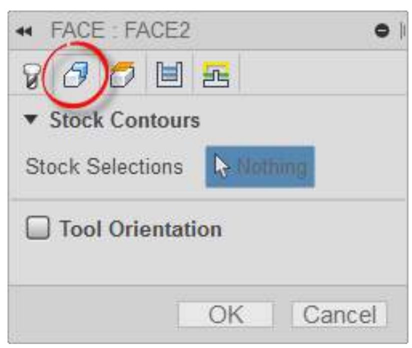
Type	Toolpath	Common Uses
Face	Face	• Finish face of part.
	Island Facing	• Finish face with open sides and bosses.
2D Contour	Contour	• Loops. • Partial loops. • Single edges. • Stick (single point) fonts. • Create dovetail, keyset, or saw cut.
	Chamfer	• Create chamfer using tapered mill or center drill. • De-burring.
	Fillet	• Creating fillet using Corner Round tool.
Pocket	Pocket	• Remove excess material. • Machining TrueType (outlined) fonts and logos.
	Slot Mill	• Straight slot. • Arc slot.
Drill	Drill	• Create spot drill, drill, tap, bore or reamed hole.
	Circular Pocket Milling	• Making holes greater than .75*in* diameter.
	Thread Mill	• Create ID threads over .75*in* diameter. • Create milled OD threads of any size.



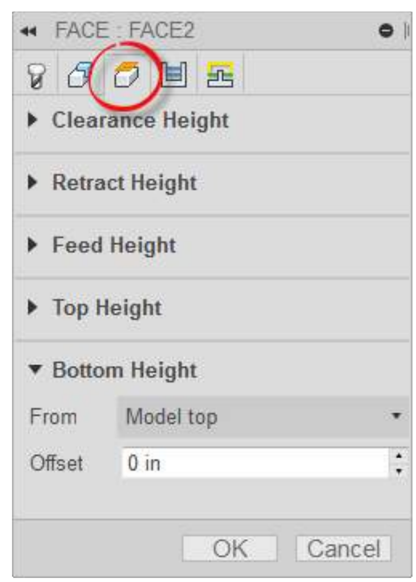
TOOL TAB– Defines the tool being used; as well as the feeds and speeds



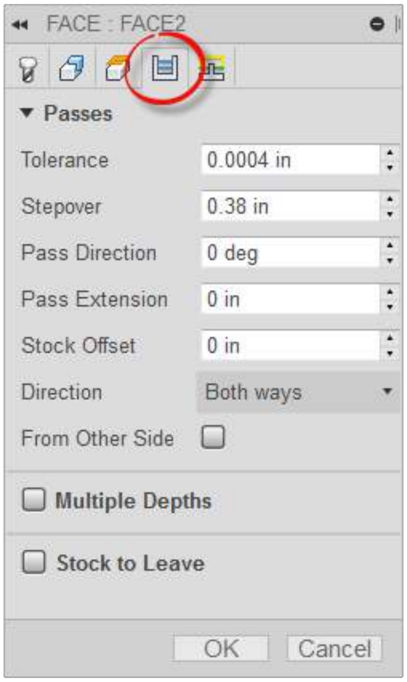
GEOMETRY TAB– Defines geometry being machined.



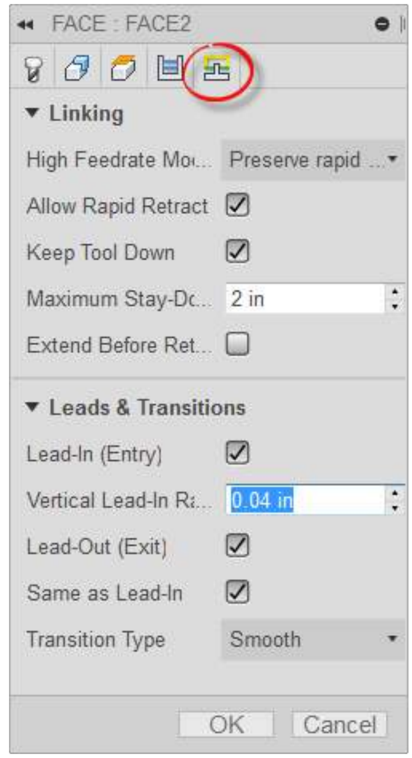
HEIGHTS TAB– Controls heights the toolpath goes to such as cut depth and retract heights



PASSES TAB– Controls how the tool will go about removing material.

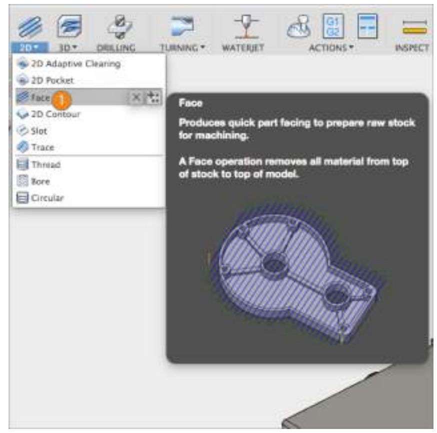


LINKING TAB– Controls how the tool enters/exits and transitions between cutting movements



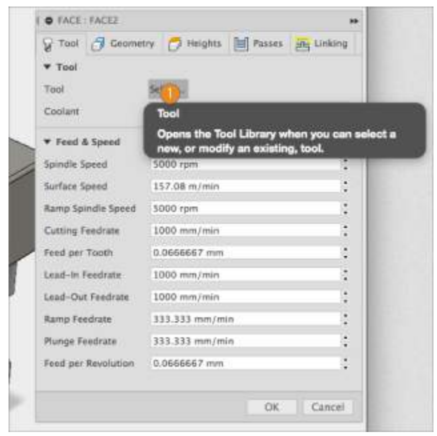
Step 10: – The **FACE** Operation

1. Under **2D Operation**, click on the **FACE Operation**



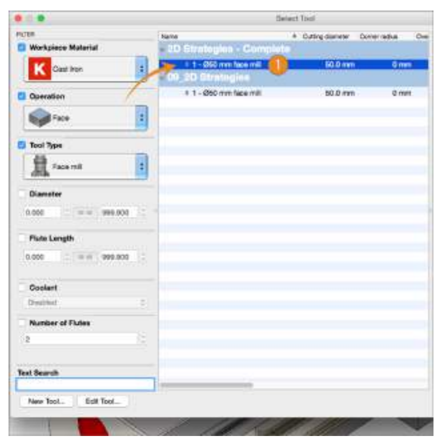
Step 11: – Access **TOOL LIBRARY**

1. Click on **SELECT** under **TOOL**



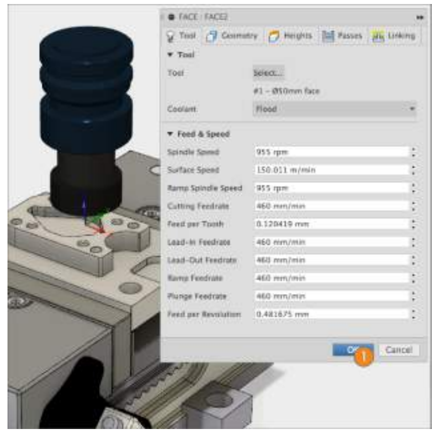
Step 12: – Select a Face Mill

1. Select the **#1 50 mm Face Mill**
2. **Click OK**

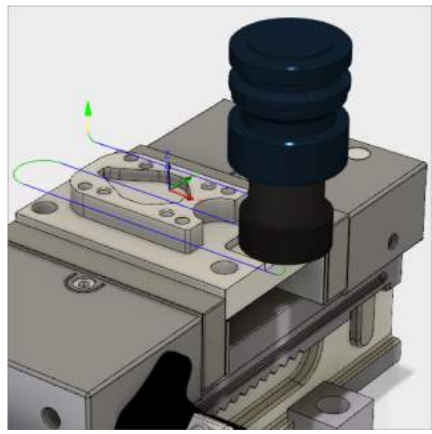


Step 13: – The **FACE** Operation

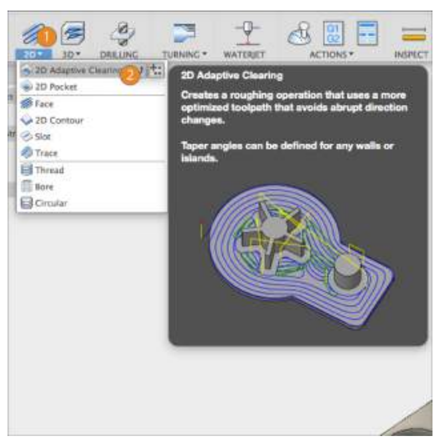
1. Click **OK**



Step 14: – **FACE Operation COMPLETE**

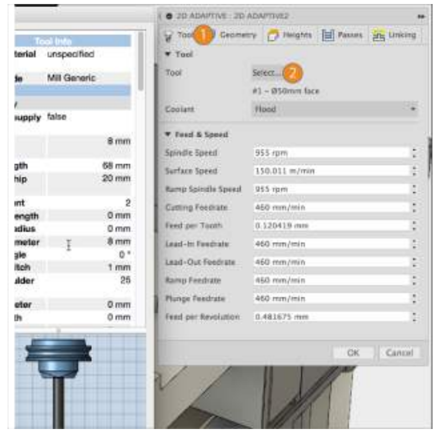


Step 15: – Apply a **2D Adaptive Clearing** Operation



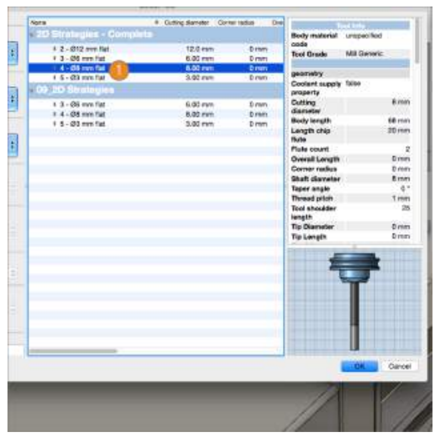
Step 16: – Select a NEW TOOL

1. Click **Tool**



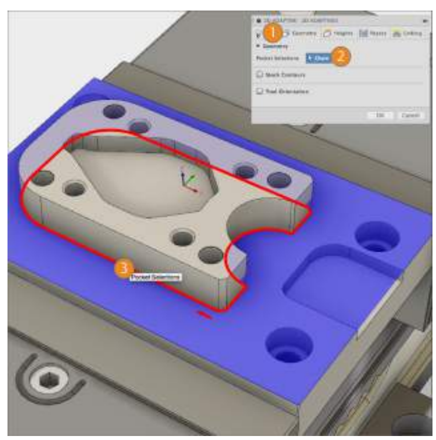
Step 17: – **Select a #4 8 mm Flat End Mill**

THEN CLICK **OK**

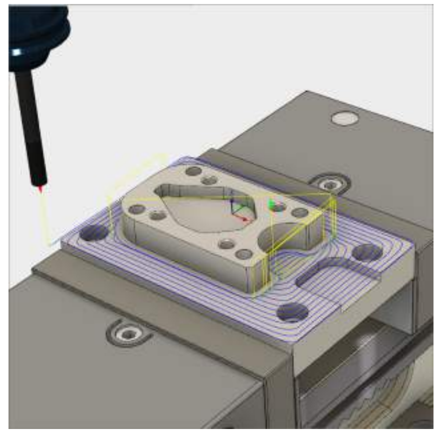


Step 18: – Select Geometry

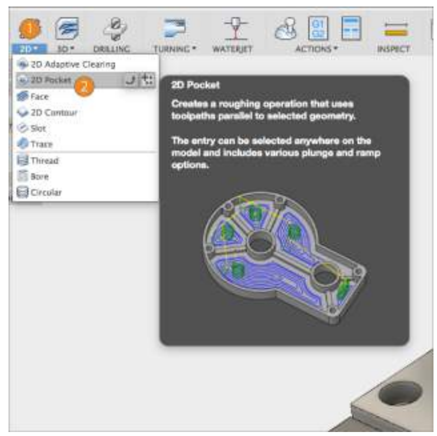
1. Click the **Geometry** Tab
2. Activate **Pocket Selection**
3. Click the outside of the Boss (IN RED)
4. CLICK **OK**



Step 19: – Toolpath Generated

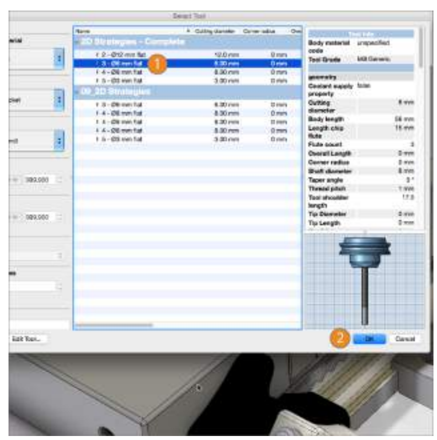


Step 20: – Apply a **2D Pocket** Operation



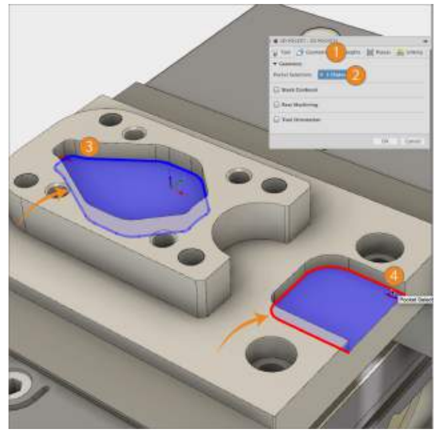
Step 21: – Pick the correct tool

1. #3 6mm Flat End Mill

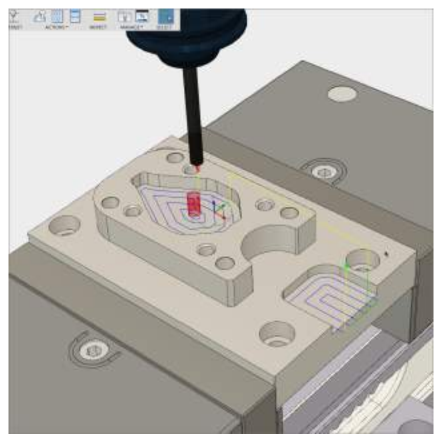


Step 22: – Select Geometry

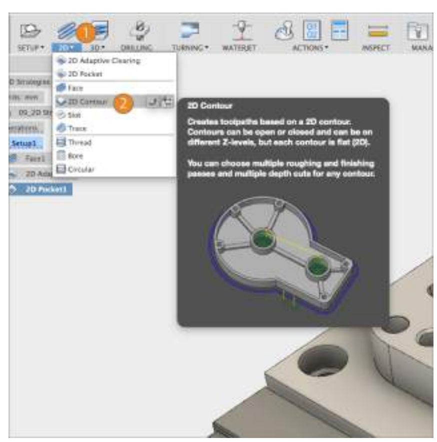
1. Click the Geometry Tab
2. Activate **Pocket Selection**
3. Click the inside embossed bottom **EDGE**
4. Click the bottom **EDGE** of the embossed open pocket (IN RED)
5. CLICK **OK**



Step 23: – Toolpath Generated



Step 24: – Apply a **2D Contour** Operation for a finishing pass.



<table>
<tr>
<td>

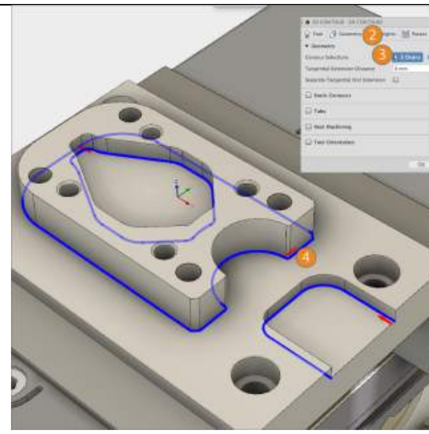
Step 25: – Select correct **TOOL** and **GEOMETRY**.

1. Under the Tool Tab, select a **#5 3mm Flat End Mill.**
2. Click the **Geometry** Tab.
3. Activate **Pocket Selection**
4. Click the **3 EDGES** shown.

CLICK **OK**

TIP: CLICK ON THE RED ARROW TO HAVE THE TOOLPATH FOLLOW ON THE OUTSIDE/INSIDE OF THE BLUE LINE.

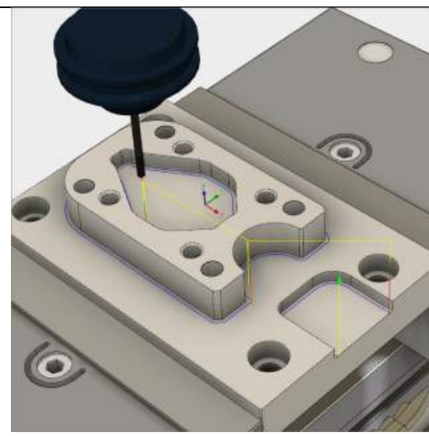
</td>
<td>



</td>
</tr>
<tr>
<td>

Step 26: – Toolpath Generated

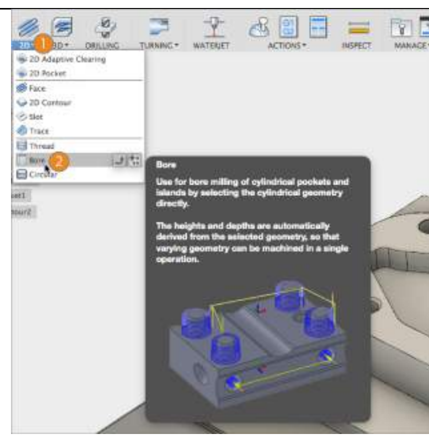
</td>
<td>



</td>
</tr>
<tr>
<td>

Step 27: – Create a **BORE** Operation

</td>
<td>

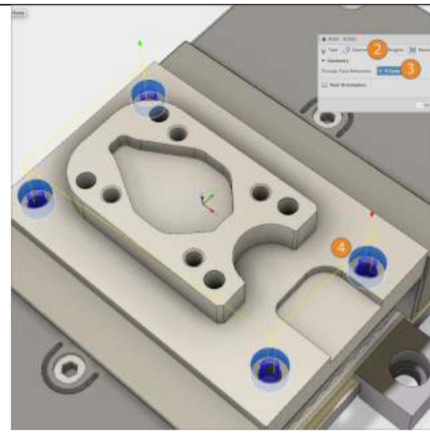


</td>
</tr>
</table>

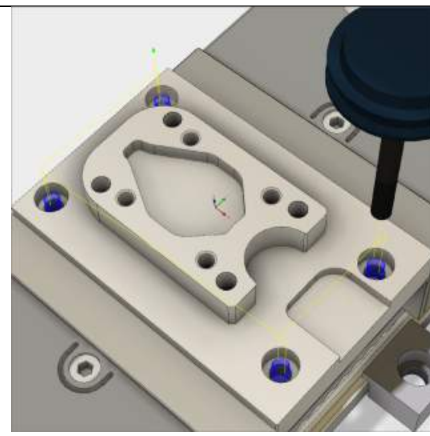
Step 28: – Select correct **TOOL** and **GEOMETRY**.

1. Under the Tool Tab, select a **#4 8mm Flat End Mill.**
2. Click the **Geometry** Tab.
3. Activate **Circular Face Selections**
4. Click the **4 Internal Faces of the Holes** shown.

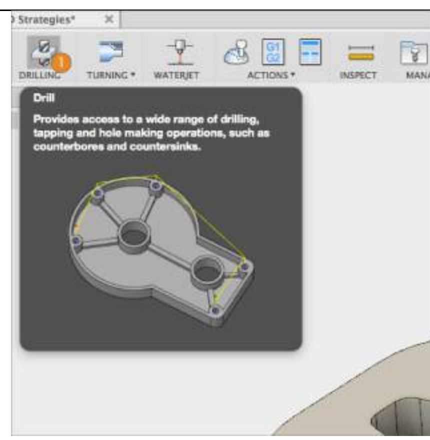
   CLICK **OK**

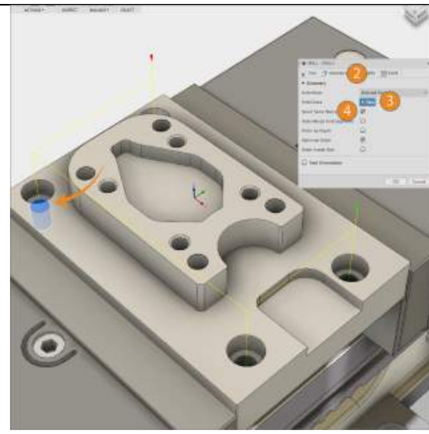
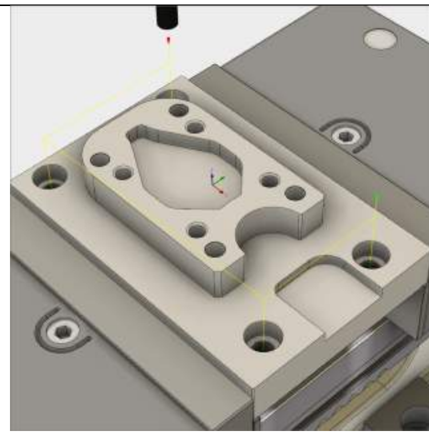


Step 29: – Toolpath Generated

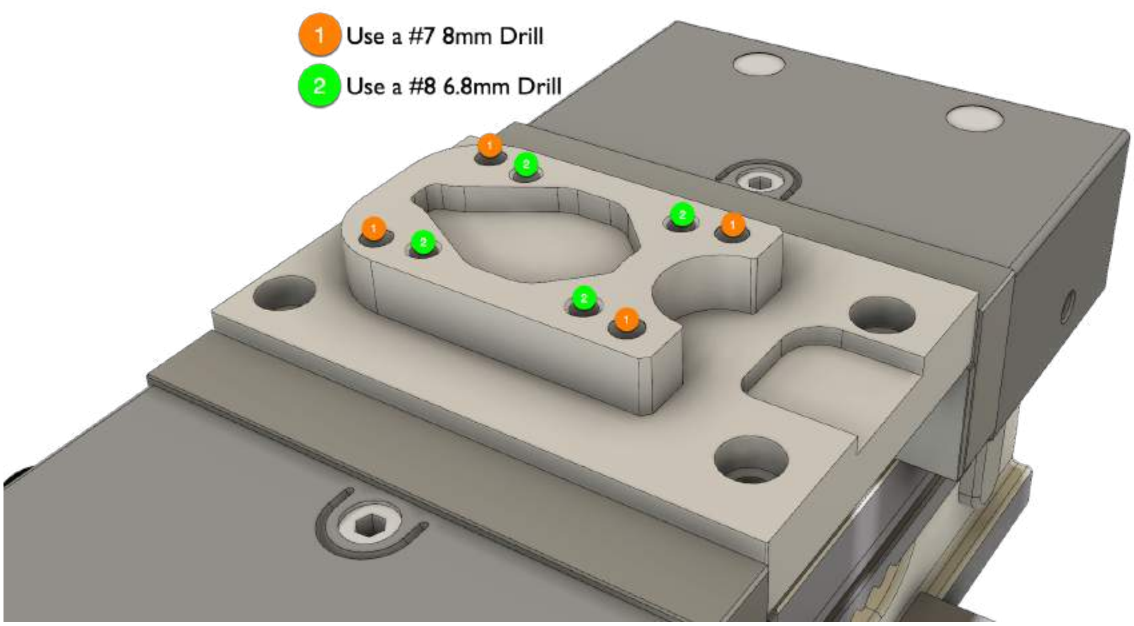


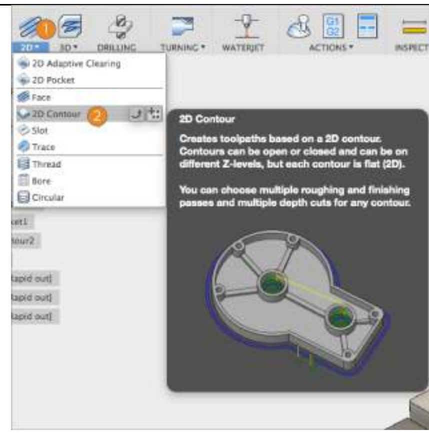
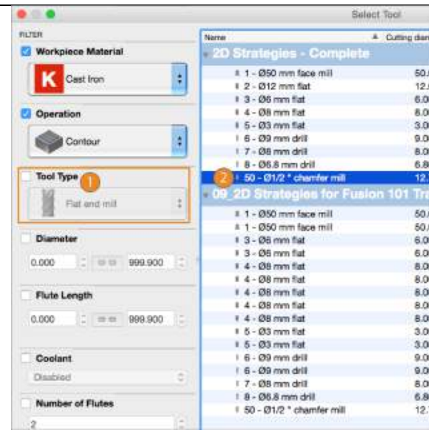
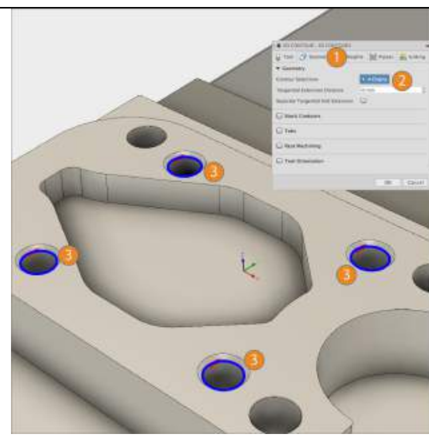
Step 30: – Create a **DRILL** Operation



Step 31: – Select correct **TOOL** and **GEOMETRY**. 1. Under the Tool Tab, select a **#6 9 mm Drill.** 2. Click the **Geometry** Tab. 3. Activate **Hole Faces** 4. Click on **'Select Same Diameter'** 5. Select hole as shown CLICK **OK**	
Step 32: – Toolpath Generated	

Step 33: – Repeat **Step 31**, and use the designed Drill bits below for the holes designated.

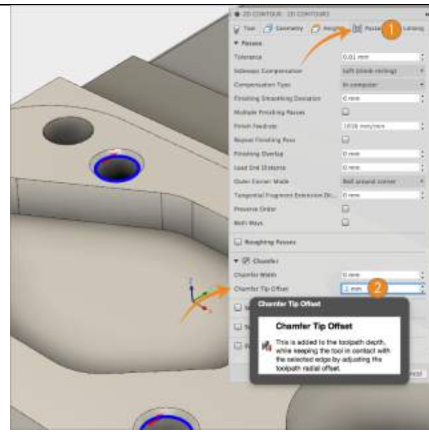


Step 34: – Create a **2D Contour** Operation	
Step 35: – Pick the correct tool 1. Check **OFF** Tool Type 2. Select **#50 1/2" Chamfer Mill**	
Step 36: – Select correct **GEOMETRY**. 1. Click the **Geometry** Tab. 2. Activate **Contour Selections.** 3. Select edges as shown	

Step 37: – Edit the **PASSES** Tab

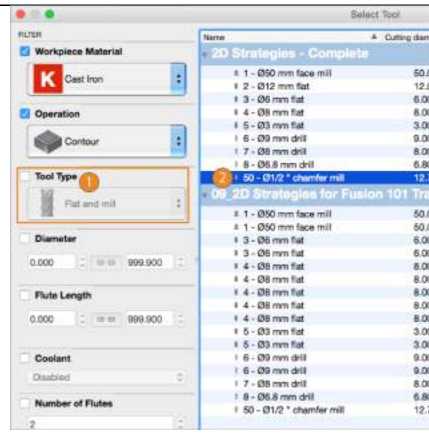
1. Click the **PASSES** Tab.
2. Under **Chamfer**, select **Chamfer Tip Offset**, and type in **0.1 mm**

CLICK OK



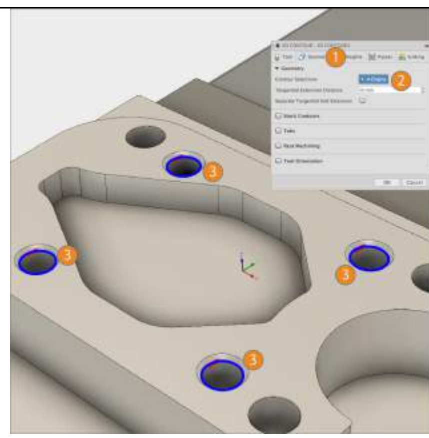
Step 38: – Pick the correct tool

3. Check **OFF** Tool Type
4. Select **#50 1/2" Chamfer Mill**



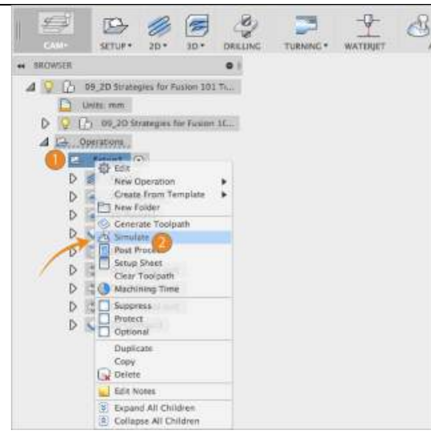
Step 39: – Select correct **GEOMETRY**.

3. Click the **Geometry** Tab.
4. Activate **Contour Selections.**
5. Select edges as shown

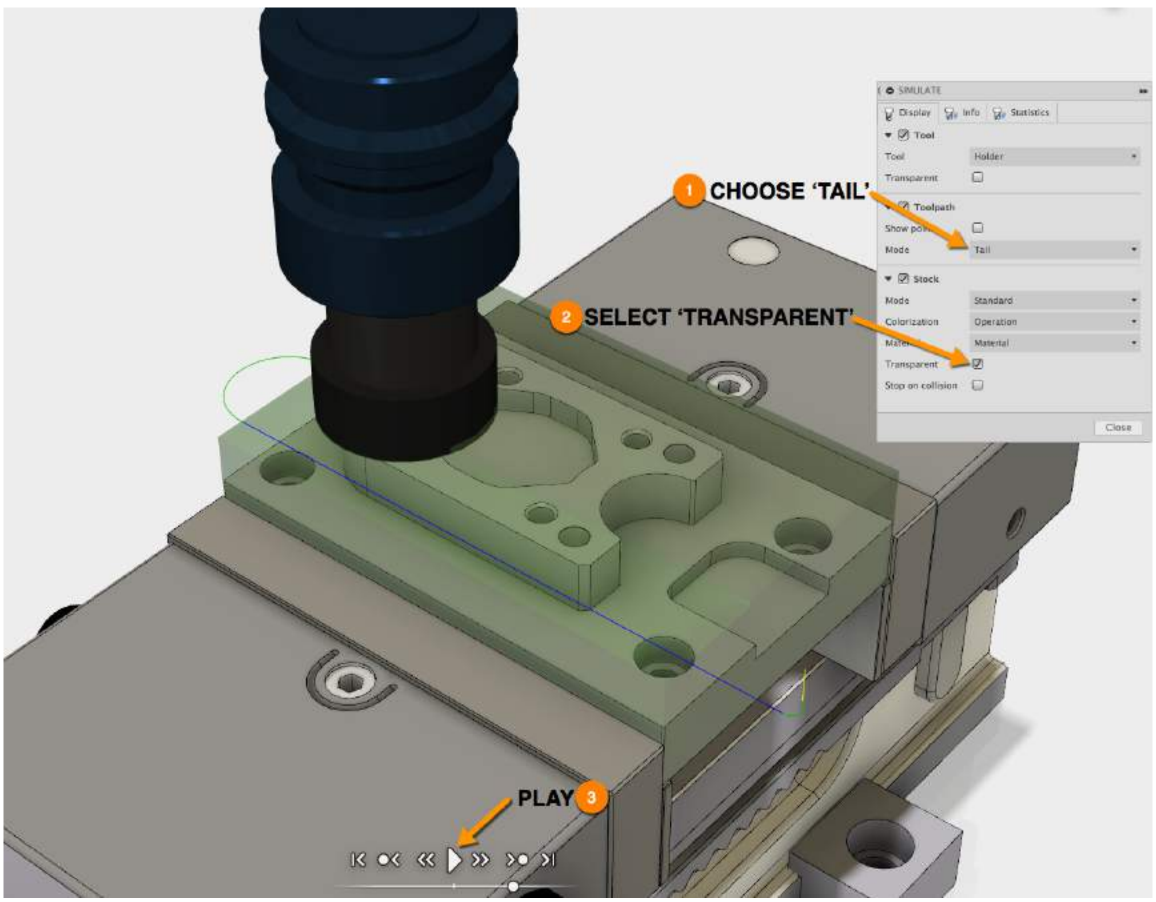


Step 40: – Simulate the Job

1. **RIGHT CLICK** on **SETUP** in the browser.
2. Select **SIMULATE**.

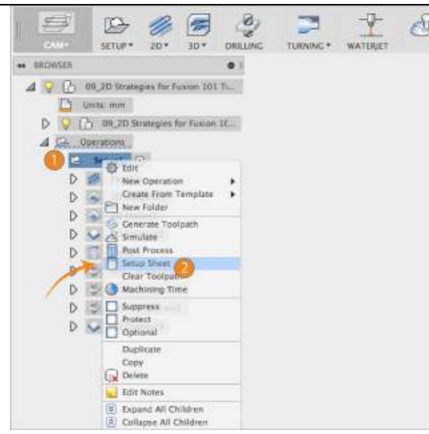


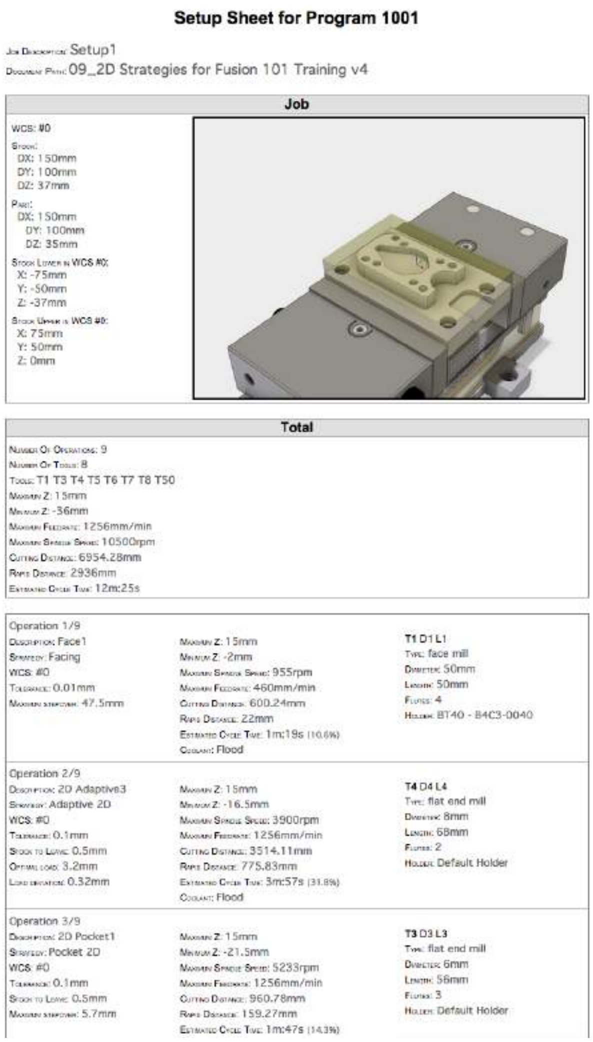
Step 41: – Choose the following **SIMULATE** presets below:



Setup Sheet

The Setup Sheet feature allows you to generate an overview of the NC program for the CNC operator. It provides tool data, stock and work piece positioning; as well as machining statistics.

Step 42: – Create a Setup Sheet 1. **RIGHT CLICK** on **SETUP** in the browser. 2. Select **Setup Sheet** 3. Choose a location where to save it. 4. HTML will be generated of a Setup Sheet (see below).	



Setup Sheet for Program 1001

Job Description: Setup1
Document Path: 09_2D Strategies for Fusion 101 Training v4

Job

WCS: #0
Stock:
DX: 150mm
DY: 100mm
DZ: 37mm
Part:
DX: 150mm
DY: 100mm
DZ: 35mm
Stock Lower in WCS #0:
X: -75mm
Y: -50mm
Z: -37mm
Stock Upper in WCS #0:
X: 75mm
Y: 50mm
Z: 0mm

Total

Number Of Operations: 9
Number Of Tools: 8
Tools: T1 T3 T4 T5 T6 T7 T8 T50
Maximum Z: 15mm
Minimum Z: -36mm
Maximum Feedrate: 1256mm/min
Maximum Spindle Speed: 10500rpm
Cutting Distance: 6954.28mm
Rapid Distance: 2936mm
Estimated Cycle Time: 12m:25s

Operation 1/9 Description: Face1 Strategy: Facing WCS: #0 Tolerance: 0.01mm Maximum stepdown: 47.5mm	Maximum Z: 15mm Minimum Z: -2mm Maximum Spindle Speed: 955rpm Maximum Feedrate: 460mm/min Cutting Distance: 600.24mm Rapid Distance: 22mm Estimated Cycle Time: 1m:19s (10.6%) Coolant: Flood	**T1** D1 L1 Type: face mill Diameter: 50mm Length: 50mm Flutes: 4 Holder: BT40 - B4C3-0040
Operation 2/9 Description: 2D Adaptive3 Strategy: Adaptive 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Optimal load: 3.2mm Load deviation: 0.32mm	Maximum Z: 15mm Minimum Z: -16.5mm Maximum Spindle Speed: 3900rpm Maximum Feedrate: 1256mm/min Cutting Distance: 3514.11mm Rapid Distance: 775.83mm Estimated Cycle Time: 3m:57s (31.8%) Coolant: Flood	**T4** D4 L4 Type: flat end mill Diameter: 8mm Length: 68mm Flutes: 2 Holder: Default Holder
Operation 3/9 Description: 2D Pocket1 Strategy: Pocket 2D WCS: #0 Tolerance: 0.1mm Stock to Leave: 0.5mm Maximum stepdown: 5.7mm	Maximum Z: 15mm Minimum Z: -21.5mm Maximum Spindle Speed: 5233rpm Maximum Feedrate: 1256mm/min Cutting Distance: 960.78mm Rapid Distance: 159.27mm Estimated Cycle Time: 1m:47s (14.3%)	**T3** D3 L3 Type: flat end mill Diameter: 6mm Length: 56mm Flutes: 3 Holder: Default Holder

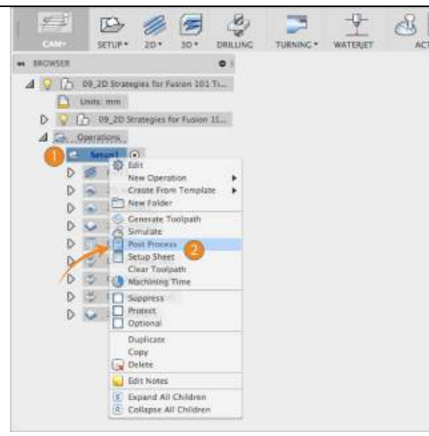
Post Processor

A post processor is essentially a printer driver for CNC machines; a unique configuration file that allows our Post Processor System to turn your programmed toolpaths into CNC programs (G-Code) that your machine control executes to cut parts.

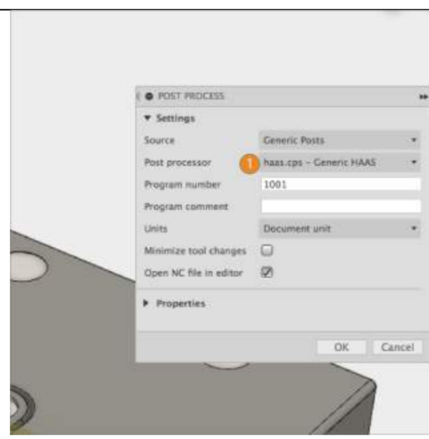
Fusion 360 comes with a standard library of "Posts". These library posts are included because they have been proven to make good parts using standard machine defaults. As the complexity of your setups increases, and you learn more about your CNC, you will probably want modifications made to one of these library posts that produce code in a particular way or with particular options enabled. This requires a post edit. Autodesk has a dedicated Post Development Team that while not working with machine tool vendors to producemore standard library posts, helps our Autodesk CAM Resellers and end-users with postrequests.

For more information on Post Processors, please review the [Autodesk Post Processor Manual.]

<table>
<tr>
<td>Step 43: – Outputting to a Post Processor

1. RIGHT CLICK on SETUP in the browser.
2. Select Post Process</td>
<td>
</td>
</tr>
<tr>
<td>Step 44: – Outputting to a Post Processor

1. From Post Processor, select your Machine

For Inquiries on specific posts, please email cam.posts@autodesk.com</td>
<td>
</td>
</tr>
</table>