EDU466374

How Fusion 360 Helped ITCC Quickly Transition to Virtual Delivery

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Learning Objectives

- Give examples of ways to share data while working virtually
- Describe best practices when giving feedback using tools in Fusion 360
- Create a CNC setup in Fusion 360
- Create toolpaths in Fusion 360

Description

Spring of 2020 was an interesting and stressful period for many professions. Teaching was no exception. Faculty at Ivy Tech Community College had one week to transition courses from in-person to entirely virtual. The use of Fusion 360 allowed this to be possible. DESN 195 (Manufacturing Principles & Design) is a course that takes students through various aspects of manufacturing. Course objectives include: learning about workplace safety, creating G-code, using metrology tools, discussing lean principles, and developing a facility layout. Students typically take this course during their first year. Come join me as we journey through DESN 195 during Spring 2020. See how Fusion 360 allows people to smoothly transition from in-person to virtual, even when creating CNC programs. During this class session I will give examples of what worked well, what did not, and what persons in education and industry can learn from this experience.

Speaker(s)

Josh Nelson is Program Chair and Assistant Professor for Design Technology at Ivy Tech Community College. He thoroughly enjoys working with CAD and teaching and has experience working in different platforms across multiple industries. He excels in ideation, applying known solutions to new problems, and technical communication. Previously he was a Product Engineer at Rebound Technologies, where he focused on system design, construction, documentation, and validation. His expertise particularly helped Rebound develop the system controllers, generate ice maker designs, build system prototypes, and create documentation.

Earlier in his career, Josh gained first-hand experience in both large and small scale manufacturing operations. He also repairs and modifies vehicles—including the construction of an electric vehicle for personal use.

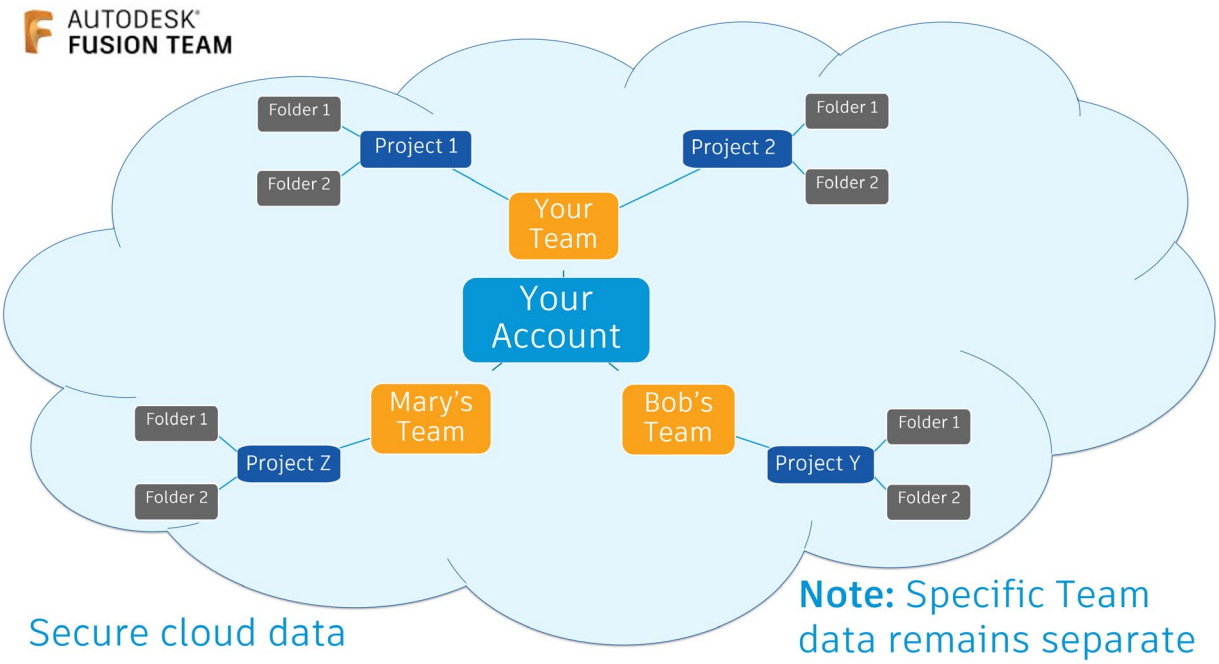
Josh holds a Masters in Sustainable Engineering and a Bachelors in Secondary English Education, both from Purdue University.

Examples of Data Sharing

Fusion 360 makes it easy to share data and work together on designs. There are a number of ways to manage data; below are two of them.

Fusion 360 Data Structure

Any discussion on sharing data within Fusion 360 is incomplete without first mentioning the way Fusion 360 organizes data. The top level of data is the account (typically one per user). Within each user's account there are Teams. Many organizations will have only one Team (that each user would join), but in an academic or contractor setting a user could be part of multiple Teams. As an example, each student may have their own Team. Users can be added to Teams and their permissions can be managed.



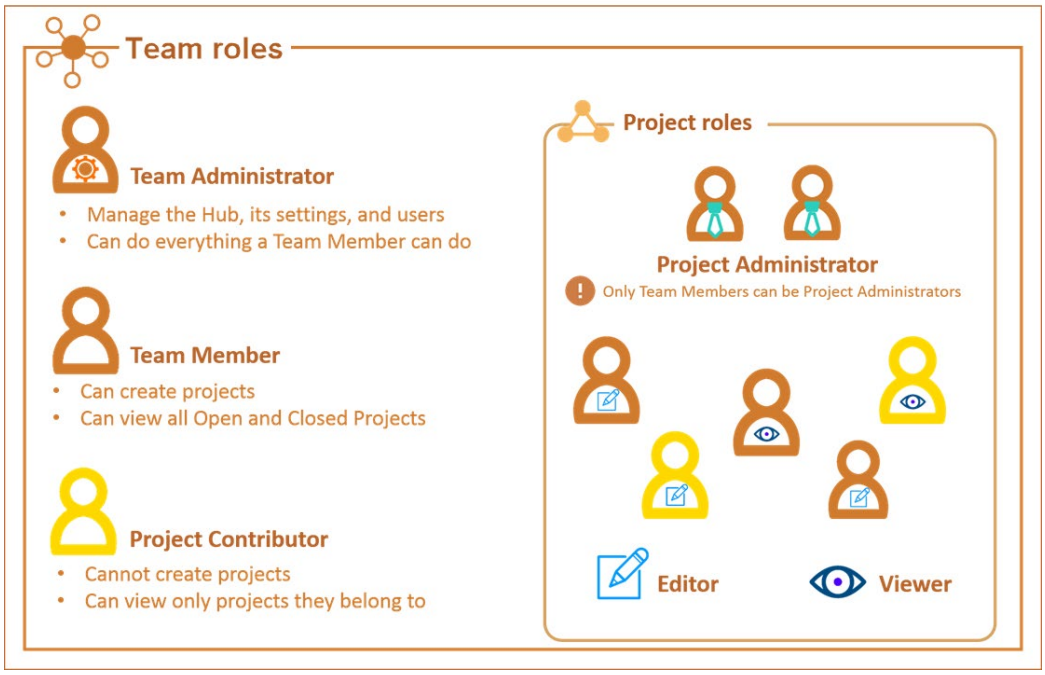
Visualization of Fusion 360's Data Structure

The next level of data is the Project. Projects can be used to organize data within the Team. For example, an instructor might create a different Project for each course, or an engineer may create a different Project for each client. Permissions can also be managed at the Project level. This video demonstrates creating a project and adding a member.

The final level of data within Fusion 360 is the Folder. Similar to mainstream operating system implementations, Folders are containers for storing data. At the time of this writing it is not possible to manage permissions at the Folder level. This functionality is, however, coming soon.

Roles

There are a few roles within Fusion 360. Roles determine what users can see and modify. Both Teams and Projects have different roles and settings. Different roles within the Team are described in detail here; project roles are described here.



Fusion Team Roles

Public Link

One of the ways to share data is via public link. This is most useful when your primary goal is to allow another user to see your design even without access to Fusion 360. For example, maybe you want to show students an example but prevent them from downloading and copying it. In order to do this, expand the Data Panel, right-click on the design in question, and select “Share Public Link”. This video demonstrates the process.

This link is for the design used in the rest of this guide. It was created using the public link functionality in Fusion 360.

Share Public Link Dialogue

Adding People to a Team

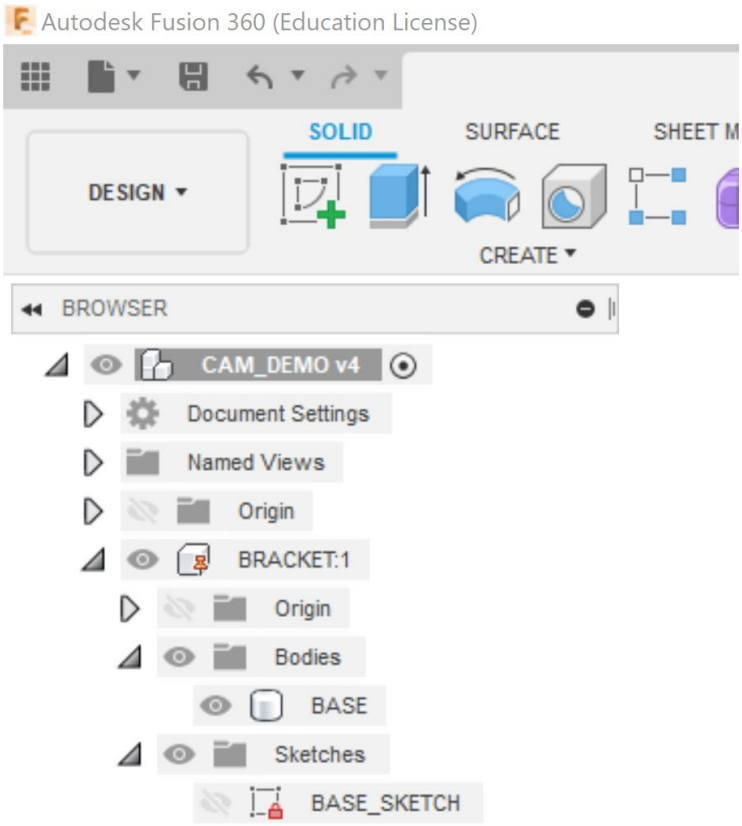
Once a Fusion Team has been created, users can be added to it. Roles and permissions can be adjusted from the Team Settings page. This video demonstrates adding members and adjusting permissions. Although permissions can be changed at any time, once added a user cannot be deleted from a Team.

Faculty may wish to have students use their own Fusion Teams and then add the faculty to them. Alternatively, faculty could create a Team and then invite students. The main difference between these methods is who 'owns' the data.

Feedback Best Practices

Feedback is integral to many processes. This is especially true in education, as students need consistent interaction with others to gauge their progress and solidify their learning.

Feedback starts with setting expectations. Students should be instructed how to document their work. An example of this is naming components, sketches, and features. Naming makes it easier to locate & identify features in the browser and timeline. This aids in editing and commenting.



Named Features in Fusion 360

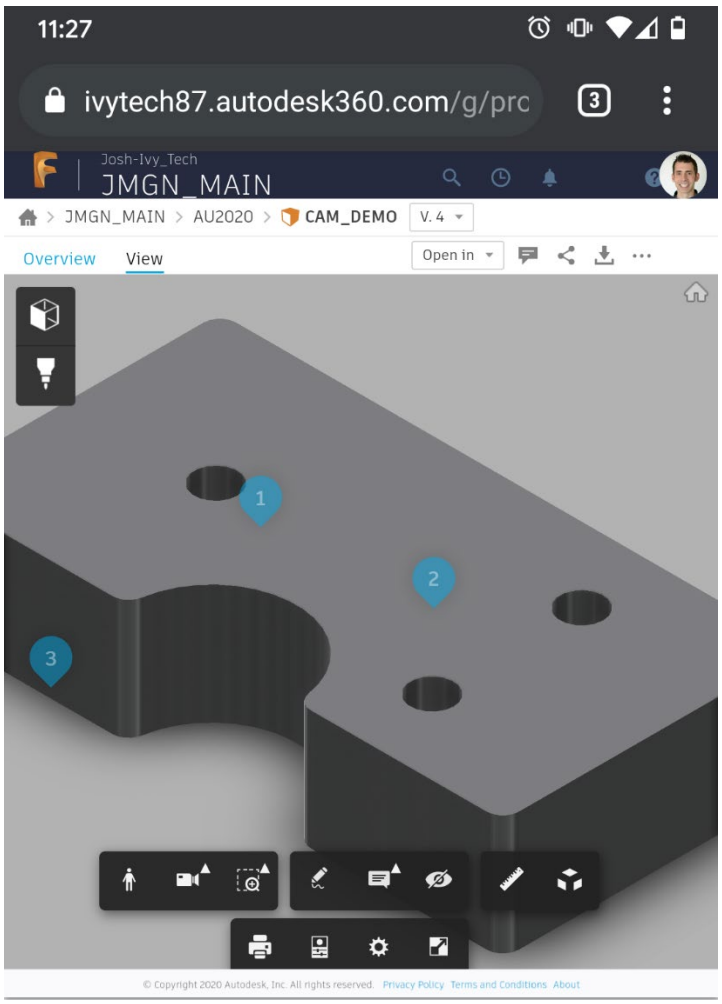
Speed and depth should both be considered before providing feedback. Different kinds of feedback are appropriate at different points in the project. Sometimes it is better to give rapid, brief comments. Other times deep, thoughtful comments are more helpful.

Email, screenshots, and text messages can provide feedback, but there is no record of this in the design. Project members may not all be aware of the feedback or may not be clear on the intent.

Communication is one of the most important aspects of CAD design. Feedback should be crafted in a way to communicate as thoroughly and timely as appropriate for the situation.

Commenting in a Web Browser with Fusion Team

Fusion Team allows users to provide comments and tie them to specific features within the model. This can be done from any device through a web browser. It is important to provide both positive and negative comments in an educational setting. This video demonstrates adding comments in a web browser. Note that Internet Explorer is not fully supported.



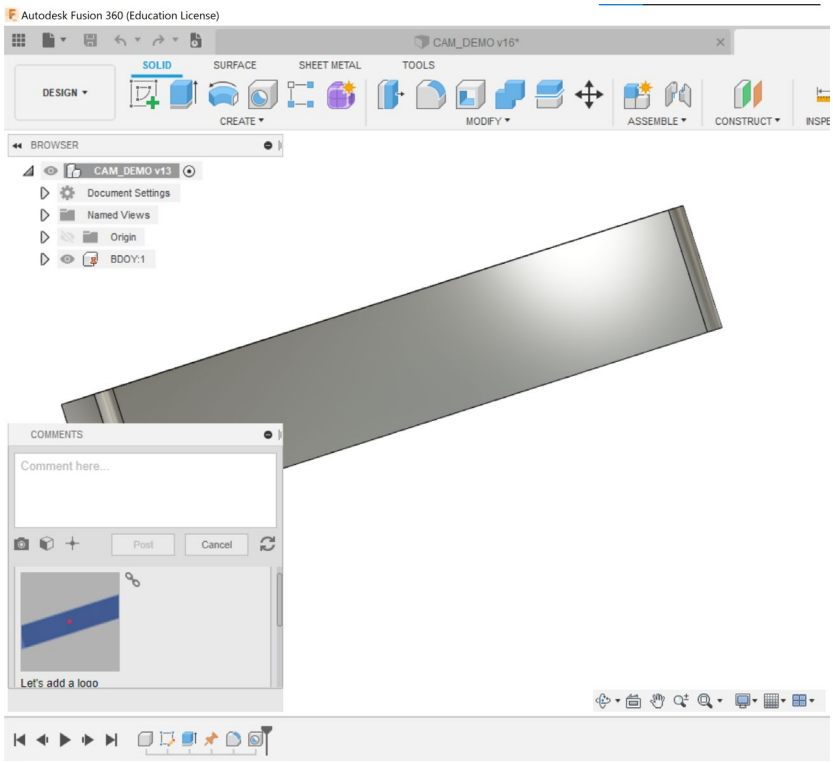
Viewing a Design and Comments from a Mobile Device

Viewing Comments in Fusion 360

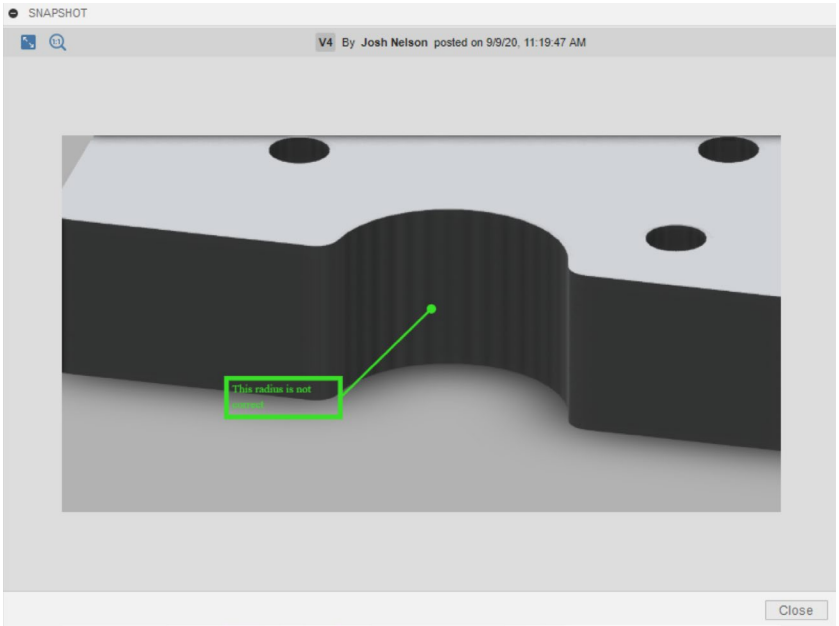
Regardless of where comments are created, they can be viewed within Fusion 360. Comments are shown in the Comment window in Fusion 360 (CTRL + ALT + A). Clicking on a comment pans, orbits, and zooms the model to the view the comment creator saw when creating the comment. When used properly this clearly documents the intent of the comment. This video demonstrates reviewing comments in Fusion 360.

One final note about comments: the way comments are created affects how they will display. If a comment is created via the “Markup” tool, a snapshot of the design with its current view will be saved; if the “Comment” tool is used, the current design view and text note will be saved. Due to

the small size of the Comment window in Fusion 360, “Markup” may be preferred over “Comment” in some situations. Users may wish to experiment with the process and develop their own workflows.



Comment created with Comment Tool in Fusion 360



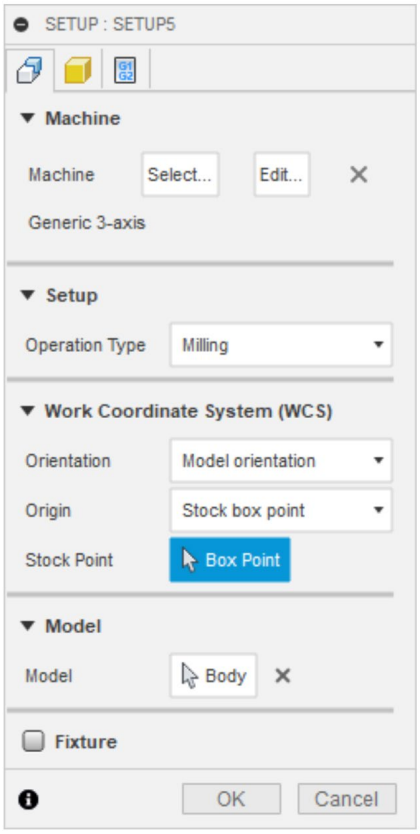
Comment Created with Markup Tool in Fusion 360

Create a CNC Setup

To demonstrate how a part would be machined, this example uses the tools in the Manufacture workspace. The first step is to create a setup. Fusion 360 makes it easy to create a Computer Numerical Control (CNC) setup with few inputs. This allows novice users to create setups, even without physical access to a machine tool.

Most of the inputs in the Manufacture workspace have illustrated tooltips with a good depth of information. If a user hovers over a field, a tooltip will display the name. Hovering over the field for an extended period will open a larger tooltip with descriptions and illustrations (in most cases). This functionality is useful for beginners.

Creating a CNC setup begins by changing to the Manufacture workspace. From there, one selects the Setup icon from the ribbon. This opens the Setup dialogue from which the machine, stock, fixture, and model can be selected. Post processor settings can also be modified. This video demonstrates the setup process detailed below.



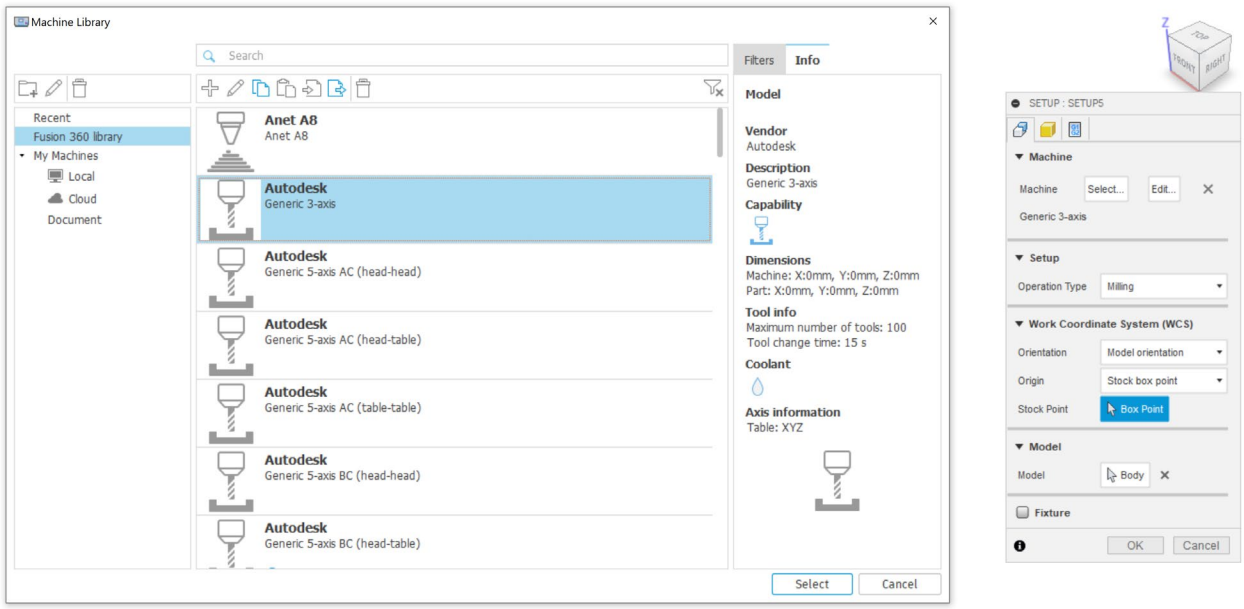
Setup Dialogue

Selecting the Machine

The first step is to select a machine. A number of stock machines are packaged with Fusion 360—including a few generic ones. These can be used as-is or as a starting point for a new machine. Fusion 360 operates most efficiently when utilizing cloud libraries. This is the case for

the creation and sharing of this machine information. An instructor or shop manager could create a machine instance for every machine on the shop floor and then share the library.

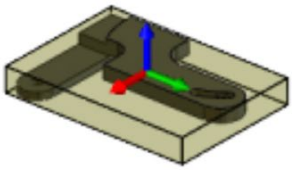
Depending on the machine configuration the Work Coordinate System (WCS) may need to be adjusted. This can be done through the Setup dialogue by changing the appropriate parameters.



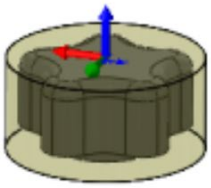
Machine Library

Setting the Stock Size

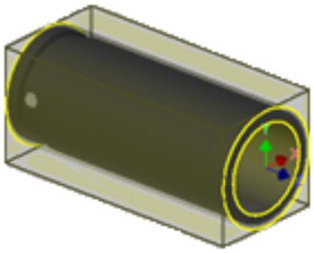
Once the machine is selected the stock size can be selected. Stock shape and size can be adjusted in several ways—including using a body in the design for reference. It is often most useful to select a relative size box and round up to a commonly available size, but the options are practically endless.



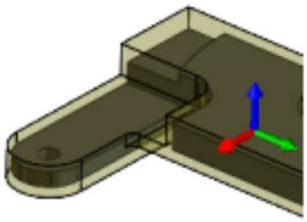
Fixed Size Box - Creates a rectangular stock body of a specific (fixed) size.
Relative Size Box - Creates a rectangular stock body larger than the model by a given set of values. Can be rounded up to the nearest specified increment.



Fixed Size Cylinder - Creates a cylindrical stock body of a specific (fixed) size.
Relative Size Cylinder - Creates a cylindrical
stock body larger than the model by a given set ofvalues. Can be rounded up to the nearest specified increment.



Fixed Size Tube - Creates a tube stock body of a specific (fixed) size.
Relative Size Tube - Creates a tube stock body larger than the model by a given set of values. Can be rounded up to the nearest specified increment.



From Solid - Creates a stock by selecting a solid body in a multi-body part or from a part file in an assembly.

From preceding setup
Creates a stock that matches the stock defined in the previous Setup. This matches the size, but not the orientation or WCS.

Options for Stock Mode

For a single-body design Fusion 360 will automatically select the body as the model to reference in the setup. If an assembly is used as a starting point, the user may need to select the body to reference.

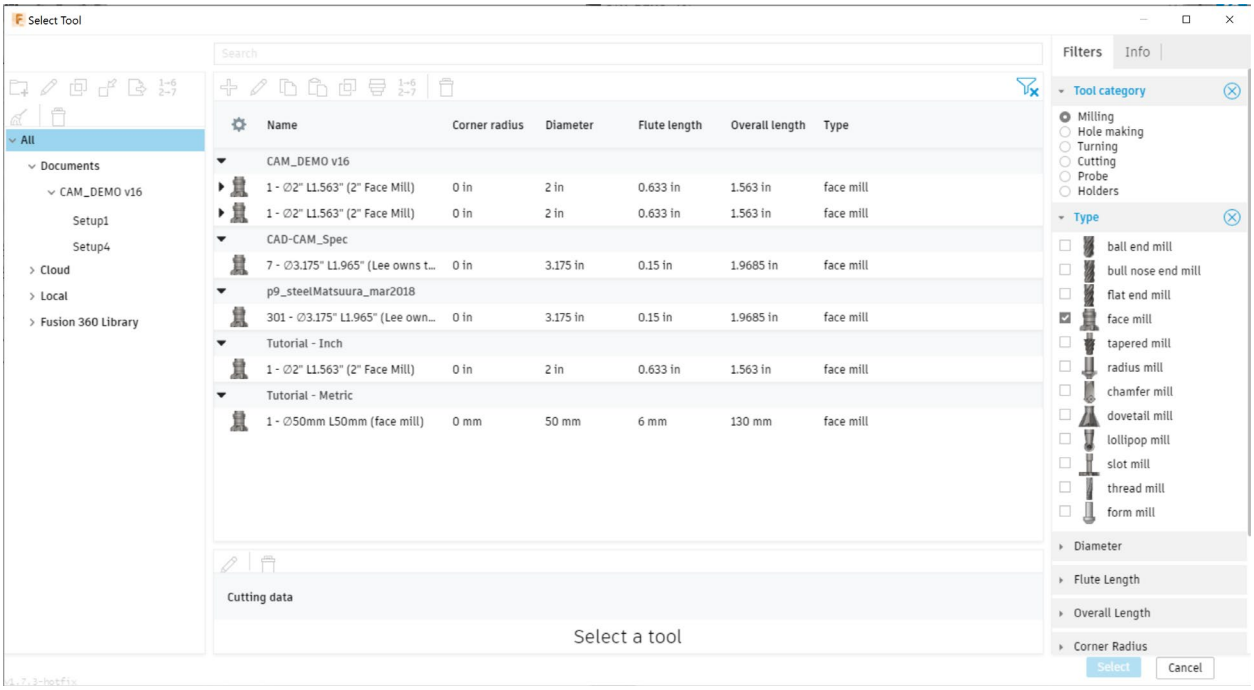
Create Toolpaths

Toolpaths can be created and simulated in Fusion 360—regardless of whether the user has access to a machine tool. This ability allows users to visualize results and make changes to toolpaths without expensive and time-consuming mistakes on the physical machine.

Facing

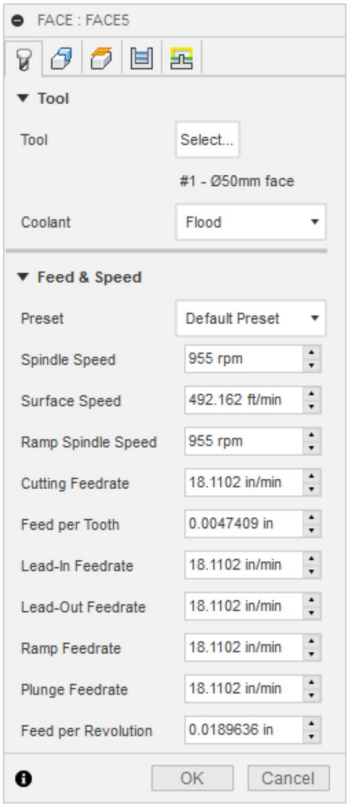
Facing is one of the most common machining operations. It involves removing a relatively small amount of material from the face of a workpiece to ensure flatness, surface finish, and so on. This video demonstrates the steps outlined below.

After selecting an operation (such as facing, drilling, etc.) from the ribbon the first step in creating a toolpath is selecting a tool. As with machines, a number of standard tools are available with Fusion 360 by default. These tools can be used as-is or modified. Typically, a faculty member or shop manager would create a cloud library of tools and then share them with other users.



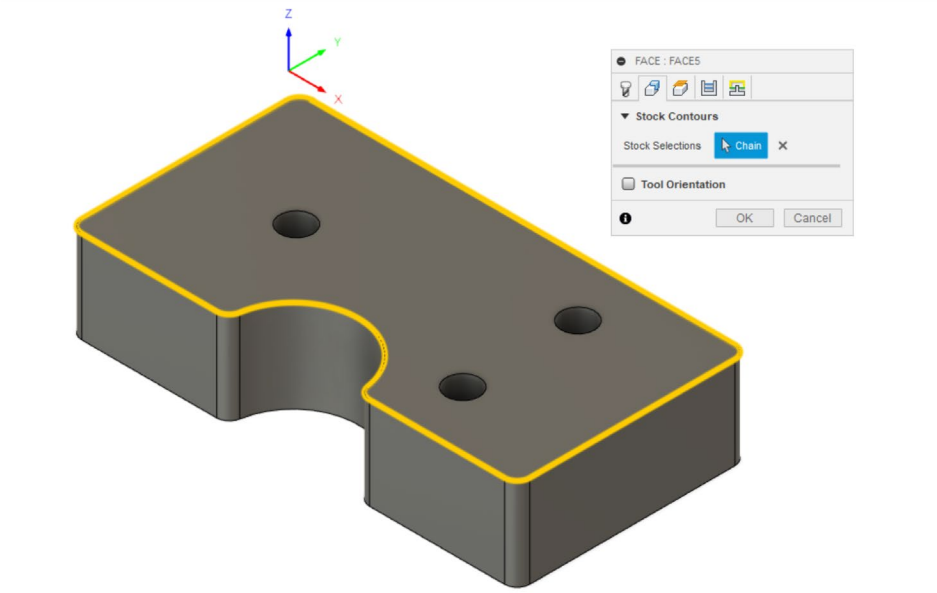
Filtering for Face Mills in Fusion 360

Filters can be used to limit the search results within the library. It is important to remember to clear the filters when moving from one operation to the next to ensure that the correct tools are shown. Once a tool is selected, all the appropriate parameters are loaded into the toolpath dialogue.



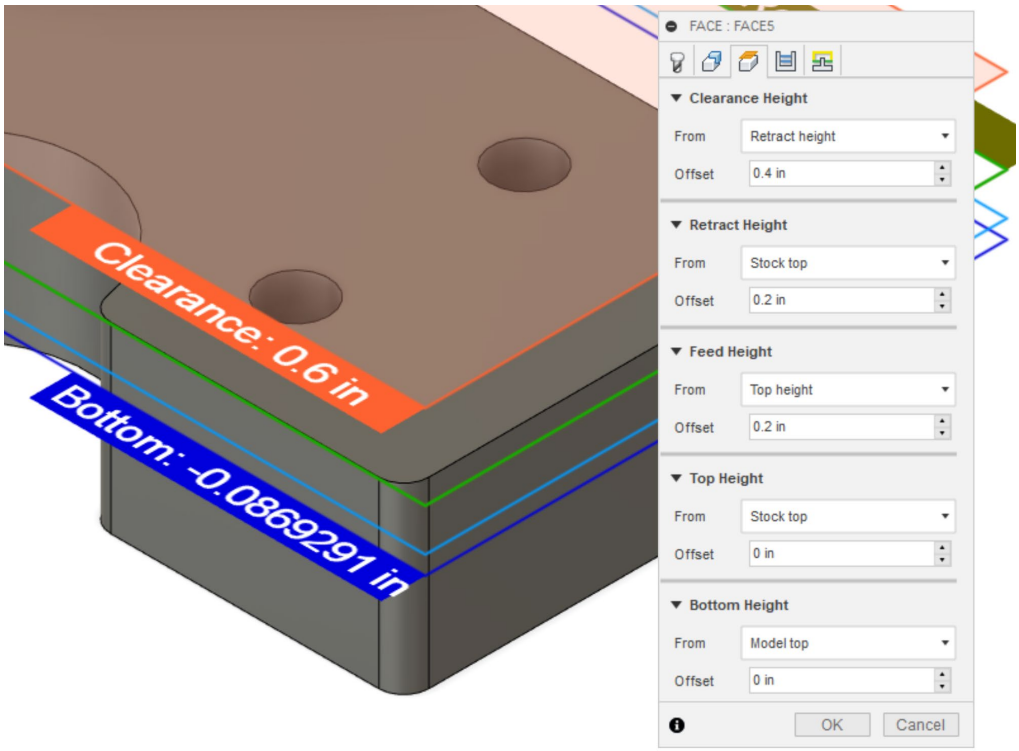
Tool Tab for Facing

The top edge/planar face of the part is automatically selected as the contour in the Geometry tab. This step indicates the boundary within which the tool must machine.



Geometry Tab for Facing

The Heights tab is one of the most important. This information here indicates where the tool should machine down to, where the top of the stock starts, and what height is the tool should retract to when not actively cutting. These parameters are described in a tooltip and shown as colored and labeled planes within the model space.



Heights Tab for Facing

The Passes and Linking tabs can be adjusted or left at their default settings.

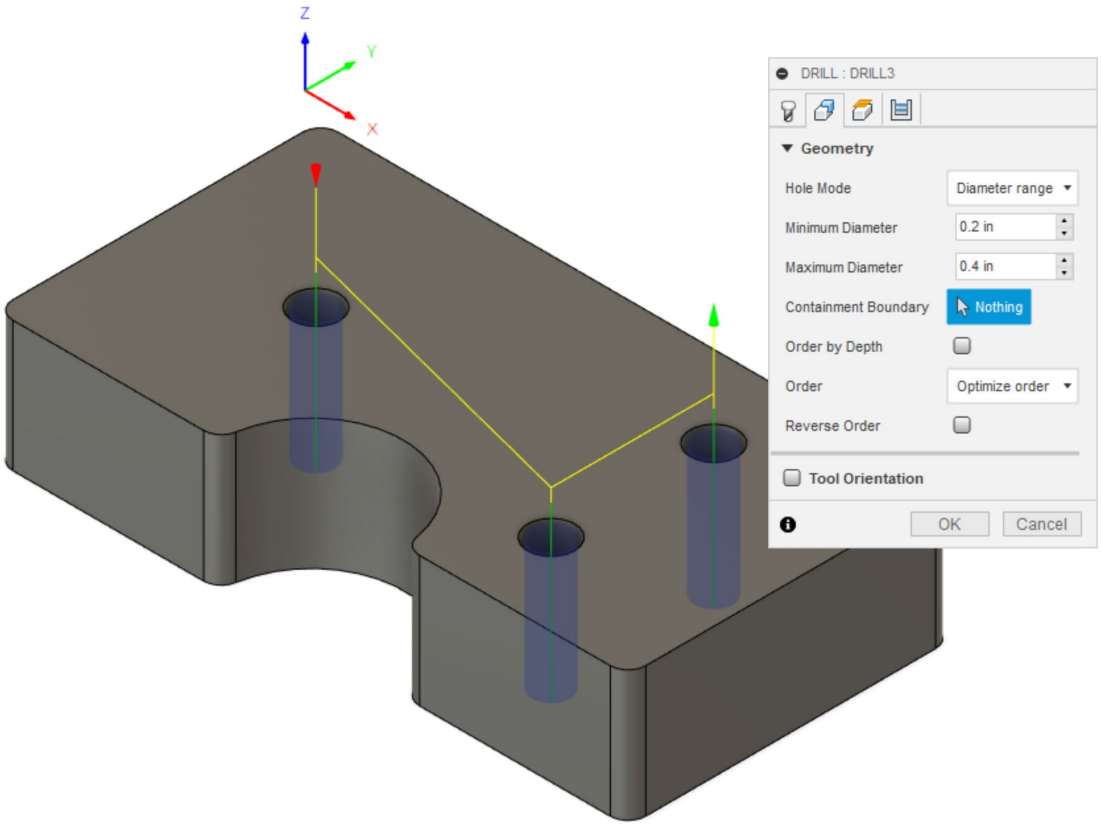
Clicking OK creates the toolpath. Any toolpath can be edited or simulated by right clicking on its instance in the browser and selecting the desired option.

Drilling

The steps to create a drilling toolpath are largely the same as those for milling with a few exceptions. This video demonstrates the steps outlined below.

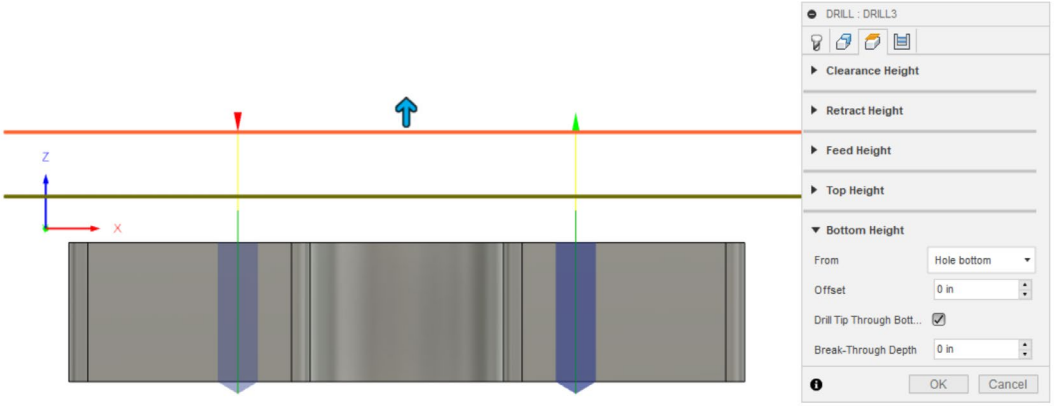
As before, creation starts with selecting the operation from the ribbon and selecting a tool. In this case one would filter for drills of the appropriate size.

The Geometry tab for drilling is a bit different. Here, one can select faces, points, or a diameter range. The diameter range option is particularly helpful, as it allows for increased efficiency when working with multiple holes on the same part.



Geometry Tab for Drilling

The Heights tab also differs from the one when creating a facing toolpath. Here, in addition to the bottom height there are options to drill through the bottom and break through. Note that although drilling through the bottom of a part is desired when creating a through hole, care must be taken to ensure that the vice holding the part and/or the table are not damaged when doing so.



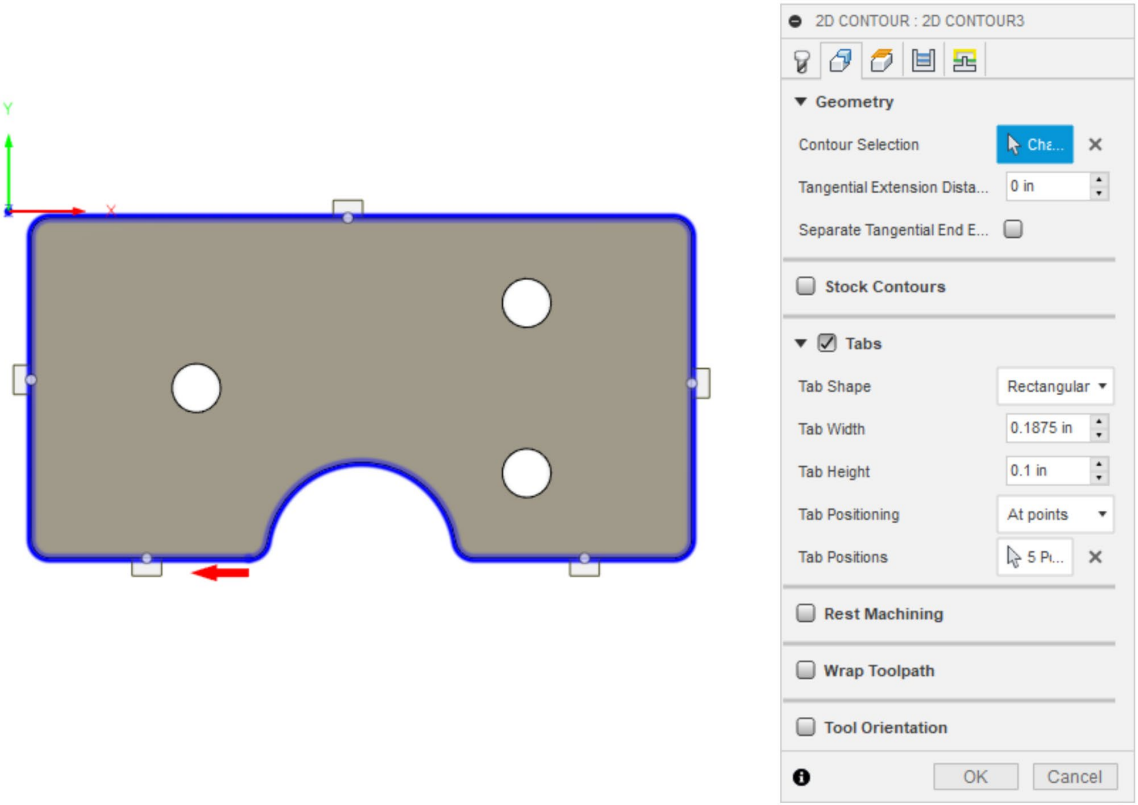
Heights Tab for Drilling

The final tab, Cycle, allows for the selection of different kinds of cycles.

2-D Milling

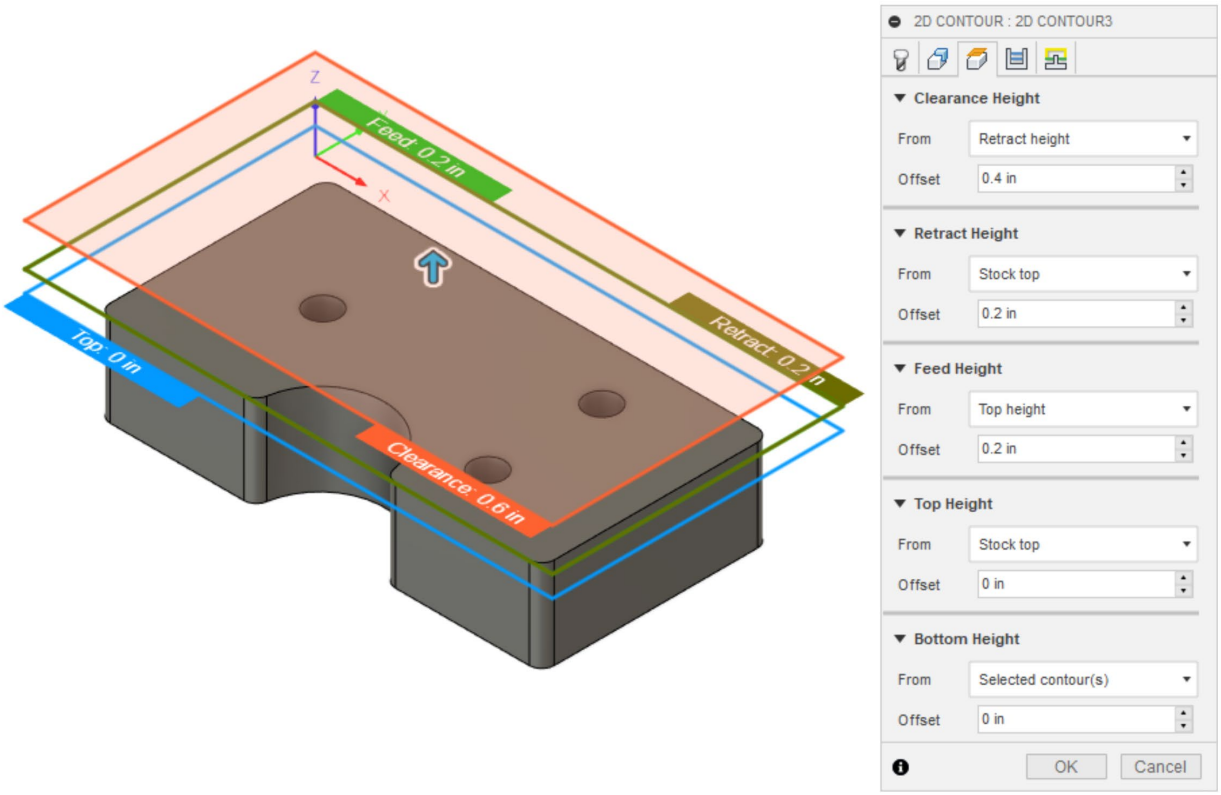
Two-dimensional milling toolpaths are created in a similar fashion. This video demonstrates the steps outlined below.

After selecting the operation from the ribbon and the appropriate tool, the next step is to select the geometry. There are a number of additional options in this dialogue; in this case it is important to select Tabs so that the workpiece does not come loose. The tab geometry can be modified with preset values or picked by clicking on points around the part.



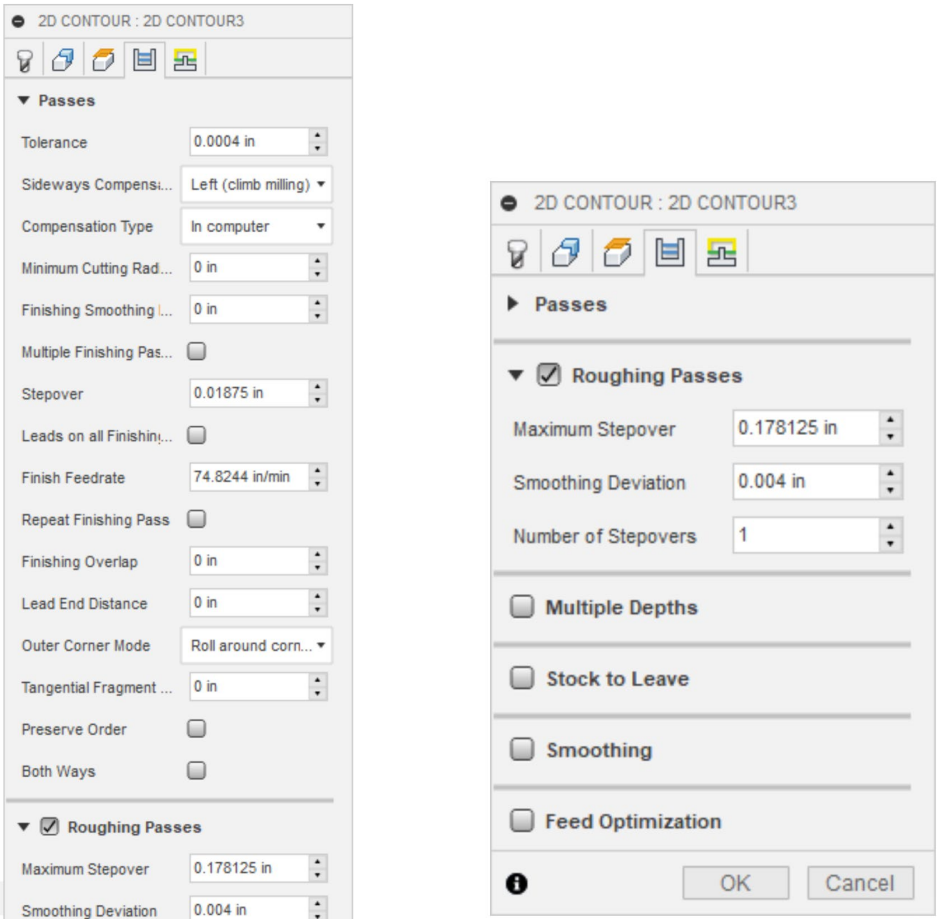
Geometry Tab for 2D Contour

The Heights tab indicates where the tool can cut and where it can travel without contacting material (thereby risking tool breakage, material damage, or both). The bottom height is generally the selected contour.



Heights Tab for 2D Contour

The Passes and Linking tabs can be left with their default values for this demonstration. One note about these tabs in particular (although this can happen in multiple tabs): sometimes the bottom edge of the tab overflows the edge of the screen and can neither be seen nor selected. In this case, collapsing the sections that are not being currently edited will help. Docking the panel to an edge of the window will also help by causing a scroll bar to appear.



Passes Tab for 2D Contour Showing Overflow (left) and a Collapsed View (right)

Lessons Learned

Ask for Help
There are many resources available. Solicit the help of other people, including faculty, Autodesk employees, industry representatives, and students.

Update Yourself
Sometimes it is easier to think that you do not have time to try a new technology. Schedule regular time to try new things; perhaps you will find something useful.

Simulation is not Just for Virtual
Fusion 360 allows for a great deal of learning, especially for entry-level users. One can easily go slowly and see exactly what is happening without having to worry about taking too much time on the machine or wrecking tools.

Think Outside the Screen
Have students organize a “junk drawer” at home to practice implementing Kaizen. Talk with students about proper posture and ergonomics at their homes.

Resources

- Fusion 360 Forum
  - Excellent support from other users and Autodesk employees
  - Responsive community
- KETIV Technologies
  - Consultant, solution provider, and Autodesk reseller
  - Great training videos
- Autodesk Knowledge Network
  - Great for quick, accurate information
- Autodesk Design Academy
  - Specifically, the course entitled “Introduction to CAD, CAM, and Practical CNC Machining”
  - Entire course or individual lessons can be used