Fusion 360 for Makers and Fusion 360 Techniques

Class Notes

Chimera Arts Makerspace classes:

"Introduction to Fusion 360 for Makers" and
"Fusion 360 Techniques for Laser Cutting"

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Presentation Slides:
Introduction to Fusion 360 for Makers

Laser Cutter Constraints

The basic rules of the medium:

- The maximum size of your parts is ultimately limited by the size of your laser's cutting area.

Laser cutter model	Max width (mm)	Max depth (mm)
Chimera's 4x3 laser cutter	1200mm	900mm
Chinese-sourced freestanding (e.g. Redsail X700)	700mm	500mm
Chinese-sourced benchtop laser cutter (e.g. Voccell DLS)	600mm	400mm
Full Spectrum H40 and Muse	508mm	304mm
Chinese-sourced benchtop laser cutter (e.g. LightObject 530)	500mm	300mm
Glowforge	495mm	279mm
Chinese K40	310mm	220mm

Remember that you can't cut a 501mm wide part on a 500mm-wide cutting bed (unless you can lay it out diagonally). Many laser controllers will refuse to initiate such a cut. Some laser controllers will just truncate the cut to the maximum size.

- Limitations in laserable material selection.
- Limitations on material thickness for through cuts.
- Limitations on material height due to physical limitations of laser cutting bed.
- Cuts are 90-degree orthogonal to the surface of the material. This is called "Prismatic".
- Cuts typically are designed to go through the entire material.
  (It is possible to cut part-way through a material, but it requires experimentation with specific materials, cutting power and speed settings.)
- No limits in the planar angles you can make in cuts: inside and outside cuts can be at any angle without the need for dogbone fillets or similar cutting tool compensation such as you might need to create for a CNC router.
- Kerf (the thickness of the material removed by a cut) is minimal (less than 0.1mm). Often you can assume it is zero.
  Experimentation can be used to determine the precise kerf. Kerf can vary based on material, cutting power and speed, laser focus, and sometimes the position of the laser cutting head on the cutting bed. A warped piece of sheet material can create variations in focus thus variations in kerf.
  (To measure your actual kerf: cut a small square of known size (e.g. 20 mm), then measure the actual size of the cut piece.)
  The DXF4Laser add-in for Fusion (see below) will do a good approximation of kerf compensation. DXF4Laser uses rules to guess which edges are inside and outside of joints. DXF4Laser can also be set to 0 kerf. In which case it doesn't do any compensation.
- The edge of many materials such as plywood and wood gets charred or burnt from laser cutting.
  - The burnt edge has a distinctive look. (You can use this for aesthetic purposes.)
  - The burnt edge is dirty and can easily mark clean surfaces.
  - The burnt edge is potentially toxic.
    (Partially burnt organic materials contain toxic PAHs. And burned chemicals such as glues and plastics create toxic byproducts.)
  - The burned edge is also smelly - if it's a burned wood smell that's sometimes OK. But some materials give off a burned smell that's less romantic - burnt leather is particularly offensive.

  You can clean a burned laser cut edge with Isopropyl alcohol or other solvents. Sanding can sometimes be used to clean laser cuts.

- Acrylic gets vaporized with a laser cutter and can leave a smooth glass-like edge. Typically, other plastics melt or burn. But some can be laser cut effectively.
- It's difficult, but possible, to do beveled joints with laser cuts.

CAD Design Tips for 3D Printing

CAD Design Tips for 3D Printing by Billie Ruben and Make Magazine
This graphic is a great summary of do's and don't when modeling for FFF/FDM* 3D printing. (The color scheme is a little confusing: yellow is "YES/good"; gray is "NO/bad".)

*The terms FFF and FDM are effectively synonyms for the type of 3D printing technology employed by desktop 3D printers using extruded plastic filament.
FDM (Fused Deposition Modeling) is a trademark of Stratasys.
FFF (Fused Filament Fabrication) was coined by the open-source RepRap project to describe a similar 3D printing process.

Fusion 360 Tips and Techniques for Laser, CNC Router and 3D Printer

Important Note: as a convention in this guide I have tried to capitalize the names of official things and commands in Fusion 360 to distinguish them from their conventional or common definitions.

• Fusion 360's Tools are like a woodworking shop

With CAD-driven laser cutting and CNC routing you move your craft into the virtual world of the Fusion 360 woodworking shop.

You must learn each tool and feature in Fusion just as you would need to learn each tool and its eccentricities in a woodworking shop.

Be patient, and don't expect that new tools will be obvious or intuitive.

• No best method

Rarely is there a "best" way to build something in Fusion.

Don't let the fear of choosing the wrong way to build something get in the way of your progress.

Experience will improve your modeling efficiency. But you can't gain experience without diving in and building.

• Don't trust dimensions

- Never trust dimensions provided by others. Always question your own real-world measurements.

  Measure twice, model once.*
  **OK, maybe twice. (see next tip)*

- **Expect your first build of a design to be a prototype**
    - Expect the first one to be a prototype. If you nail your design on the first build consider yourself lucky. Realistically, expect the first one to not quite work. Sure you can sometimes salvage a design, but do you want your projects to be known for the near misses?
      Plan your use of materials and resources accordingly.

- **Use little steps to make progress**

  A good way to make progress with your overall CAD and making design skills is to take one or a couple small steps with each project. Try to make each project deliberately more challenging in some specific respect, and be able to articulate what the challenge is. But don't try to take on too much at once. Successful projects are important for motivation.

- **Fusion Gaslighting**

  "Is it me or is it a bug?"

  Don't let Fusion gaslight you.

  Some features just don't work or are very confusing. The lack of complete or up-to-date product documentation doesn't help.

  Solution: Master the sections and features of the program you use. Get to know their options and the details of their behavior, and then gradually expand the range of features you know.

Measurements and Dimensions

- ***Recommendation*: Model in metric units or decimal inches**
- ***Recommendation*: Manufacture in the units of your machine and tooling**
    - When you start Fusion for the first time it defaults to *mm* units. Thereafter, your default units are specified in your user Preferences. There are separate Units settings for the Design Workspace (where you may choose between *mm, cm, m, in* or *ft*) and for the Manufacture/CAM Workspace (where you may choose between *mm* or *in*).
    - Millimeters: It's a good idea to use metric *mm* units
        - Metric is less error-prone than inches.
        - The math needed to manipulate metric decimal values is simpler than the math needed for compound fractions used with inches. (And decimal math can be directly performed with any calculator.)

Consider:

        - 3-7/16" + 5-7/8"
        - 3-7/16" + 5.875"

    vs

        - 87.3 mm + 149.2mm

    - Even when tolerances are not given, decimal values have an implicit accuracy. So decimal measurement accuracy can be managed more consistently.
        - You know 87.3 mm was measured to an accuracy of 0.1 mm.
        - You know 5-3/4" was measured to an accuracy of 1/4", but you don't know if it's any better than that.
    - It is easier to read a real-world physical metric measuring tape than a fractional inch measuring tape.
    - Inch measurements transition from base-2 fractions (1/64th inch) to 1/1000ths aka "thou" or "mils" when additional precision is needed, and while mils are intuitive to machinists, there is nothing intuitive about the transition.
        - There are 15.625 thou in 1/64 inch.
- Inches: On the other hand, there are a few advantages to using inches:
    - In the US most tools are marked or measured with fractional inches, and hardware stores primarily sell goods by the inch or foot. Keep in mind that many "nominal" dimensions can not be relied on as actual dimensions. (¼" plywood is not ¼" thick. 2x4s are not 2" by 4". "1 inch dimensional lumber" is usually closer to ¾")
    - 0.001" (aka a "thou" or a "mil") is a common tolerance for machining metal parts achievable with readily available machine tools. (The metric equivalent is approx. 0.025mm = 25 microns or micrometers.)
    - Approximate measures often require remembering fewer digits with the inch/foot system.
      Consider:
        - 6 inches vs 150 mm
        - 3'-4" vs 1016 mm
    - Because the inch/foot system is based on multiples of 2 and 12 there is often math you can easily do in your head.
      For example,
        - 10 feet divided by 3 = 3 feet + (1 foot divided by 3) = 3 feet 4 inch
- In Fusion 360 you can override your units default preference and change the units for a specific Design in the Document Settings field at the root of the Browser in the Design workspace. Likewise, you can override the default preference for Manufacture/CAM with the Browser's Cam root Units setting. These two units settings are independent. So

you can easily model with *mm* in the Design workspace and machine with inches in the Manufacture/CAM workspace.

- Fusion allows you to easily input values in any supported units no matter the Design or Manufacture units settings. Internally the values will be converted and stored in the correct units.
  CAD and CAM operate in a virtual world where the coordinates and dimensions of objects are just numbers in the computer, and the computer can easily translate and interoperate between different unit systems. So if your CNC system is set up for inches you can use inches in Manufacture/CAM and still model in metric. It really makes little or no difference to the computer. Just be careful to keep track of which units you are using for what.
- But try not to mix units within a Design. While Fusion can handle whatever units you input, mixing units creates a confusing workspace for us humans. The display of units in a Design (e.g. the current selection info in the lower-right of the workspace or in the Inspect->Measure tool) is determined by the Browser - Document Settings: Units.
- User Parameters carry an intrinsic Unit type and the Values displayed in the User Parameter dialog are always in that unit irrespective of the Design's Units setting. It is difficult or impossible to display a User Parameter in any other unit than its intrinsic units, and you can't change a User Parameter's intrinsic units. But you can input an Expression into a User Parameter using other units.
    - Example: User Parameters

name	*unit*	*expression*	*value*
Width	in	100.00 mm	3.937
Height	mm	50.00 mm	50.00

      When used the value will be converted as necessary to the Design's units setting.
- Entering compound fractional and mixed feet-inch units:
    - To enter compound fractional inches:
        - Type "1<space>1/4 in" for 1-¼ inch
        - Or type "1+1/4 in"
    - To enter feet with compound fractional inches:
        - Type "3 ft + 4<space>5/16 in" for 3' 4-5/16"
    - You can substitute " for in and ' for ft.

Accuracy

The approximate accuracies possible with processes using well maintained machines:

- Laser Cutting: 0.004" = 0.1mm
- CNC Routing: 0.005" = 0.125mm

    - CNC Milling: 0.001" = 0.025mm
    - FFF/FDM 3D Printing: [0.1mm on the Z-axis and 0.3mm on X and Y. 0.05mm with additional calibration.]

- **Essential Tools - real-world**
  - A notebook with graph paper. Experiment with different sizes, grids, papers until you find what works for you.
  - Good pens or pencils. Again, experiment until you find what you like.
  - A scale/ruler with metric and inches.
  - A tape measure with metric and inches. (These are often hard to find at Home Depot et al. But they can be found online, and IKEA has one for under $2.)
    I find this pocket-size model handy:
    https://www.amazon.com/Keychain-Tape-Meas-6-2m/dp/B01CGP5XFI
  - Dividers/calipers (analog) - these look like a drafting compass but with sharp points at the ends of the arms. They are used to pull small measurements off of real world things.
    Example:
    https://www.amazon.com/Toolmakers-Precision-Dividers-Calipers-Compass/dp/B0181SPG88
  - An angle measure or protractor - to measure angles of real world things.
    https://www.amazon.com/General-Tools-29-Plastic-Protractor/dp/B00004T7P5
  - An adjustable sliding T-bevel is useful to pull angles off real world things. Then the angle can be measured with the protractor (above).
    - Generic:
      https://www.amazon.com/Degree-Sliding-T-Bevel-Carpenters-Protractor/dp/B08J82SYDT
    - This Shinwa model has the locking screw at the end rather than the side. So it's out of the way.
      https://www.amazon.com/Sliding-T-Bevel-8-Blade/dp/B0037XS27A
  - Digital calipers.
    - The Mitutoyo 500 series is very nice but costs $125 for the unit that can measure 150mm/6 inches.
      https://www.amazon.com/Mitutoyo-500-196-30-Advanced-Measuring-Resolution/dp/B00IG46NL2
    - Inexpensive units are available from other manufacturers, and they generally work well with some compromises in convenience, feel and battery life.
    - Harbor Freight has a very good digital caliper for $9. It's mostly plastic, but it works, and if you drop it you can just get another. No tears.
      https://www.harborfreight.com/6-in-composite-digital-caliper-63586.html

- **Fusion 360 Preferences**
    - *recommendation*: It's a good idea to get used to metric *mm* units. (See above)

    - *recommendation*: use Perspective with Ortho Faces. The display projection snaps to Orthographic mode when the view is looking directly at a canonical face such as Front, Top, Left etc. (for example, via clicking one of the faces of the View Cube). When the view is shifted off of the direct view of a canonical face it will switch back to the more normal-looking Perspective projection.

    - *recommendation*: It doesn't matter which axis configuration you choose for "default modeling orientation". Choose the one you are comfortable with, but you should probably be consistent.

      Y-up is the traditional CAD industry standard (remembered by the mnemonic "Wise Up!").
      Some people like to use Z-up because that is what a CNC router or laser cutter uses. But those machines are a long way from your Fusion 360 design workspace.

      When you are designing in Fusion 360 you are typically designing toward your final assembly and part configuration where the part axes won't match the manufacturing axes. (Consider a chair where the legs are cut from ¾ inch sheet material on a CNC router. You are going to model that chair with legs upright, not flat as they will be during cutting.)

      When you use the Manufacturing/CAM workspace in Fusion 360 you must create Setups for cuts. You must define the cutting axes in that process, and the choice of axes can be independent from the design space axes.

      There are some things in Fusion 360 that define the Z-axis as up (or "normal") from the plane. Joints, for example, always define Z as normal to the Joint Origin. Manufacturing (CAM) Setups often define Z as normal to the surface (although there you can define other axis orientations).

      Note: you can't change the modeling orientation after you have created a new Design.

- **Snaps, Grids and Sketch Constraints**
    - Command on Mac/Ctrl on Windows suppresses grid snapping while the key is pressed.
    - Turn on Center and hold Shift in Sketch to center to midpoint.

• ***Tip*: Set grid to Material Thickness**

When modeling for laser or CNC with uniform material try setting the Grid to your material thickness.

- In the Navigation Bar -> Snaps and Grids -> Grid Settings
- Set to Fixed.
- Major Grid: material thickness (e.g. 5.0 mm)
- Subgrid: 1 (no subgrid) or 4 (quarter grid)

Remember that Snaps and Grids settings are persistent between Designs.

• ***Tip*: When modeling for sheet material set the Subgrid to 4 for half and quarter-width snaps.**

If using the above tip to set the grid to material thickness, try setting the Subgrid to 4. That will enable grid snaps for ½ and ¼ material thickness.

Remember that Snaps and Grids settings are persistent between Designs.

• **Double-click Middle Mouse Button or F6 performs a Fit to view**

• **Orbit Center**

The Orbit Center is indicated by a small green dot while in Orbit-drag mode.

Orbit Center starts at the center of mass. But Orbit Center migrates through a complicated algorithm following a series of orbits, pans and zooms. Sometimes it will migrate to an awkward location.

You can reset the Orbit Center with a right-click on an empty place in the workspace and choose "Reset Orbit Center".

Also you can explicitly place the Orbit Center at a specific point, but this can sometimes be difficult to control unless the desired point is at a corner or edge. Right-click on an empty space and select "Set Orbit Center" followed by clicking on the desired point.

• **User Parameters**

It is useful to create a user parameter for any dimension that will be repeated throughout your model. That way you won't mistakenly use different values in different places.

Create a parameter called "materialThickness" or "mt" and set it to the thickness.

You can also create parameters for other design values that will be useful such as "height" and "width".

Some people use short abbreviations like "h" or "w". It's ok if it's obvious what it means. But it is best avoided for anything not obvious or commonplace. For example, don't create a parameter called "bto" instead call it "boxTopOverhang". Fusion will do word completion for the user parameter names as you start to type them.

I often make user parameters for hard external constraints such as measurements of physical world objects with which your design will ultimately have to interface. e.g. If you were making a box that had to fit exactly within a drawer. Then measure the inside dimensions of the drawer and create user parameters for them.

Hardware dimensions are another area where user parameters can be helpful -- e.g Outside diameters for bolts or screws.

I also find it handy to create user parameters for human factors such as minimum handle widths or finger widths.

Creating Parameters in the Creation Dialogs
Fusion 360 recently added the ability to define new parameters when inputting parameters into an object creation dialog. For example, when inputting the dimensions of a Box you can now enter "boxWidth = 100mm" to define a parameter called "boxWidth", to set its value to 100mm and to define the width of the new Box object.

Parameters are not magic. Robust parameter-driven models are hard work.
Don't expect your model to magically update if you change these parameters later. It might, but chances are it will not work properly.

Robust parametric modeling is very hard. It requires lots of experience and debugging skills. It is sometimes possible to chase down and debug the causes of problems with parameters. The result will sometimes make your model more robust to parameter value changes, but it is often very difficult, and it may not be worth the time required.

You can go into the Edit Parameter table and fix up all the "d" dimensions and model parameters that should be based on user parameters but aren't. This may fix a good percentage of your parametric errors. But it might not catch all of them.

There may be features in your model that are based on absolute positions. How does this happen? "Snap to Grid" and "Align" are handy tools that can result in absolute positions.

But there's value in having a standard reference for repeated measurements.

Remember that parameter values can use algebra and mathematical formulae.
So you can create a user parameter that simplifies a complicated formula down to a single term. e.g. overallLength = headLength+(4*bodyWidth)+basecapOverhang.

User Parameters cannot be created in Direct Modeling mode. But they stick around if you create them before you switch to Direct Modeling or Base Feature modeling. Changes to User Parameters are not reflected in the model in DM. They are applied immediately on input.

Note: you can't change the units of a User Parameter after you create it. But you can enter an expression for the value using other units explicitly. E.g. "width" is in mm units, but you can enter "12 in" and it will get translated to mm correctly.

- **Parametric vs Direct Modeling**

  Fusion 360 has two completely different "personalities" available. It would be easy and tempting to just pretend that this schism does not exist, but you would be missing a very powerful way of modeling with Fusion 360. Both modes are equally important and should be considered. You may find that you sometimes prefer the "other" way of modeling with Fusion 360.

  Parametric Modeling is one of the most powerful capabilities of Fusion 360. Parametric Modeling mode captures all relationships and interdependencies as you define features in your model. Features can be dynamically linked to the parameters that define them. All that information is captured in the Design History as presented by the Timeline.

  But Fusion 360 also has the ability to work without a Timeline recording every relationship and parameter in your Design -- It is called Direct Modeling mode.

  Direct Modeling mode can facilitate exploration by being more fluid and less constrained.

You can switch from Parametric Modeling to Direct Modeling at any time. But you will lose the entire Design History for the Design. You can also switch from Direct Modeling to Parametric Modeling at any time and Fusion will start to record Design History from that point onward.

There is also the option of creating a Base Feature (sometimes called a DM Feature) within Parametric Modeling mode. This allows you to model with the fluidity of Direct Modeling until you Finish Editing the Base Feature. (Base Feature mode is a lot like editing a Sketch or a T-Spline Form in Fusion where you must explicitly "finish" editing and you exit the mode.) (No history is recorded while editing a Base Feature, but the Design History for the rest of the Design is retained.)

There is a Fusion 360 Preference to specify which mode you want to use as the default when you start a new Design. Alternatively, you can choose for Fusion 360 to ask you every time you start a blank Design.

Parametric Modeling	**Direct Modeling (DM)**
Capture Design History (Timeline).	Do not capture Design History (No Timeline). (There is still undo/redo and versioning.)
Creates designs with an editable design history. Use parameters and relationships to capture your design intent.	Explore form and function with less regard for model features and relationships.
Delete is often unavailable or can cause unintended side effects due to relationships. (There will usually be a warning dialog.)	Delete is a powerful modeling tool.
Remove available as an alternative where deleting is dangerous.	Remove unavailable. Use Delete (or Dissolve for Features).
Features can be Suppressed.	Suppress unavailable.
User Parameters are available and editable. Connections between User Parameters and the objects they inform are live and update dynamically.	User parameters are not "available", they are still present (but hidden) if they were defined previously. They can be referenced in inputs and formulas. But connections to those references are not maintained. Parameter values can not be edited.

Component Drag is always enabled. Changes in Component position must then be confirmed with Capture Position or declined with Revert.	Component Drag is only available when Select->Component Drag is enabled or while the Alt key is pressed. Component position is always updated in DM mode. There is no Capture Position without the Timeline.
	Find Features available.
	Imported models show up as DM Features. Models from other programs open in DM mode.
Create new Base Feature will create a DM Feature without losing the Timeline.	
Timeline Features can be converted into DM Features.	
	Direct Modeling mode applies to the entire Design (not just the active Component).
	Import Component is not available.
	Insert into Current Design is not available.
	Make Independent is available.
Deleting the base object of a Pattern will delete the Pattern.	You can delete the base object of a Pattern and the Pattern will not disappear.

• **Hybrid Direct/Parametric modeling**

This hybrid method allows you to leverage the best qualities of both Parametric and Direct modeling. It allows you to use Direct Modeling without losing the Timeline.

- Start with the Timeline ON.
- Define your User Parameters.
- Create a Base Feature and start Direct Modeling within that mode on the Base Feature component. Finish editing the Base Feature. The rules and restrictions for Direct Modeling mode apply (mostly).
- If you want to make changes to the Base Feature select it in the Timeline and choose Edit Feature.

Remember that the Base Feature still exists within the Design History. Features and modifications that are created after the Base Feature are inaccessible to it.

(DXF4Laser does not work in Base Feature editing mode.)

- **Robust Parametric vs Efficiency and Velocity**

  Opinion: My experience has been that the benefits of truly parametric models are often overstated. When doing limited production, complexity is time, and with a limited production run you don't have the opportunity to leverage your design time investment across thousands of products.

  As models grow more complicated it becomes more and more difficult to build robustly reusable models with adjustable user parameters. I wish it were not so. But I find building a non-trivial model with configurable width or thickness is about twice the work.

  The effort of learning to model more efficiently and learning the details and quirks of the tools will reap more rewards than the effort of making models robustly parametric.

- **Name Everything ("Rule #2")**

  Try to name things as you go along. You can always change names as you go along. Adopt a schema for naming. But it's only as important as it is useful.

  I generally start with lowercase letters for the names of basic things like bodies, sketches, and construction objects.

  I tend to start with capital letters for the names of higher level objects such as organizational tools like Groups, Selection Sets and Components.

  I use "camelcase" naming where words are joined together without spaces and new words are capitalized. E.g. *doorLower*. Alternatively, you might use spaces, dashes or underscores to separate words.

  Fusion cuts off the ends of object names in the Browser tree if they get too long. So it's a good idea to have the most important information in the name first and avoid redundant information.

I generally name things from general to specific. That way objects organize themselves when sorted alphabetically.

- doorLower
- doorUpper
- panelBack
- panelFront
- panelLeft
- panelRight

I don't include the containing object name in the object name. For example, I might name a Body *panelLeft* in a Component called *LowerBox* rather than *lowerBoxPanelLeft*. This also makes it easier to move the bodies between Components without needing to rename all the Bodies.

I will sometimes use abbreviations in names, but be careful of ambiguity. Make sure you can figure it out from context. *panelL* might mean *panelLeft* or *panelLower*. *panelB* might mean *panelBack* or *panelBottom*.

- **"Rule #1" - make a new component and activated before starting a new part**

"Rule #1" says "always make a new component and activate it when starting a new part." Rule #1 is a workflow guideline that is promoted in the Fusion 360 design community.

The essence of "Rule #1" is that the workflow allows you to use Components to create parts that are complete sub-designs within a Design. Consequently, the workflow provides additional organization and structure to a Design, and that has some benefits.

In case you are wondering, these are the benefits of using Rule 1:

- Many objects created after activating the Component such as Sketches, Bodies, construction geometry, Joint Origins are created in that Component. (Selection Sets and Analyses are always created at the top-level of a Design.)
- When a Component is activated, the timeline is filtered to show only the events that pertain to that active component. This makes the timeline more manageable.
- The "Rule 1" workflow facilitates that the complete parametric design history for a component is contained in the component's design history. Thus, if such a component is exported with Save As, then the complete parametric design history for that component will be exported with it.
- Fusion 360 Joints in the Assemble menu only work with Components.

- Only Components show on a Drawing BOM.

• Ignore "Rule #1" for most laser cutting

But in the case of workflows on designs intended for laser cutting, individual parts need to be created and positioned easily. And the parts for laser cutting are often not isolated or created on their own. Laser cut parts are often split off of larger bodies. The additional structure imposed by Component containers doesn't add enough benefits in this application.

I recommend the use of Components for assemblies of laser cut parts that will move independently or that will be assembled to other parts in the final product using an external fastener like a bolt, screw or hinge. For example, a lid or a door.

• Press-Pull, Move Face and Extrude

Press-Pull (Modify) is a shortcut to several different tools depending on what is selected.

- Fillet -- when applied to an edge or edges or a point (from which it assumes an edge).
- Offset Face -- when applied to one or more faces. The selected face is offset tangentially and may redefine the original feature or offset the face. (The Automatic setting will try to choose for you.) In both cases edges are extended by continuing the edge -- Straight edges are continued straight; curves are continued along the curve. Make sure it is doing what you want.
    - Offset Face is now also accessible as its own command under Modify.

Move (Modify) - Face is a more general and customizable version of Press-Pull/Offset Face.

- Move Face - Translate: allows you to move a face in linear dimensions using Component or Design axes or a user defined axis.
- Move Face - Rotation: allows faces to be pivoted on any axis.
- Move Face - Free Move: allows you to select a pivot point about which any angular and linear translation is applied. (The axes are defined by the tangents of the first selection point, or they can be chosen using the Set Pivot option.)

Extrude (Create) on a planar face or co-planar faces or a closed path is different from Press-Pull/Offset Face and Move Face.
Extrude extends the body by moving the selected face perpendicularly to the selected face. By default edges do not grow from the starting face/profile's original outline, but a taper can be specified that grows or contracts edges at an angle from perpendicular. Extrude concludes with an explicit operation: Join, Cut, Intersect, New Body, New Component.

Beware: Join will sometimes suck connected bodies into the newly extended body. (i.e. the feature selects implicit targets.)

- **Import SVG**

  Historically, SVG import has been a fussy feature in Fusion, but Autodesk has improved it recently.

  When an SVG is imported all the points and lines are "Fixed" (green lines and dots) after you do the initial placement. If you want to edit the points or lines in the Sketch you must Unfix them (blue lines and dots). The easiest way is to drag select the entire imported drawing, right click elsewhere and choose "Fix/Unfix".

  You must Fix or Hide them again if you want them to not move. (They, like any other Unfixed line, can be moved even outside of Edit Sketch.)

  Control point mesh on SVG import gives you spline control points on the imported drawing where they can be interpolated.

- **Rectangular Pattern (Part I) - whacky axes!**

  Rectangular Pattern in Design modeling is a bit unpredictable!
  (Rectangular Pattern in Sketch is a little bit more predictable.)

  Rectangular Pattern creates a 2-dimensional array pattern of a selected object with a designated spacing.

  In addition to selecting an object to repeat throughout the pattern, you must select a single direction: typically an origin axis, an edge or a construction axis. (In Sketch Rectangular Pattern you may select one or two directions.)

The relationship between the single direction and the 2 axes of the matrix is not intuitive. The array constructed will include the direction selected and another axis. The second axis is usually orthogonal, but it is often difficult to predict.

Test to make sure you are getting your array in the desired axes.

If not, try choosing other edges such as the opposite edge to get different results. Sometimes you just can't get the pair of axes you want.

- **More Pattern oddities (Part II) - Extent Dimensions**

  When specifying a Rectangular Pattern or a Pattern on Path you must specify one or two dimensions.

  Spacing dimensions specify the dimension of the repeating interval of the selected object+gap.

  Extent dimensions specify the overall distance of quantity n-1 (objects+gaps).

  As a result, Extent dimensions may not work the way you expect.

  For example, an Extent dimension of 100mm for a quantity 3 will result in:
  Thing1-gap-Thing2-gap spaced across the 100mm and then a Thing3 starting at 100mm hanging off the end!

  You can, however, easily suppress the last object with the Suppress checkboxes.

- **Symmetric Patterns (Part III) - the secret box joint tab tool**

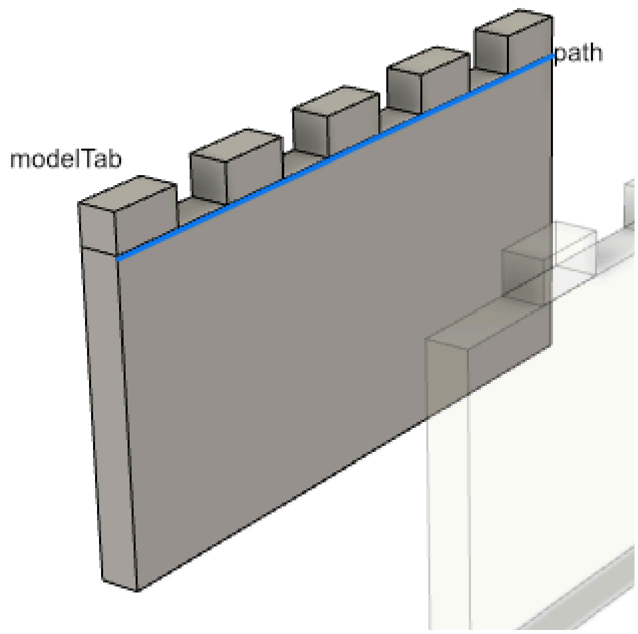
  You will often want to create symmetric patterns in the form:
  Object-gap-Object-gap-Object with a specific N number of Objects filling the extent of a path. That is, your goal is to place Objects at both ends of a path while equally spacing the remaining N-2 Objects in between the end Objects. This is a typical pattern used when creating box joint tabs, floating tenons, finger joints, bolt holes, fork tines, shelves etc.

  Likewise, you may want to make symmetric patterns in the form:
  gap-Object-gap-Object-gap-Object-gap.

Unfortunately, the parameters for the Pattern-on-Path feature don't make it obvious how to create such patterns because of the odd Pattern Extent Dimension behavior (discussed in More Pattern Oddities, above). But there are simple recipes to get the Patterns to come out exactly as desired.

Finally, I provide a recipe to control positioning of asymmetric patterns of the form: *Object-gap-Object-gap-Object-gap*.

(These recipes assume you are creating box joint tabs, but they can be used for creating symmetric and asymmetric patterns of any object.)

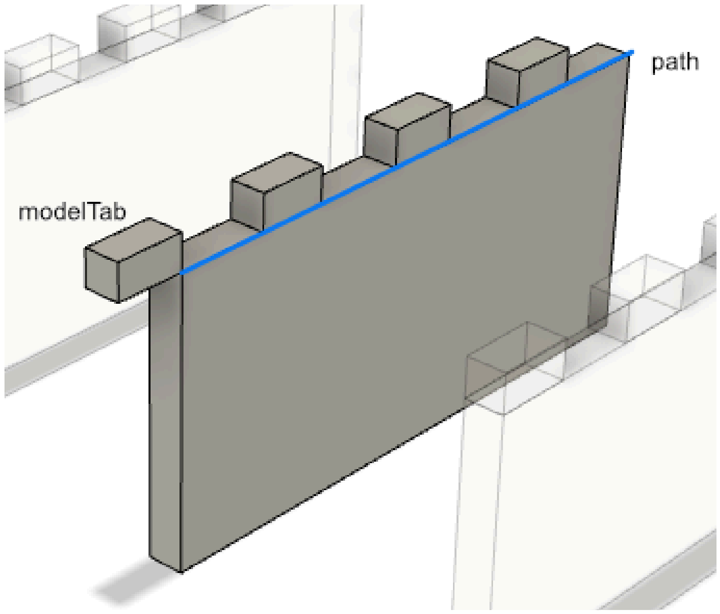


Recipe 1:
To create equally spaced tabs in a **symmetric** pattern: **Tab-gap-Tab-gap-Tab**

- Choose a path along which you want to evenly place the tabs (e.g. the edge of a face panel).
- Create a model tab Body of tabWidth at the beginning of the path.
- Use the Pattern-on-Path tool and select the model tab Body as the Object.
- Make sure Distance Type is set to Extent.
- Select the edge along which you want to place the tabs as the Path.
- Set the Start Point to 0.
- Drag the Distance to the length of the edge. And then subtract 1 tabWidth from the Distance. That is, use (edge length-tabWidth) for your Distance.

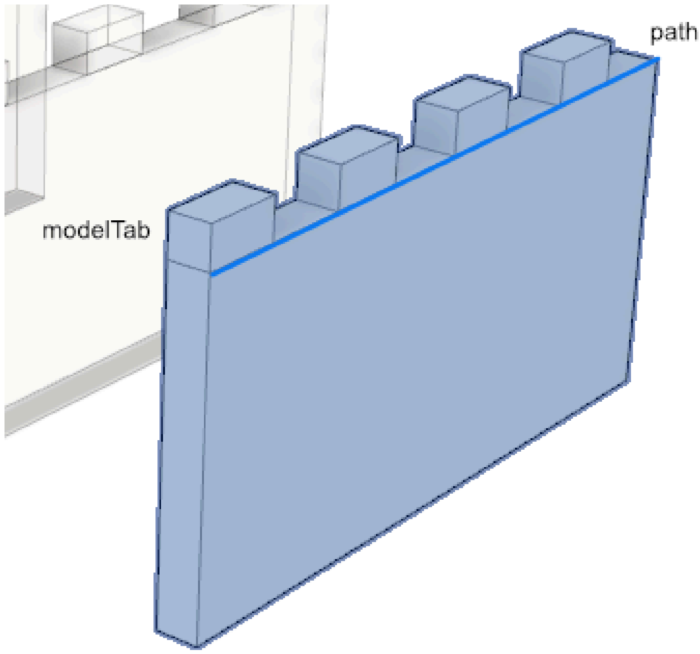
You can interactively adjust the number of tabs by adjusting Quantity.

Gap size is always evenly divided.



Recipe 2:
To create equally spaced tabs in a **symmetric** pattern: **gap-Tap-gap-Tab-gap**. ie. gaps on the ends.

- Choose a path along which you want to evenly place the tabs (e.g. the edge of a face panel).
- Create a model tab Body of tabWidth, but **position the model tab before the beginning of the path.** That is, make it so that the model tab ends at the beginning of the path. The model tab should hang over the edge of the face.
- Use the Pattern-on-Path tool and select the model tab Body as the Object.
- Make sure Distance Type is set to Extent.
- Select the edge along which you want to place the tabs as the Path.
- Set the Start Point to 0.
- Drag the Distance to the length of the edge. And then subtract 1 tabWidth from the Distance. That is, use (edge length - tabWidth) for your Distance.
  You will end up with **1 less** tab than the Quantity specified because you must also...
- Remove the model tab in History mode or Delete the model tab in Direct Modeling mode. (If you Delete the model tab in History mode the Pattern will get deleted too.)



Recipe 3:

To create equally spaced, tabs in an **asymmetric** pattern: **Tab-gap-Tab-gap**

- Choose a path along which you want to evenly place the tabs (e.g. the edge of a face panel).
- Create a model tab Body of tabWidth at the beginning of the path.
- Use the Pattern-on-Path tool and select the model tab Body as the Object.
- Make sure Distance Type is set to Extent.
- Select the edge along which you want to place the tabs as the Path.
- Set the Start Point to 0.
- Drag the Distance to the length of the edge.
- You will end up with **1 less** tab than the Quantity specified because you must also...
- Suppress the last tab. (Check Suppress in the Pattern-on-Path dialog and uncheck the checkbox on the ghost image of the last tab before pressing OK.)

If you forget to suppress the last tab in Recipe 2 or 3 when in History mode, you can Edit the Feature in the timeline or just Remove the extra tab hanging off the end. If you are in Direct Modeling mode you can just Delete the extra tab object.

- **Pattern-on-Path Start Point**

Pattern-on-Path Start Point is a decimal fraction that is subtracted from Distance. e.g. if Distance is 100 mm and Starting Point is 0.1 then distance becomes 0.9 * 100 mm = 90mm. If Starting Point is 0.9 then Distance becomes 0.1 * 100 mm = 10 mm.

(Pattern-on-Path has been changed recently to set the Start Point to half the width of the pattern Object by default. I don't know the rationale for this, and I just set it to 0 to get the recipes above to work.)

- **Organization Tools**

  The organization tools in Fusion 360 are not all obvious, but they are there.

  Groups
  There are Groups for many kinds of items. But most importantly Groups of Bodies.

  In the Browser schematic view, Right Click the main Bodies folder for your Component or the top-level, main Component; you will see New Group.

  A Group is just a folder for organizing Bodies within a Component's main Bodies folder or another Group folder. You can select multiple things and just drag them into the Group folder in the Browser, or select a Body or many Bodies and right click; Choose "Move to Group". (An awkward thing about the Move to Group command is that it requests a Target Group, but there is no provision to create a new Group if an appropriate one doesn't already exist. So you need to make sure you create your new Group first.)

  Selection Sets
  An alternative for making collections of things is Selection Sets. Selection Sets exist in parallel with Groups and other Design organization tools.

  Multi-select something (like a number of Bodies); right click and choose Create Selection Set. There's now a new top-level folder in the Browser called Selection Sets. Inside you will find your selection set. You can rename it to something meaningful. When you click on a Selection Set in the Browser tree two buttons appear next to it: Select and Update. Update will update the Selection Set with the currently selected things.

  Selection Sets can include things from different Components whereas Groups can not.

  Selection Sets can also be non-heterogeneous. I.e. the things in a Selection Set don't have to be of the same kind.

  Components as groups
  Components also behave as groups. Components are containers first and foremost.

You can also have a group of Components called an Assembly. Complex designs often end up with a tree hierarchy of Components within Assemblies within Assemblies.

Bodies and Components

Bodies - Bodies are just shapes

- Bodies are more fundamental than Components.
- Bodies only exist within a Component.
- Bodies just contain shapes.
- There are three different kinds of Bodies in Fusion 360.
    - Solids and Surfaces ("B-Reps") - these are the core modeling objects in Fusion 360.
    - Forms (T-Splines) - for organic shapes.
    - Meshes (of triangles or quadrilaterals) - typically imported STL/OBJ data.
        - The Mesh modeling tools in Fusion 360 are limited.
    - The different kinds of Bodies can't be mixed within a single Body. Conversions are available.
- Bodies always have a position relative to their containing Component. But Bodies do not have an Origin or Grounding of their own.
- Bodies are constrained by default and must be explicitly moved.
- Copies of Bodies are not linked instances of the original Body. Each copy is unique. (Patterns and Mirrors of Bodies are different from copies.)
- You can not create a Joint between Bodies.
- You can not create a Bill-of-Materials for a Drawing with only Bodies.
- Bodies can be Isolated/Unisolated in the workspace only if there is no Assembly in the Design. (i.e if the Design consists only of one root Component and its Bodies.)

Components - Components are for structural organization

- Components are containers.
- Each Component can contain groups of Sketches, Bodies, and Construction geometry.
- Components
    - Group objects and control their scope.
    - Connect objects and define their relative position and motion.
- Components can contain one or more Components, in which case they are Assemblies. Assemblies have a unique icon (side-by-side boxes) in the Browser. (Sometimes the term "sub-Assembly" is used to refer to an Assembly within an Assembly.)

- Assemblies (groups of Components) behave as Components. Pretty much anything you can do with a Component can be done with an Assembly.
- A Design always contains a Component or an Assembly (a group of Components) at its root.
- Components and Assemblies can be Activated in the Browser via the radio button on the right side of the item. All activity is then targeted to the current Active Component/Assembly.
  - New Bodies, Sketches, Construction geometry are created in the Active Component.
  - The Timeline is filtered to the currently Active Component/Assembly, and new events are added to the Component's Timeline.
  - There is only ever one Active Component/Assembly in a Design.
- Drawing Bill-of-Materials (BOMs) are made up of Components.
- Components help organize and layout parts for Manufacture/CAM (see **Tip: Use a second Assembly of parts when creating cutting layouts**).
- Components are unconstrained by default. You can select and drag a Component.
- Any change to the position of a Component must be captured. If you move a Component you must capture the new position if you want it retained in the Design (sometimes called "Taking a snapshot"). (Capture Position is an event on the Timeline.)
- Components can be assembled to other Components using Fusion 360's Joint system. (See **Fusion 360 Joint Fun** below.) The Joints themselves live in the highest shared level of the Component/Assembly hierarchy in common between the Jointed Components.
- A Copy of a Component is by default a linked instance of the original Component. Instanced Components are designated in the Browser with a shared name and colon nomenclature.
  - For example "Component1:1" and "Component1:2" are two instances of the same Component1.
  - Changes to one instance will always be reflected in the other.
  - Instances don't have to be contained in the same parent Assembly.
  - To make a unique unlinked copy of a Component you must use "Paste New". The result will have a unique name and ":1" Changes to the copy will not be reflected in the original copy.
  - The position of an instance of a Component is not shared between the copies. (This is because the position is defined relative to the Origin of the containing Assembly.)
  - To unlink an instance of a Component use Cut then Paste New.
  - A Mirror of a Component is not a Copy. The mirrored Component is unlinked from its source. (Go figure.)
  - Components created from Patterns are Copies and are linked to their source.

- Components can be Isolated/Unisolated.
- Component Origins and Grounding (Origin Locking)
    - Each Component or Assembly has an Origin. The Origin defines a Component's position and orientation relative to its parent Component.
    - Only Components/Assemblies can be Grounded. Grounding is essentially locking a Component's Origin and all up-stream Origins.
    - Any Grounded Component or sub-Assembly within an Assembly effectively grounds its parent Assemblies all the way to the root. But a Grounded Component or Assembly doesn't ground other sub-Components of the same parent Assembly. Grounding an Assembly doesn't Ground its sub-Components or sub-Assemblies.
    - The root Component/Assembly is always Grounded.
    - Grounded Components don't show Move as an option in the context (rt.-click) menu and they can't be selected for Component Move operations.
    - Grounding is not copied with a Component
    - Consider using Rigid Joints when locally relative positioning must be maintained in a sub-Assembly that will be duplicated.
- Components can be colored to differentiate them in the Workspace. Use Inspect->Component Color Cycling Toggle (shift-N). Individual Component colors can be changed using the Cycle Component Color context (rt.-click) menu on the Component. Note: color coding is applied to the Component, the Browser and the Timeline.

Derive, Insert, Save Component As, Associative Edit in Place, Assembly Contexts

tbd

- **MacOS Selection Bug**

When Fusion 360 won't allow you to select faces or bodies. This typically occurs after you have just switched into Fusion from another app like Chrome.

Solution: switch away from Fusion to another app. Then switch back in.

Selection details

Fusion 360 has a range of tools for object selection. Take some time to learn the details.

Select is always available. The Esc key will almost always return you to Select mode.

In all selection modes holding down Command (Mac)/Ctrl (Win) will add to the current selection.

• **Rectangular and Freeform Selections - window selection vs crossing selection gestures**
In rectangular "Window Selection" mode there are two gestures:
• drag from **left-to-right** and only objects entirely within the selection rectangle will be selected. (The selection rectangle will have a solid purple border line and the area will be shaded orange.)
• drag from **right-to-left** and all objects that are either within **or** crossed by the rectangle will be selected. (The selection rectangle will have a dashed blue border line and the area will be shaded yellow.)

Likewise, Freeform Selection mode has the same two gestures:
• lasso drag the mouse in a **clockwise** direction around the workspace and only objects entirely within the border line will be selected. (Again, the selection area is orange and the selection border line is solid purple.)
• lasso drag the mouse in a **counter-clockwise** direction around the workspace and all objects that are within or crossed by the selection border line will be selected. (Again, the selection area is yellow and the selection border line is dashed blue.)

The selection area color may be different if you are using an Environment Display Setting (in the Navigation Bar) other than Photo Booth.

• **Paint Selection**
In Paint Selection mode the mouse acts as a brush. Each new click and drag starts the paint selection over. Click in an empty space to deselect all.

• **Selection Filters and Priority**
The Selection Priority commands are speed commands for the Selection Filters. Note: they are modal as indicated by a tiny check on the icon. Select them once to turn them on. Select the same command again turn it off.

Selection Filters determine what types of things can be selected and are mostly self-explanatory. Select All is a speed command to select all or none of the object types in the list.

• **Select Through**
But don't overlook Select Through at the top of the list of Selection Filters which determines whether you may pick objects which are obstructed by other objects, or, just as importantly, to **not** select obstructed objects.

• **Selection Tools = Find Commands**
Selection Tools are a set of object search tools to constrain/filter the selection set. Each has its own specialization - name, location, size. Remember these are here.

• **Create Selection Sets to save a selection**
Selection Sets are a way of saving your current selection of objects. You will find "Create Selection Set" in the Rt. Click menu. Once you have created a Selection Set you will find it at the top of the Browser where Selection Sets can be named, updated or selected.

One nice aspect of Selection Sets is that you can use them to create groupings across objects. E.g. you could define a Selection Set of just the inside faces of all the objects.

• **Selectable/Unselectable toggle in Browser**
Each Body or Component object in the Browser has a Selectable toggle available via the Rt. Click menu. When an object is Unselectable it will get a small "no" symbol on its icon in the Browser. Unselectable objects can still be selected via the Browser or Selection Tools.

• **Click-hold to select obscured object**
Click-hold will present a list of selectable objects under the mouse pointer. As you go through the list each object will be highlighted. You can follow the list to select Parent objects too.

Tip: As you go through the list object faces facing the camera appear light (as if illuminated by the view camera). Object faces facing away from the camera appear darker.

• **Box joints (tabs)**

Box joints are the most common type of construction joint used with Laser Cutting. (They are often called finger joints.)

- Create user parameters for tabWidth and tabGap and materialThickness as needed.
- Make the tab bodies using one of the methods below.
- Modify->Combine-Join the new tab Bodies to the target panel Body (do not "keep tools").
- Modify->Combine-Cut the tabbed Bodies with the complementary target Bodies as tools (keep tools).

- Check for interference (Inspect->Interference).

Method: Extrude from an on-Surface Sketch.
- If necessary create a Sketch on a profile face (perpendicular to tab projection) or Surface of the Body on which the tabs will be added.
- Draw the tabs on the Sketch.
    - Use constraint, dimension, pattern, mirror as needed.
    - Stop Sketch
- Create->Extrude the tabs from the Sketch to create the tabs as new Bodies.
    - It may be necessary to select the Sketch in order to select the tab rects as profiles.

Method: Model a single starter tab with a box and use Pattern-on-Path and/or Mirror to replicate. (see **Symmetric Patterns (Part III) - the secret box joint tab tool**)

• **Using "Measure" for dialog input**

Learn to use Measure in dialog input. (Different from Inspect->Measure). Measure allows you to pull input dimensions directly off of a model.

Get good at using Measure to input measurements.

Sometimes you can just click to select something which has a size. But sometimes you need to click once on the workspace to get Measure to start measuring. You can select planes, points, faces, edges, snaps.

Sometimes you need to set the input field to 0 before you choose measure or the measure will be added to the pre-existing value.

• **Interference Checking**

Interference testing is an invaluable tool to check that your model doesn't have any internal collisions or overlaps that will prevent it from being assembled or functioning properly (especially for models that will be assembled using box joints or mortice and tenon joints).

Select the bodies and/or components that are to be assembled. And choose Interference under the Inspect menu. Click on the Compute button. There will either be a dialog indicating there is no interference, or there will be a table identifying the sizes of all the areas of overlap in the bodies with color coding indicated in the design. In the latter case, you must figure out what you missed or what's going wrong to cause the interference.

Caution: Under some circumstances Interference checking will present cuts to eliminate the interference. This is subtly indicated by the columns labeled Target and Tool in the table. Clicking OK will cause cuts in any checked rows to occur. Click Cancel to avoid the cuts.

- **Section Analysis**

  Inspect -> Section Analysis allows you to see a dynamic cross-section of bodies. If you OK the Section Analysis dialog, the visibility of your design can be a bit confusing. It's important to realize Section Analysis doesn't modify your geometry even though it looks like it has sliced away parts of it.

  If you OK the Section Analysis dialog there will be a new Analysis folder created (as needed) at the top level of the Browser schematic view. From there you can hide, edit or delete the Section.

- **Minimum Radius Analysis**

  Inspect -> Minimum Radius Analysis can be used to check that a body can be cut with a specific CNC router bit. You can enter the radius of your finish cutting mill or bit. (Note: enter the bit radius, not diameter.)

  You can also use this analysis tool to determine the maximum sized CNC router bit that can be used to cut a body. For example, after a body has been scaled or modified.

  If you OK the Minimum Radius Analysis dialog there will be a new Analysis folder created (as needed) at the top level of the Browser schematic view. From there you can hide, edit or delete the analysis.

- **Outputting Cut Files -- Export DXF**

  Primary Method - Create Sketch/Save as DXF

  - Select a body Face to be output and cut
  - Right Click: Create Sketch
  - Name the Sketch
  - Click: Stop Sketch
  - Right Click: Save Sketch as DXF
  - Name the file appropriately

Alternative Method - use DXF4Laser

- Install and Run DXF4Laser add-in script. Check "Run on Startup".
- Select a body Face to be output and cut.
- Choose: Export DXF for Laser Cutting from the ~~Sketch~~ Modify menu. (See note about 2019 UI update below.)

DXF4Laser (aka DXF for Laser) is available free on the Autodesk Fusion 360 app store. https://apps.autodesk.com/FUSION/en/Detail/Index?id=7634902334100976871&os=Mac&appLang=en

Sometimes DXF4Laser will fail and not create the file. Sometimes it will create a warning message in the temporary sketches about the manifold nature of the face. It will usually leave a folder full of temporary things at the bottom of the Browser. You can usually just delete that folder. But you will have to work around the failure usually by falling back to the Create Sketch/Save as DXF method.

Warning: DXF4Laser does not work in Base Feature editing mode. It may crash Fusion 360! (It does work in DM Mode though.)

- **Embellishing your parts with ornamental design**

An easy way to embellish your projects with ornamental design is to use a workflow with a 2D drawing program. (Fusion 360's Sketch module is nowhere near as flexible as Illustrator, Inkscape, CorelDraw or Affinity Designer.)

If your 2D drawing program of choice has the ability to handle DXF files robustly then just export your parts for cutting from Fusion 360 as you normally would. Then import and modify them in the 2D drawing program.

I like Affinity Designer as a cheaper and very capable alternative to Illustrator, but it doesn't support DXF which is inconvenient for laser cutter use. It supports SVG and PDF, but be very careful with units and scale when using SVG in particular.
(In Affinity Designer I've had success exchanging SVG files with Fusion 360 and Lightburn by setting the Document Setup DPI and SVG Export DPI to 96 dpi.)

Note also that Lightburn laser control software supports DXF, SVG and PDF formats.

Fusion has built-in support for exporting DXF format versions of body faces, but Fusion doesn't have built-in SVG export. The Shaper Utilities plug-in (by Shaper Tools) available free on the Fusion 360 App Store (see below) can export SVG versions of Body Faces and Sketches.

This external 2D drawing program workflow is more flexible too. You can create "blanks" from a Fusion design then customize the design in the drawing program. Consider a votive candle box. A blank one could be made in Fusion, then you could create all kinds of variations with different ornamental designs in Illustrator.

2D drawing programs are also better for creating detailed designs for laser engraving.

Either way, I don't much recommend trying to do artistic drawing tasks in Fusion's Sketch module. Do 2D drawing and editing in Illustrator, Inkscape, Affinity Designer or your preferred 2D drawing program. Then you can import and position the drawings in Fusion and extrude the completed designs on your model, or add the drawings to cut files after exporting them from Fusion.

For a CNC router it would all depend on your CAM workflow whether external embellishment is feasible. Post export embellishment would work for the Shaper Origin which can input SVGs directly. Or if you were using VCarve Pro for CAM you can import 2D drawing files. But if you are using Fusion 360 for Manufacture/CAM you would obviously have to import the designs into Fusion to output them as G-Code (or whatever data output Fusion 360 Manufacture is configured to produce in post-processing).

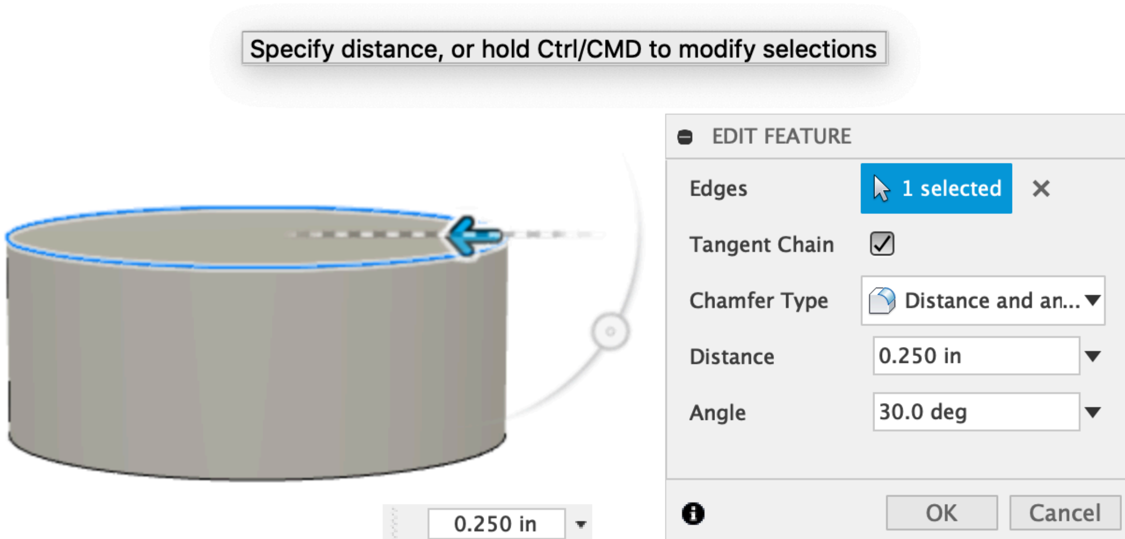
- **Chamfer - working around the shortcomings of the Chamfer feature**

  This workaround is limited to 90-degree edges.

  Picture your desired chamfer being created by a new face defined by the hypotenuse of a right triangle with the other two triangle legs defined by the insets into the two adjacent faces of the selected edge.

Fusion's 3 chamfer tool options have some limitations in how you can specify the dimensions of that triangle:

- Equal Distance - in spite of how it is depicted on screen, both legs of the triangle are defined by insets of the same distance parameter along the two adjacent surfaces of the edge.
- Two Distances - this is the only completely flexible chamfer. You specify distances for both the legs of the triangle. Using trigonometry we can achieve any chamfer we need with this option.
- Distance/Angle - Fusion chooses a face adjacent to the selected edge and creates the chamfer by applying the specified distance and angle into that face. But there is no option to direct Fusion to apply the distance and angle into the other face.



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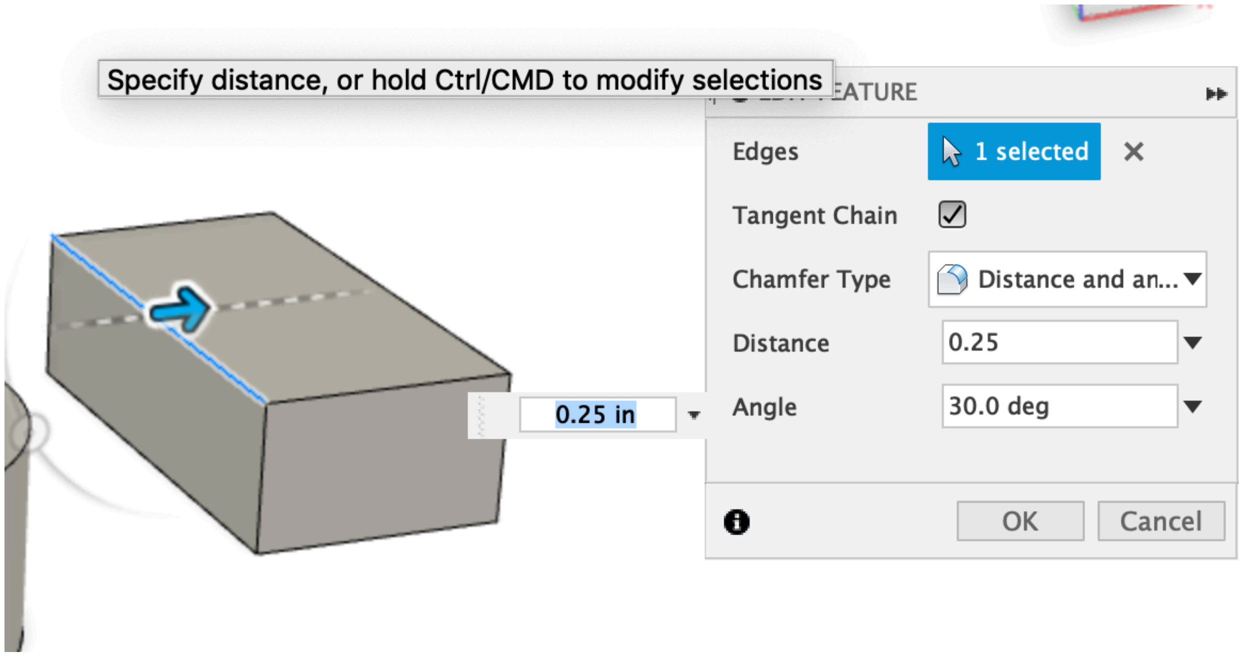
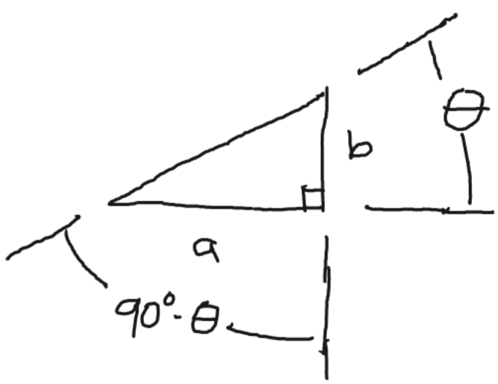


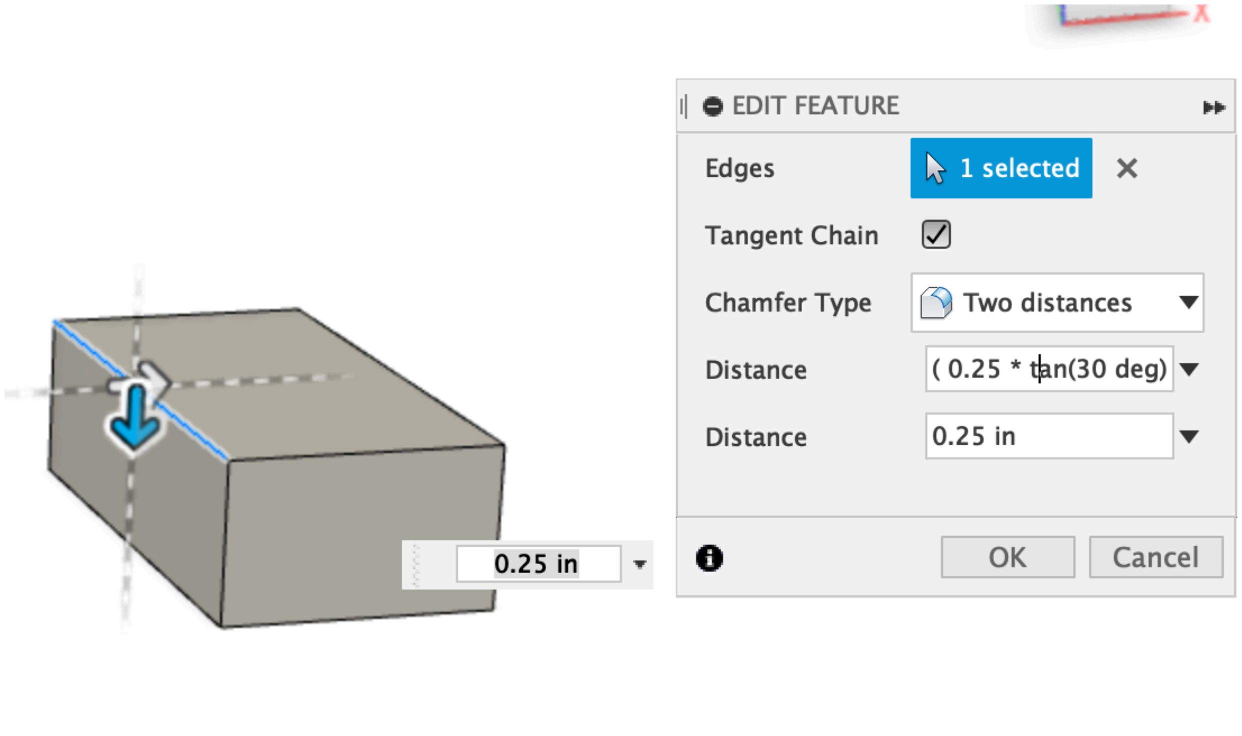
(In the above examples Fusion has chosen the top face and the angle formed into its plane.)

Solution One: Use the more flexible Two Distances type instead of the Distance/Angle type. In order to use the Two Distances option, you must convert from a Distance and Angle chamfer with the "wrong" distance to a Two Distances chamfer using the following formula.

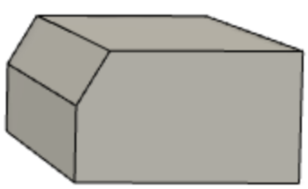


Given angle θ and an adjacent distance *a*, the other distance *b* can be calculated with the tangent formula.

*b = a * tan(θ)*



So just enter "***a * tan(theta)***" and "***a***" for the two distances.
(***a*** is 0.25 in, and ***theta*** is 30 degrees in the above example.)



Solution Two: Use the Distance/Angle type and convert your angle θto the opposite angle by subtracting it from 90-degrees and convert your adjacent distance *a* to the opposite side distance *b* using the formula *b = a * tan(θ)*.
Enter Distance: ***a * tan(theta)***; Enter Angle: ***90 deg - theta***.

Sketch

- **Sketch is a Timeline Feature**

  When you create a new Sketch it is created at the current position in the Timeline. The consequence is that if you subsequently Edit Sketch, it rolls back the Timeline to the point when the Sketch was created and nothing that was created or modeled after the Sketch was created can be referenced.

- **Visible Sketches are editable in the workspace**

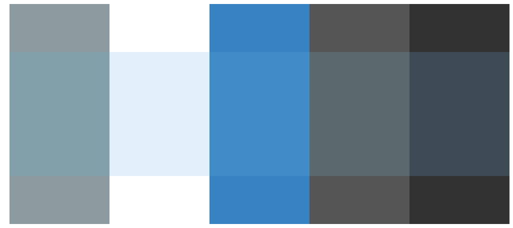
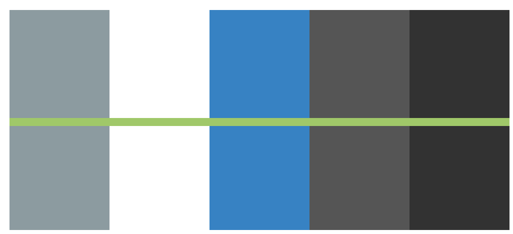
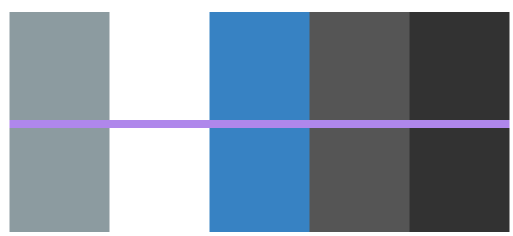
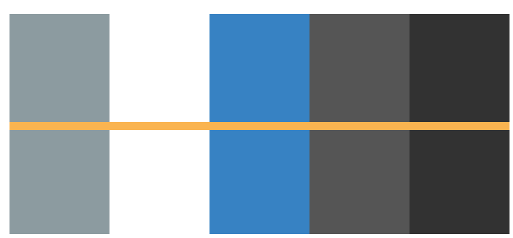
  A visible Sketch and its elements can be modified with Move and Delete at any subsequent point in the Timeline without going into Sketch Editing mode. This is a way to modify a Sketch to reference features that didn't exist when the Sketch was first created.

- **Sketch module line colors**

  Lines and Profiles in Sketch are colored based on state.

  Different types of lines are shown below in the various work environments. (The only one that has different colors in different environments is Fully Constrained which is black in the default Photo Booth and Infinity Pool and white in the other dark color environments.)

	Light background environment	Dark background environment
Profiles	Light blue fill	Light blue fill
Under constrained	Light blue line	Light blue line
Fully constrained	Black line	White line
Fixed	Olive green line	Olive green line
Projected	Violet line	Violet line
Cached projected	Gold line	Gold line

Profiles	
Under Constrained	
Fully Constrained	
Fixed	
Projected	
Cached Projected	

- **3D Sketch**

  3D Sketching is finally usable and useful.

  Why use 3D Sketch?
  Sketch in 2D provides an accessible approach for people experienced with drafting and on-paper sketching to start designs in Fusion.

  3D Sketch provides a way to extend 2D drawing into 3D space. It adds a 3D Manipulator tool to quickly and easily change the current Sketch plane. So you will likely continue to do drawing in 2D with 2D geometry, but you can do that on different planes within a single sketch. If you restrict yourself to orthogonal planes the techniques are easily manageable and need not be any more complicated than managing multiple Sketches with standard projections. An obvious example of the utility of 3D Sketch is that a single Sketch can now contain a plan and one or two elevations, and those sketch planes are positioned in the Design exactly where they will be most useful.

  Of course, you can also use 3D Sketch outside of the constraints of orthogonal planes and even 2D geometry, but that quickly gets a lot more complicated.

  How to use 3D Sketch
  Turn on 3D Sketch from the Sketch Palette inside the Sketch editing module. (Once you get comfortable with 3D Sketch, you might just leave this setting checked most of the time. It is a sticky setting that will remember how you last set it. When you need to constrain your Sketch to a 2D plane turn off 3D Sketch in the Sketch Palette. When you might be making multiplane sketches turn it on.)

  Make sure 3D Sketch is checked. Then if you orbit the view away from the 2D "look at" Sketch Plane view you will see 3D manipulators when creating Lines, Rectangles, Points, Splines, Circles, Conic Curves, Ellipses, and Polygons.

  There are additional modifier keys for the 3D Sketch Manipulators (some of them are a little tricky to master):

  Up-Arrow = tap to toggle locking the mouse cursor to the axis normal to the current Sketch Plane.
  Alt/Option = tap to cycle through the available canonical sketch planes.
  Tab = tap to cycle through the rotation handles.
  Command (Mac)/Ctrl (Win) = disable snapping while pressed.

Changing the Sketch Plane in 3D
An easy way to use 3D Sketch productively is to just do 2D drawing, but use 3D Sketch to quickly and easily change the current Sketch plane. The 3D Manipulator just changes the current Sketch Plane. As you hover over the central area of the manipulator you will see blue rectangles for the local canonical planes (XY, YZ, XZ). Click one of these rectangles to change the current Sketch Plane to that plane. (Also the Alt/Option key cycles through the 3 canonical planes.)

The angle handles allow you to change the current plane's angle and orientation beyond just the 90-degree choices of the canonical planes.

Constraints in 3D
Also note that Sketch tools like constraints work in 3D.

Add-in Utilities

- **Shaper Utilities for Fusion 360 add-in**

  https://apps.autodesk.com/FUSION/en/Detail/Index?id=3662665235866169729

  The Shaper Origin is a hand-held CNC router. It uses special adhesive tape to put optical registration marks on the uncut surface of your sheet material. Then a camera in the Shaper Origin tracks the registration marks and precisely guides the cutter as you push the router roughly around the surface.
  This add-in generates and exports cutting files from Fusion 360 body faces in the SVG format for subsequent input into the Shaper Origin.
  You don't have to have a Shaper Origin to use this add-in. This add-in can also be used to generate SVG files for use with other cutting systems or software tools that work with SVG since Fusion 360 doesn't have a built-in SVG export feature.

- ***Slicer for Fusion 360***

  https://knowledge.autodesk.com/support/fusion-360/downloads/caas/downloads/content/slicer-for-fusion-360.html

  Slicer is a fun tool with many features applicable for laser cutting and CNC routers. Primarily, it creates a set of stacked layers that can be cut and assembled into 3D shapes.
  (*Slicer for Fusion 360* was previously called *1-2-3D Make*.)
  (S*licer for Fusion 360* is not the same as the *Slic3r* FFF/FDM 3D printing utility.)

Slicer for Fusion 360 consists of an add-in for Fusion 360 that hands off STL model data to an application that runs outside of Fusion 360. The application doesn't require Fusion 360 to run.

[AUTODESK VIDEO: Slicer for Fusion 360]

Slicer for Fusion 360 is no longer being maintained or supported by Autodesk, but the final build for Mac and Windows is available at the link above.

- **DXF Import Utility add-in**

  https://apps.autodesk.com/FUSION/en/Detail/Index?id=3146198746757677787

  A utility to import multiple DXF files. It has features for automatic grid layout and auto-extrusion from the DXF profiles.

  This utility can be used to import DXF files in preparation for layout with the Arrange tool or Nester add-in (see below).

  This add-in tool is also handy for importing the output from *Slicer for Fusion 360* (see above) back into Fusion 360.

- **Parameter I/O**

  Import and export all the parameters in a design as a CSV file. This is useful when inserting the Components from one Design into another Design (especially now with Edit in Place).
  https://apps.autodesk.com/FUSION/en/Detail/Index?id=1801418194626000805

CNC Routing

- **Tip: Use a second unique Assembly of parts when creating cutting layouts**

  A good strategy when laying out Components for cutting within Fusion 360 is to have two sets of instances of your Component parts. That way you can model to a final assembly positioning of all the parts. One set of Component parts can remain in their final or working assembly (e.g. an assembled chair). But a second Assembly of Component parts can be arranged for cutting layout (such as on a sheet of plywood stock) without affecting the final assembly positioning. Model changes to the individual

Components will be updated in both instances, but positional changes and Joints can be independent between the two Assemblies.

How to:
It won't work to put all the Component parts inside an Assembly and copy the Assembly because the two Assemblies will then share the same Joints. And Copy-Paste New will create a unique new Assembly **and** unique Component parts.

Instead, you must Select all the individual Components parts (Selection Sets work well for this), and copy them all. Then Paste them (not Paste New) into a new Component (which then becomes an Assembly). That way you will end up with two unique Assemblies that can have different Joints between the same set of Component parts.

Nesting

Nesting is the process of arranging 2D parts onto sheets of material for manufacturing either using Fusion's Manufacturing/CAM module or for export as flat DXFs to other manufacturing workflows such as for laser cutters or CNC routers using a 2D cutting utility like *LaserCut*, *Lightburn* or *Vectric VCarve*.

Three major options for nesting in Fusion 360

Arrange feature	Built-in	Not available in Personal Use license
Advanced Arrange features	Pay-as-you-go Extension	Not available in Personal Use license
Nester	Free add-in	Available to all users
Nesting & Fabrication Extension	Pay-as-you-go Extension	Not available to Personal Use license users

Arrange and Nester

- **Arrange**

  (Modify - Arrange) (not available in the Fusion 360 Personal Use subscription)
- ***Nester***

  free add-in https://github.com/tapnair/NESTER

Fusion has a built-in nesting tool called Arrange accessible in the Design and Manufacture workspaces. (The Arrange tool must be enabled in Preferences > General > Design.)
The Arrange tool evolved out of the free Nester add-in.

With either the Arrange tool or the Nester add-in, you create on-the-spot arrangements of multiple selected Components onto planes, sketches, or planar faces with uniform spacing between the bounding box of each Component. (Arrange and Nester don't work on Bodies.)

Arrange also has advanced options called "Advanced Arrange" that are currently in Preview, but eventually the options will be part of the pay-as-you-go *Nesting & Fabrication Extension* (see below).

Alas, the Arrange tool is not available in the Personal Use subscription, but Nester is available to all Fusion 360 users.

You can then use the new nested layout to create cut operations for a CNC router or other 2D cutter in Fusion 360's Manufacture (CAM) module. Or you can export the cut layout as a flat DXF to a 2D cutting utility (e.g. *LaserCut*, *VCarve*, *Lightburn*).

Nester uses Fusion 360 Planar Joints to create the layout positioning. You should be comfortable with Fusion 360 Joints to use the Nester add-in tool effectively.

It's a good idea to apply Arrange or Nester to a new unique Assembly of Component parts as described in the Tip above.

[Autodesk tutorial: Arrange in Fusion 360]()

[Nester tutorial]()

- **Nesting & Fabrication Extension**

  In addition to Arrange and *Nester*, Autodesk also has a new, sophisticated Nesting feature in the "Nesting & Fabrication" optional Extension. The focus of the Nesting & Fabrication toolset is setup efficiency and efficient use of material.

  "Nesting & Fabrication" is a pay-as-you-go Extension feature. (It costs $25 in Credits for 1 day of use or $1600/year (or $200/month) for a continuous subscription. Serious stuff.)

The Nesting & Fabrication Extension will eventually also include the Advanced Arrange options for rotation constraints and target sheet selection. Currently Advanced Arrange is in Preview and available as part of the Arrange command (see above).

https://www.autodesk.com/products/fusion-360/nesting-fabrication-extension

> Autodesk describes the feature:
> *If you're cutting a large number of parts out of flat sheet stock with varying dimensions and prices, Nesting is a crucial piece of the workflow to help eliminate guesswork and minimize waste. Fusion Nesting includes integrated, automated, and associative nesting capabilities to optimize the arrangement of flat parts, both sheet metal and non-sheet metal, on flat raw material in preparation for cutting operations. It allows you to generate nests from your Fusion 360 designs and create toolpaths and NC code to drive your CNC machines*

- **Dogbone generator add-ins for CNC routing:**

  These add-ins for Fusion360 create fillets on inside corners for CNC routing. Trying to create dogbones without one of these add-ins is a lot of fussy work.

  Dogbone v2 in static mode (non-parametric) is the most robust of these tools. Nifty Dogbone and Dogbone v2 (in parametric mode) seem to have trouble dealing with a mixture of dogbone types within a single Design or part.

  Warning: Neither of these add-ins seems to work in Direct Modeling mode or Base Feature modeling.

  - **Dogbone v2**

    https://github.com/DVE2000/Dogbone
    This free add-in for Fusion360 creates fillets on inside corners for CNC routing. Choose a face and the add-in will select all eligible corners. You can create dogbone fillets, "hidden" fillets cut into the short side of a mortice, "t-bone" fillets cut into the long side of a mortice. Another option is "minimal" dogbones which results in dogbones with a smaller visible gap between the parts compromised by a small amount of interference between the parts. This interference or overlap can maybe be filed off after cutting or the parts can be hammered together. (See "Minimal dogbone": http://fablab.ruc.dk/more-elegant-cnc-dogbones/)
    An option creates parametric dogbones that will automatically update with model changes.

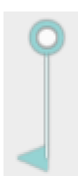
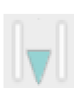
(An older version of this add-in which only does one type of dogbone fillets is here: http://tapnair.github.io/Dogbone/)

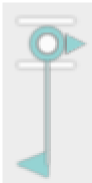
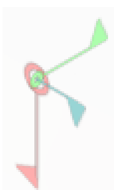
- **Nifty Dogbone by Ekins Solutions**
  https://apps.autodesk.com/FUSION/en/Detail/Index?id=3534533763590670806
  Nifty Dogbone has similar capabilities to the Dogbone v2 add-in above, but it has some additional handy configuration options such as a parameter to select the amount of interference for "minimal" dogbones. It costs $20. (There is a 30-day trial.) The add-in will automatically select corners on a body or face, and there is a feature that will update your dogbones after you have made changes in your model.
  https://ekinssolutions.com/product/nifty-dogbones-f360/

Fusion 360 Joint Fun

Joints in Fusion 360 reduce degrees of freedom of motion between Components. But rather than controlling individual degrees of freedom, Fusion 360 Joints define a collection of restrictions and freedoms based on the intent of the selected Joint type. When creating Joints, start by choosing a Joint type based on your intention.

Joint Types (their intents and their degrees of freedom)

	**Rigid**	Locks components together, removing all degrees of freedom.	**No DOF**
	**Revolute**	Allows the component to rotate around joint origin.	**1 rotation**
	**Slider**	Allows the component to translate along a single axis.	**1 translation**
	**Cylindrical**	Allows the component to rotate and translate along the same axis.	**translation and rotation on 1 axis**

	**Pin-slot**	The component can rotate about an axis and translate along a different axis.	**1 translation** **1 rotation**
	**Planar**	Allows the component to translate along two axes and rotate about a single axis.	**2 translation** **1 rotation**
	**Ball**	Allows the component to rotate about all three axes using a gimbal system (three nested rotations).	**3 rotation**

• Axes in Fusion 360 Joints have no connection to workspace or component axes or modeling orientation

When you create a Fusion 360 Joint you are often required to select one or more axes. For example, Revolute requires selection of an axis about which the Joint will rotate. The default is a "Z-Axis" which is normal to the plane of the selected Joint Origin and is always designated "Z". This "Z" axis has no connection with your workspace default modeling orientation axis preference ("Y-Up" or "Z-Up").

• Joint definitions require two Joint Origins on two Components.

• Joints constrain the motion of Component 1 relative to Component 2.

"Component 1" is the "from" or moving Component.
"Component 2" is the "to" or anchor Component.
e.g. a Revolute Joint will cause Component 1 to revolve about Component 2.

• Joint Origins are like magnetic couplers

When you select a Joint Origin on a snap point or when you create a Joint Origin you can think of that Joint Origin as an embedded magnetic coupler on the Component.

Additionally, you can define an offset to adjust the position of the Joint relative to the Joint Origins. e.g. You could define a Planar Joint that glides 1 mm above a surface.

• **Joint origins don't scale**

If you have a Component and you scale it, the Joint Origins do not scale along with it. Joint Origins must be manually repositioned to the scaled Bodies. Edit Joint will take you back in the Timeline to the pre-scaled context. So it may be easier to just recreate the Joint in the post-scaled context.

• **Joints can be used for Component positioning as well as for defining motion**

It is sometimes overlooked that joints can be used to position Components.
Multiple Joints can use the same Joint Origin.

• **Global Joints vs Sub-Component Joints**

Joints are created at the highest level of the Component hierarchy in common between the two Components. As a result, you can control the scope of a Joint to be more global or more local. Joints are always at the same level (peers) or lower in the Component hierarchy than their Components.

• **Rigid Groups are just a special kind of Rigid Joint that can have more than 2 Components**

• **You can't select a Grounded Component as the "from/Component 1" in a Joint.**

Grounded Components can only be the "to/Component 2".

• **Grounding an Assembly doesn't ground the Components or Joints within that Assembly.**

This can get a bit complicated.

Grounding a Component or Assembly locks the position of its Origin relative to its parent Assembly. And it implicitly locks its parent Assembly's Origin position and so on down to the root Assembly of a Design. (The root Assembly is always grounded/locked. ie. its Origin position can not move.)

This can sometimes lead to some unintuitive behavior, especially if you have a mix of Bodies (not contained within sub-Components) and sub-Components within an Assembly.

1. Ground the Assembly.
2. Click and drag a Body (not contained in a sub-Component); it doesn't move.
3. Click and drag a sub-Component; it moves.

But the problem is solved:

- Explicitly ground the sub-Component within an Assembly. (This will not ground peer Components.)
- Create Rigid Groups or Joints between grounded Components and ungrounded Components.
- A combination of the above two methods.

• **Component Drag is a Select menu toggle in Direct Modeling Mode and Base Feature Mode**

Component Drag allows you to directly reposition Components and exercise their Joints with the mouse. You can always drag Components in Parametric Modeling Mode, but Component Drag is an explicit setting that applies to Direct Modeling Mode and Base Feature Mode.

The Component Drag is a sticky global setting -- it retains its *on* or *off* state between Designs. (There is no default for Component Drag.) But it can be turned *on* and *off* from the Select menu in Direct Modeling Mode or Base Feature Mode.

There is no such menu toggle in Parametric Mode. But in Parametric Mode you must Capture Position to make any changes in Component position permanent.

Of course, Component drag will not override Joint constraints and grounding. e.g. if you can't move a Component because it is part of a grounded Rigid Group then it can't be dragged.

Component Drag changes the position of Components and Component Origins, but it does not change the position of Bodies relative to their Component Origin.

More Microcarpentry.com Laser Cutting Resources

https://docs.google.com/document/d/1aGGTGpdbrBCby33OYXgPP82Xol_6u2Pvb3i1L3AUqwE/
This is my curated list of useful resources for laser cutting in general.

More Learning Resources

Important: Freshness matters

Fusion 360 changes rapidly. New features and capabilities are added and older features are improved. Occasionally, features are removed and replaced. Terminology and user interface icons and dialogs change. So, it is particularly important to check the publication date of any learning resource. While some older resources may retain their relevance, you should generally start with newer (less than 3-year-old) resources first.

- **[Autodesk.com - Fusion 360 Learning and Support]**

  These Autodesk on-line resources are typically open to everyone at no additional cost. Many of these resources are easily accessible via the "?" icon in the upper-right of the Fusion 360 app screen.

  - Product Documentation - start here
    Autodesk's Fusion 360 Product Documentation is (to put it politely) "uneven", but it is still the first place to look as you learn the ins and outs of Fusion 360.
    There are also many tutorials embedded in the Product Documentation.
    Plus many parts of the Fusion 360 Product Documentation are much better than circular "uneven" entries. For example, the Manufacturing module documentation is very high quality. So don't assume that it's all junk.
    Autodesk is also working to flesh out the content in Fusion's interactive Learning Panel (invoked by Command-/ (Mac) or Ctrl-/ (Windows)). But (as of 6/2021) the Learning Panel is an "uneven" work-in-progress.
  - Self-Paced Learning (Multi-lesson Tutorials and Courses)
  - Events and Webinars - regular and special topic webinars
  - Community Forums - research and ask questions here.
    This is the 2nd place to look for answers after checking the Product Documentation. There's a good chance someone has asked your question already.
    Autodesk staff and community members on the forums will typically respond to queries within a day or so.
    Also report bugs here.
  - Autodesk Fusion 360 channel on YouTube - hundreds of tips and tutorial videos published by Autodesk staff
    - **YouTube: Designing a Lasercut Laptop Stand with Fusion 360**
      This essential tutorial by Autodesk's Taylor Stein demonstrates best practices for using Fusion 360 to create designs for laser cutting.
  - **Autodesk University** - free online classes and tutorials
    - **Design and Manufacturing Tips and Tricks for Fusion 360**
      A useful collection of tips.
      - Instructional Video: **Design and Manufacturing Tips and Tricks Fusion 360 - Scott Moyse (2019)**
      - PDF handout - **Design and Manufacturing Tips and Tricks in Fusion 360**

- **Debugging your Fusion 360 Design**
  How to fix your Fusion 360 design when it breaks, and how to prevent Fusion warnings and errors from occurring in the first place.
  - Article: **Debugging your Fusion 360 Design**
  - Companion Class: **Debugging your Fusion Design - Let's Get Rid Red and Yellow Features (2020)**

- **LinkedIn Learning**

  LinkedIn Learning is formerly Lynda.com

  The service has more than 20 courses on Fusion 360.
  LinkedIn Learning is $30/mo or $240/year, and a one month free trial is available.
  (A Sonoma County Library card will give you free access to courses on Linkedin Learning. Go to https://sonomalibrary.org/eresources)
  - **Learning Autodesk Fusion 360** - Taylor Hokanson, 2017
    Covers Fusion 360 interface, sketching, geometric modeling, and organic sculpting.
  - **Autodesk Fusion 360 Designing for Wood** - Thom Tremblay, 2016
    A brief exploration of the process of modeling a chair to be made from sheet materials.
  - **Learning Fusion 360 CAM** - Jon Helfen, 2018
    Covers taking a CNC-router-ready model into the Manufacture/CAM module, creating setups and CNC router cutting operations for each of 4 sheet parts.
    (Does not cover creating a CNC-router-ready model or operating the CNC. Nor does it address tool selection, speeds & feeds or climb vs conventional cuts.)

- **SolidProfessor.com - Solid Professor Fusion 360 Tutorials**

  Solid Professor has many courses in Fusion 360 including Sketching, Design, and CAM. A few of their videos are free, but complete courses require a subscription ($49/month).

  They have a series of courses from 2016 and a set of refreshed courses from 2019 and 2020. Make sure you do the more recent courses where available and fill in with the other interesting topics from the 2016 series.

  Many of these courses are similar to and complement Autodesk's Fusion tutorials. (They may have produced some of the content for Autodesk.)

- **YouTube.com**

  In addition to the significant library of videos produced by Autodesk, there are hundreds of independent videos related to using Fusion 360.

- **Books**

  Fusion 360 - There aren't a lot of books specific to Fusion 360.
  I've looked at these two. They are both lightweight and can easily be skipped.

  - ***Fusion 360 for Makers: Design Your Own Digital Models for 3D Printing and CNC Fabrication*** - Lydia Sloan Cline, 2018
    This is a very cursory book. It's mostly screenshots with very little discussion or detail. (There is a new version of this book coming out in June 2021.)
  - ***A Beginner's Guide to 3D Modeling: A Guide to Autodesk Fusion 360*** - Cameron Coward, 2019
    This book is more detailed than the *Fusion 360 for Makers* book. But the lessons and discussion are still very brief. It would be a very quick-read, but it might be useful to a beginner, and there are some tips presented.

  Design and CNC

  - ***Fundamentals of CNC Machining: A Practical Guide for Beginners***
    Autodesk, Inc., 2014;
    Free PDF eBook from Autodesk.
    https://www.autodesk.com/campaigns/fundamentals-of-cnc-machining
    https://academy.titansofcnc.com/files/Fundamentals_of_CNC_Machining.pdf

    Fundamentals of CNC Machining is a textbook which solidly presents the concepts, software and techniques of CNC machining. The textbook is written to accompany a course teaching how to use a Haas CNC mill in conjunction with HSMWorks CAM software and Solidworks (ironic because Solidworks is a Fusion 360 competitor). But there is very little material that is specific to Solidworks nor Haas-brand CNC mills. Autodesk acquired the HSMWorks technology in 2012, and that technology became Fusion 360's Manufacturing/CAM module. So the concepts and methods presented here are applicable to the use of Fusion's Manufacturing software and can be applied to any CNC machine supported by Fusion 360.

  - ***Design for CNC*** - Anne Filson, Gary Rohrbacher, Anna Kaziunas-France

Make Community, LLC, Nov. 2017
http://www.designforcnc.com/
http://atfab.co/

Design for CNC was written by the AtFAB group who have put together a wonderful collection of open-source [CC BY-NC-SA 3.0] furniture designs.
This book provides a gentle introduction to Design with a capital D. They go through the problem domain, the design constraints and they develop a design framework for furniture made with CNC router techniques and plywood. They then demonstrate how they have applied the design framework in the design of a progression of eight furniture models.
But the approach falls short in that they have based their design tutorials on the integral use of SketchUp for 3D CAD (SketchUp Shop: $119/year) and VCarve for 2D CAM (VCarve Pro: $699 or VCCut Pro: $449, Windows only). (The VCarve free trial versions, the free version of SketchUp and the "lite" VCarve/VCut "Desktop" versions do not include the features needed to complete the lessons in this book.) SketchUp and VCarve are both very good programs, but they are a different toolset from what we have been focused on here. And this book creates a very high barrier to learning unless you have access to the full versions of these programs.
In summary, Design for CNC attempts to present some complicated topics, but it succeeds only about half the time. It's still worth reading for the discussions of design and the presentation of CNC router concepts.