AUTODESK.

SLICER
FOR FUSION 360

Table of Contents

What is Slicer for Fusion 360?

Slicer for Fusion 360 is a tool to turn your digital ideas into something you can hold. It slices and converts digital 3D models into 2D patterns you can cut out of any flat material. Slicer for Fusion 360 also creates 3D instructions you can interact with, to help build a model.

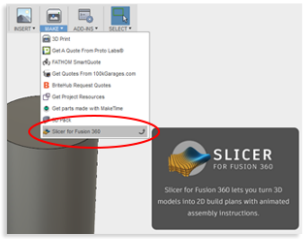
What’s new

Now you can send a model from Fusion 360 to Slicer for Fusion 360

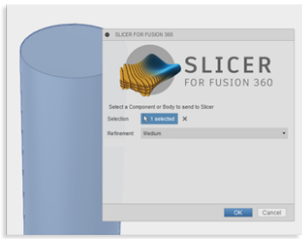
Create a model in Fusion 360 and with a few clicks you can send your model to Slicer for Fusion 360. Apply various slicing techniques to your model and create 2d plans in EPS, DXF or PDF formats that you can cut using Laser cutter or CNC machine.

It’s fairly easy to send a model from Fusion 360 to the Slicer application.

1. Install Slicer for Fusion 360 from the Fusion Store.
2. Select the Slicer add-in from the list and click on Run and you will see the add-in installed under ‘Make’ as shown in the image below.
3. If Fusion 360 is not running, at the time you install the Slicer for Fusion 360, then when you launch Fusion 360, the add-in gets automatically added under ‘Make’.
4. Click on the Slicer add-in in Fusion 360.
5. Select a component that you want to send for slicing.
6. After clicking ok, Slicer will launch with the component showing in the Slicer for Fusion 360 scene.
7. Perform the required slicing operations and export the plans to your local machine or to Fusion Team, in 2d formats like EPS, DXF or PDF.
8. Remember, that any mesh cannot be sent from Fusion 360 to Slicer application.

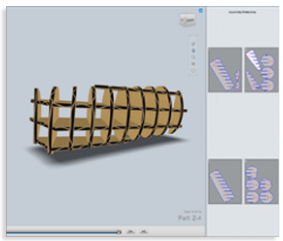


Click on Slicer for Fusion 360 option



Select component

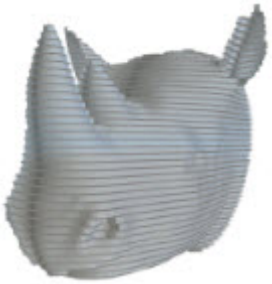
Select component and click ok. Slicer for Fusion 360 will be launched.



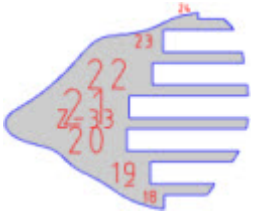
Slice model using different techniques

Use any of the techniques from the application and save or export the files locally or to Fusion team.

Important Terms



Slice: The cross sections you get by cutting straight through your model at any angle. A slice can contain multiple parts.



Part: The individual pieces that make up a slice. They are assembled together later.

Cut Sheets: The sheets of material your parts are cut out of to build the model.

Slot Offset: Set the cut width to determine how wide the actual cuts on your cut sheet are. The cut width is based on how much material your cutting tool (laser, saw, etc.) removes.

This makes assembly easier. A slot offset of "0" creates a notch that equals the material thickness. Depending on the material you intend to use, a Slot Offset of 0 could make assembly difficult to slide one piece through another. Decreasing the slot size will force slices to have to squeeze together (fine for cardboard), increasing it allows for "breathing room" in the notches during assembly (helpful if your material, for example plywood varies slightly in thickness).

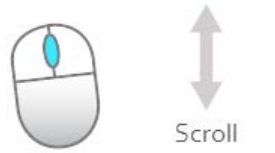
Navigation

There are three ways to change your view: with your mouse, the Navigation tools, and View Cube.

Mouse navigation

Tumble: Right-click anywhere around your model and drag the cursor to change your view.

Pan: Click and hold the scroll wheel and drag the cursor to sweep your camera view in any direction.



Zoom: Roll the scroll wheel up and down to zoom in and out.

Navigation tools

Use the navigation tools to tumble, pan, zoom, and frame your content.

Tumble: Click and drag in any direction to change your view.

Pan: Click and drag in any direction to pan your camera view.

Zoom: Click the right side of the wheel and drag the cursor up and down.

Look-at: Click to center and frame your view on the current side.

Toggle View: Click to switch between orthographic and perspective view.

> **Orthographic** - Snaps your camera view to the closest straight-on view of your model to see it without any perspective. This is great for checking the alignment of particular parts or features in 3D.
> **Perspective** - Snaps your camera view to see your model in perspective.

Using the ViewCube

The ViewCube lets you quickly and easily switch between scene views. It is the cube that appears in the upper-right corner of the scene. Its faces are labeled with the camera view in relation to the 3D scene.

Click a face to automatically adjust your view to match that face. Click the edges or corners of the ViewCube to adjust your view.

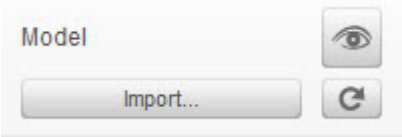
There are 26 standard available views (6 face views, 8 corner views, and 12 edge views).

When you mouse over the ViewCube, regions of it appear highlighted. These are the areas of the cube that will be selected when you click. **Home** appears, as well. Click it to return to the default perspective view.

Open/Import

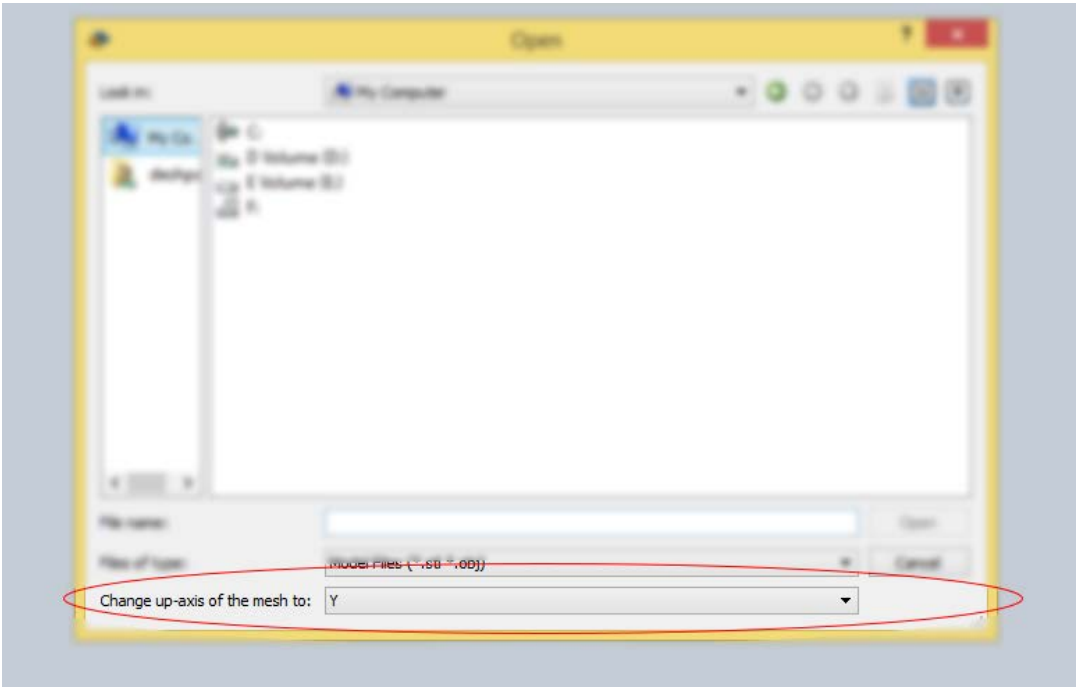
Open and import from my computer

You can **open** 3DMK file locally. If you have STL or OBJ on your computer, you can **import** your model by clicking on the import button at the top of the menu. Import button is explicitly used for local files.



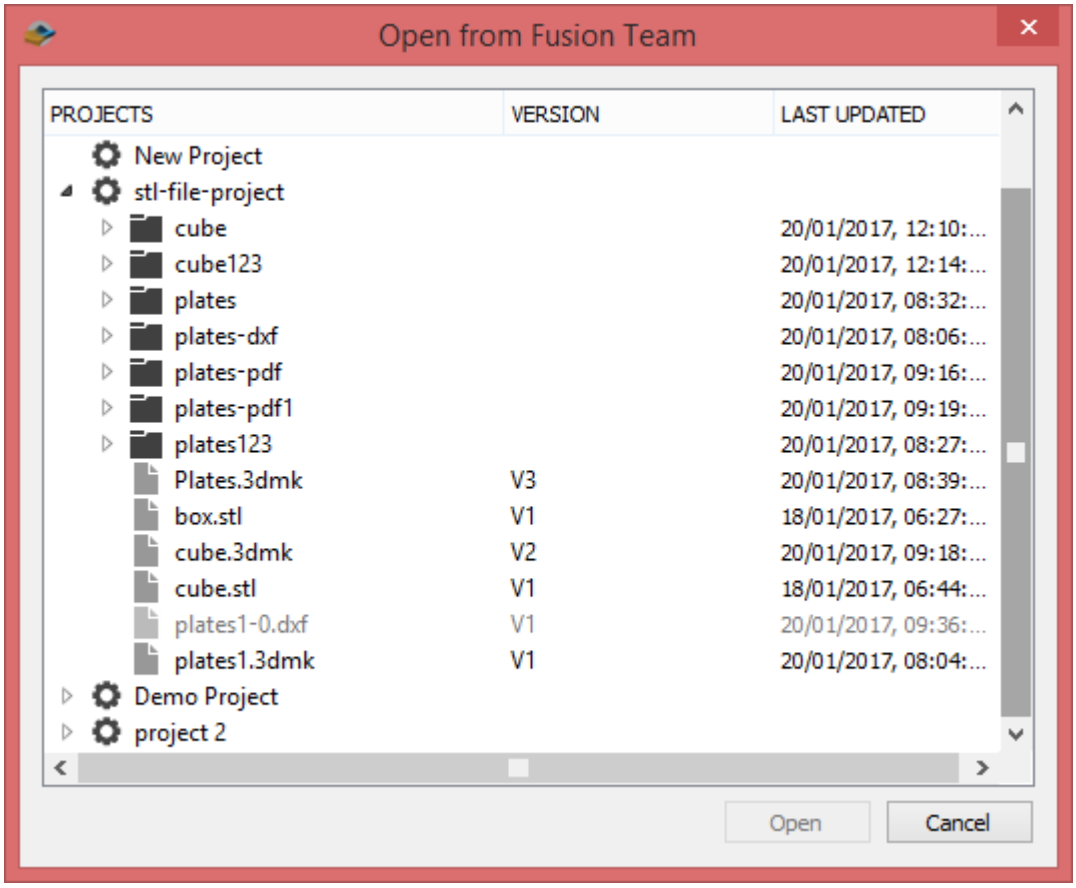
If your model comes in looking misaligned, click  to reposition it.

While importing an STL or OBJ, if you are coming from another application which has Z/X axis up, then you can change the axis of your model within the system dialog, so that you get the model in the right position. Please check the image below.



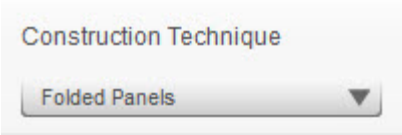
Open models from Fusion Team

You can open 3DMK, STL, OBJ files from Fusion team by selecting **'Open from Fusion Team'**.

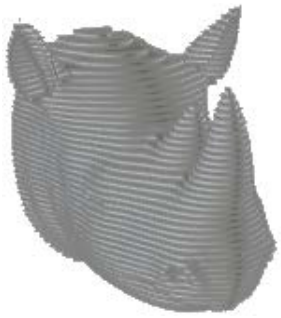


Sometimes, if you haven't used the application for a longtime, it may happen, that you send something from Fusion 360 to Slicer, and when you try to open/save/export to Fusion team, these dialogs may not contain any data. If this happens, sign out and sign in again.

Construction Techniques



With a project loaded, the next step is to preview the different construction techniques to use for slicing.

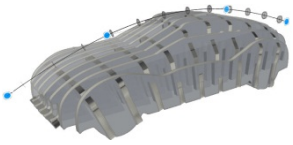


Stacked Slices

Cross sections your 3D model, cutting it into slices you can glue and stack on top of one another. Use the Dowels option to make it easier to line up and assemble your model. You can recreate the model using any flat material you can cut.

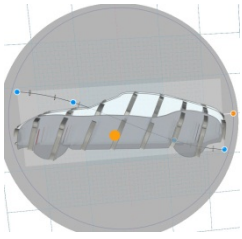
Interlocked Slices

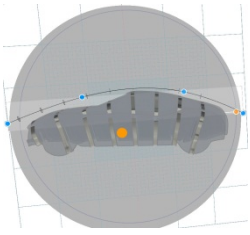
Cuts your 3D model into two stacks of slotted slices. Lock them together in a grid, like when building a 3D puzzle. This uses less material than stacked slices.



Curve

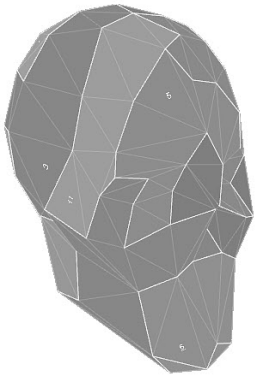
Cuts slices perpendicular to a curve, resembling ribs. Use this for organic shapes, such as for modeling a brontosaurus. Also, use the Navigation tools to help rotate your view to see the curve.





Radial Slices

Cuts your 3D model into radiating slices from a central point. Use this for a round symmetrical object, such as a vase.



Folded Panels

Separates your 3D model into 2D segments of triangular meshes. These segments (panels) are folded multiple times, then attached using one of ten different joint types. Use paper, cardboard, even sheet metal.

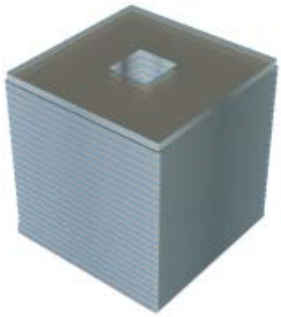
Exporting Panelized Mesh as a Single File

See the section titled "**Simplify Form**" to learn how to export your entire model as a simplified, panelized mesh ready for 3D printing.

3D Slices

Cross sections your 3D model, similar to Stacked Slices. Rather than a stepped section for each slice, the section conforms to the surface of the 3D model.

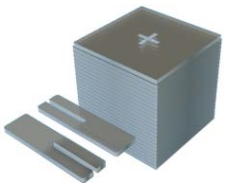
Dowels



Anchor your stacked slices in place, keeping your slices aligned during assembly.

Diameter sets the diameter of the dowel used.

> **Note** The units set in **Object Size** (inches) are the same units used when setting the dowel diameter.



Shape sets the shape of the dowel. Choose from **Square**, **Pencil**, **Round**, **Cross**, **Horizontal Slot**, and **Vertical Slot**. When you select Cross, Horizontal Slot, or Vertical Slot, Slicer for Fusion 360 creates the dowel for you.

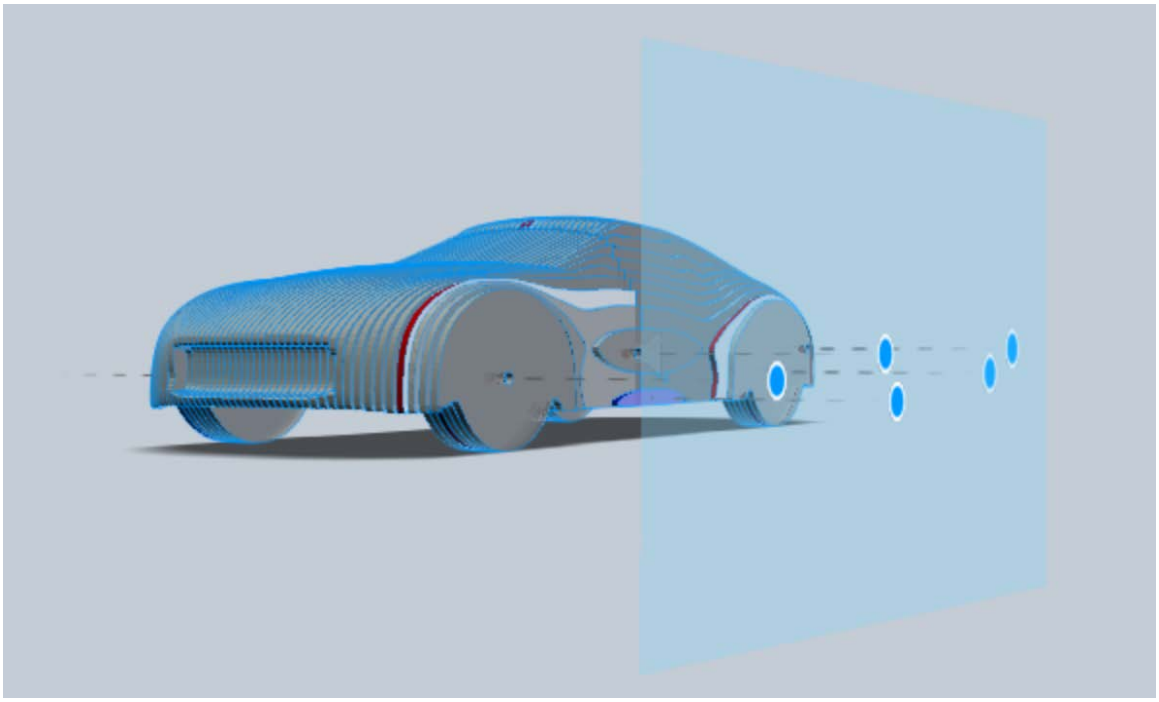
Dowels can be added and subtracted.

To add a new dowel highlight the dowel pane in the left nav then click on the light blue 3D pane above the model.
A dowel will appear with a dashed line signifying its trajectory.

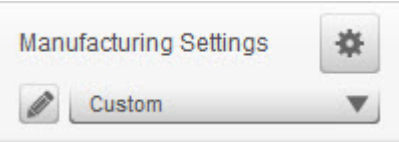
To move an existing dowel click its circular marker on the light blue pane, it will turn orange, then reposition it with the mouse.

There are two ways to delete a dowel: highlight it by clicking once then pressing the Delete key, or hold down the Control key and single click on the dowels marker.

To regenerate dowels automatically press the “Automatic” button.



Manufacturing Settings



Now, set the size of the sheet your project is cut from. The smaller your object is compared to your sheet size, the more parts you can fit on it.

Selecting a standard setting

From the **Manufacturing Settings** drop-down menu, choose from a number of standard cardboard sizes to cut your model out of. The height, width, and thickness of your sheet size are automatically updated.

Changing settings

If you plan to use materials different from the presets, click to enter the units, slot offset, and length, width, and thickness of the material in the field along the bottom of your screen.

Creating a custom preset

If you plan to use a custom material repeatedly, create your own preset. Once created, your new setting will appear in the **Manufacturing Settings** drop-down menu.

1. Click to access **Manufacturing Settings**.
2. Click+, then name the preset and enter its manufacturing settings.
3. When finished, click **done**.

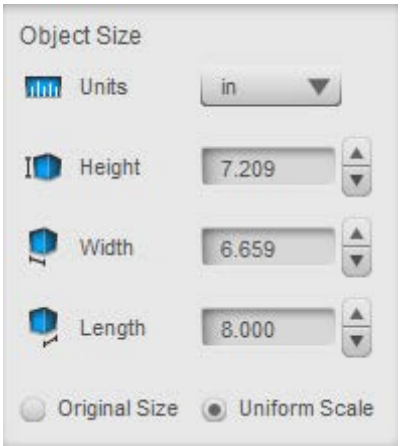
Variations

To create slight variations of a setting, click the setting to select it, use to make a duplicate, and then tweak the settings.

Delete a manufacturing setting

In **Manufacturing Settings**, click a setting to select it and click−.

Object Size



How big do you want your model to be? Set its physical size. The larger your object, the more sheets you'll need to create your project. We put this step later in the process on purpose because depending on your computer's capabilities you're better off not scaling your object up right away. The larger your object the more slices are generated and the more CPU & is necessary to render.

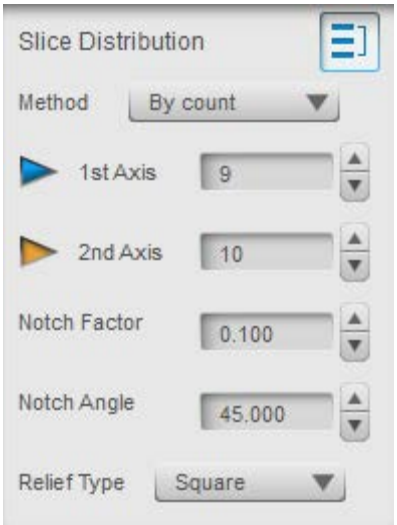
Original Size

Eliminate any sizing changes made and revert to the STL size, once you've set the units.

Uniform Scale

Uniformly scale your project in all directs (height, width, and length) when selected. When not selected, scale your project in one direction at a time.

Slice Distribution



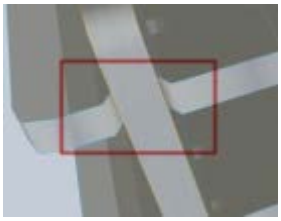
(*Only available for interlocked, curve, and radial sliced projects.)* Each construction technique has some options unique to it.

Set the method for how slices will be distributed. Choose from count, distance, or create a custom setting. Use the two options found after Method to determine:

- the number of slices in each direction
- the distance between slices

Increase these numbers to give more definition to your model. Decrease these numbers to use less material. Click  to access tools for duplicating, deleting, and evenly distributing slices.

When working with large pieces, parts can rack and twist, making assembly almost impossible. Also, by default, Slicer for Fusion 360 creates perfect squared outside corners on its slots. This can also cause issues during assembly. Use **Notch Factor** and **Notch Angle** to eliminate these issues by changes the mouth of the notches, flaring them.

		
Significant notch flaring	Some notch flaring	Slight notch flaring

Notch Factor flairs the mouth of the slot by a specific amount, relative to the width of the slot.

Notch Angle sets the angles relative to the direction of slot the notch will be cut from. Using an angle of 45 degrees should aid with easy assembly.

Relief Types

Set the type of notches used to connect your slices. This is particularly useful when cutting a model using a computer controlled (CNC) machining device. They help eliminate issues during cutting. It makes it possible to choose from the following:

Square *(default setting)*

Horizontal Use if the tool diameter in Manufacturing Settings has a value greater than zero.

Vertical Use if the tool diameter in Manufacturing Settings has a value greater than zero.

Dog Bone *(also known as Relief)* enables slices to fully penetrate one another, since there are no inside rounded corners. Use it in areas with high tensile stress for stress relief.

To control the creation of these, you can specify the cutting tool diameter.
In the Manufacturing Settings, tap  . Create a custom setting, and then set the Tool Diameter.

Note: If Tool Diameter is set to zero, no Dog Bone notches can be generated.

Tip: When getting your model ready for slicing, use **Slice Direction** in conjunction with Slice Distribution.

When working with *interlocking slices*, you can click-drag any slice to change its position and the spacing between slices. Use the tools in the toolbar to do the following:

Evenly Distribute

To evenly space slices after moving individual slices, click  .

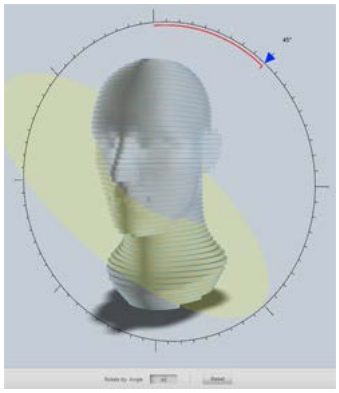
Duplicate

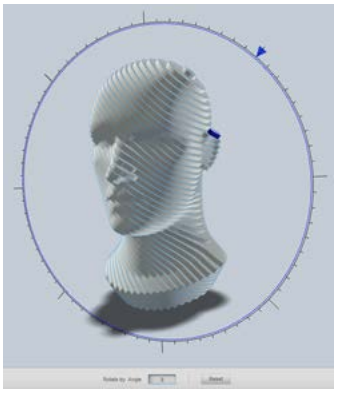
To add a slice next to the selected slice, click  . It will follow the contours of the model.

Delete

To remove the slice, select a slice and click  .

Slice Direction



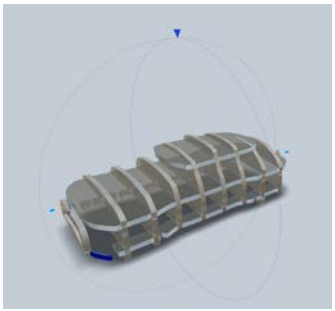


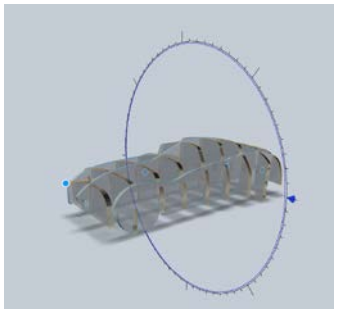
Click  , then click on one of the two rings to constrain the direction of the slice direction to 5 degree increments. Drag the blue handle and release to update the slices.

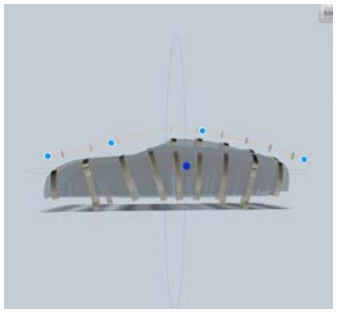
You may disable snapping for a freeform slice direction by clicking off of the rings and then dragging the blue handle.

Clicking Reset at the bottom will return the blue manipulator to the starting position.

With *curve slicing*, the curve lives on one plane. Making changes to the slicing direction makes it easier to reshape the curve.

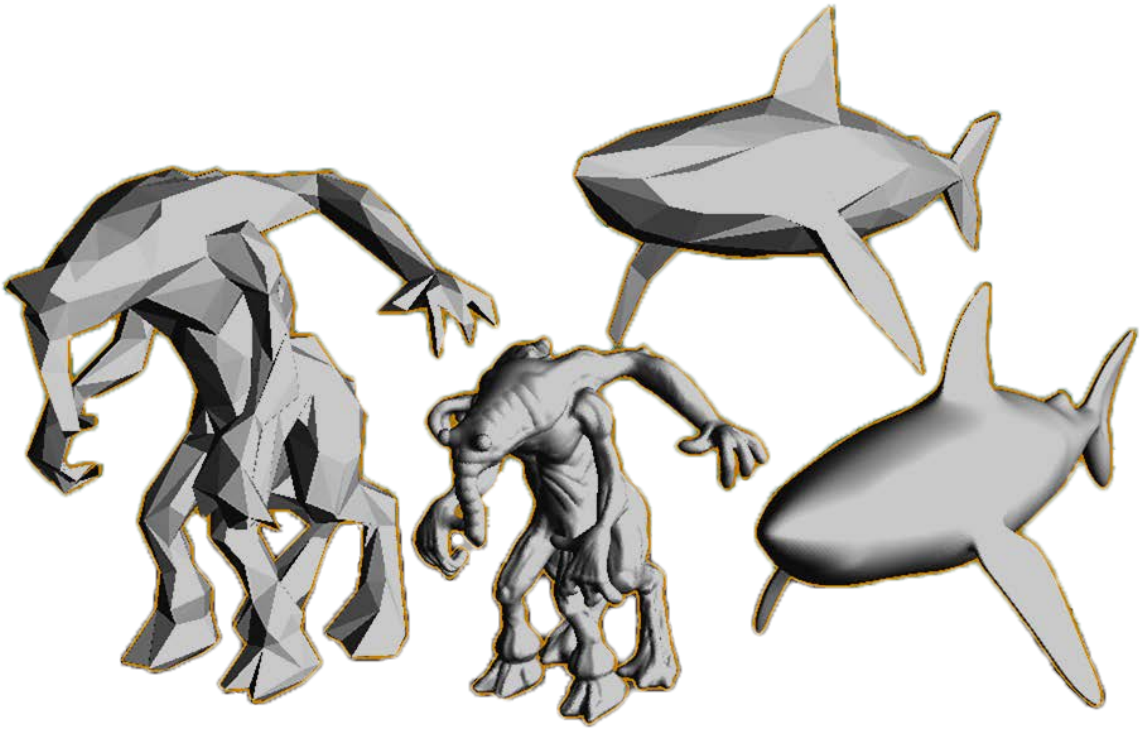




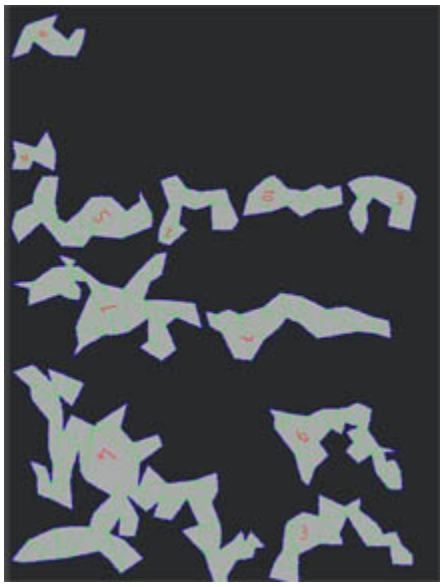


Simplify Form

(Only available for folded panel projects.) Changing the number of vertices and faces to simplify a model's shape serves two purposes: 1) If you have a model with tens of thousands of triangles Slicer for Fusion 360 could potentially export complex strips that may be exceedingly difficult to assemble once you are building the actual object. Reducing the number of faces and/or vertices will make assembly quicker and easier. 2) Reducing the number of triangles (sometimes called "decimation") can create interesting results you may find aesthetically pleasing. For this second reason we have added an item to the main menu called "Export Sliced Mesh…" so you can keep the intact, decimated model.

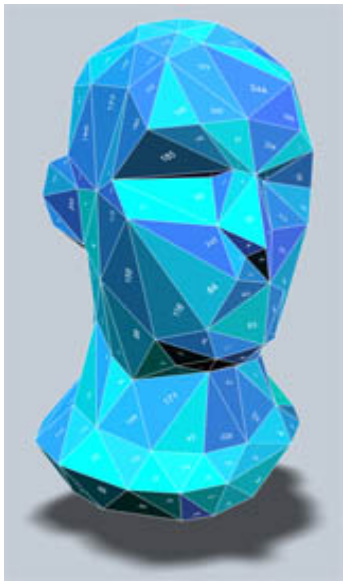


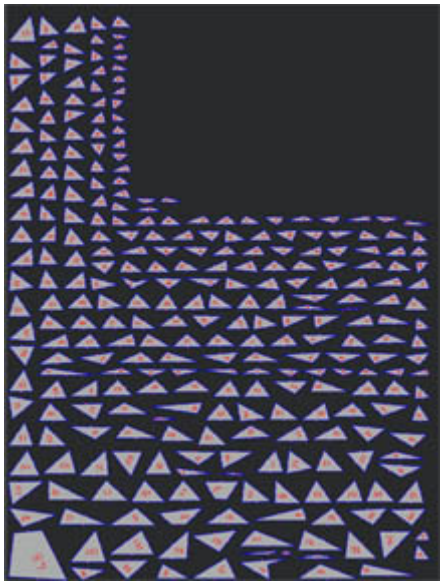
Optimize Panels



Adjust the vertex count to reduce the number of faces your imported model is represented by. In general, more faces can make for a more detailed end result and complex assembly.

Use the **Perforate** option to add perforation lines on the 2D panels that have a thickness equivalent to your chosen material. By default you will have fold lines where faces of the same panel meet.

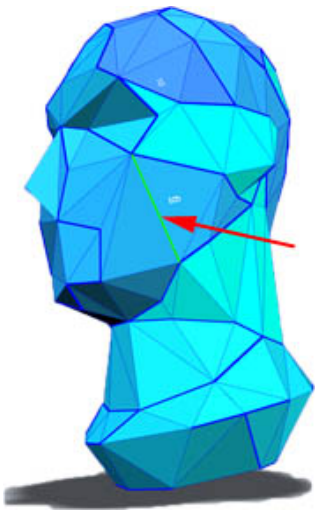




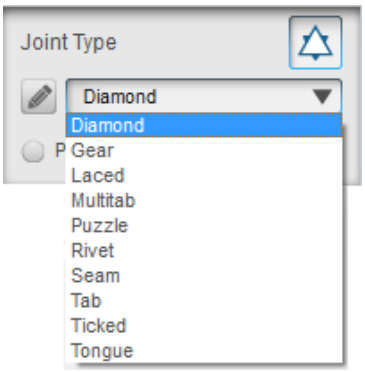
Split Panels will create a panel from every face in the optimized model.

Add/Remove Seams

Use **Add/Remove Seams** to interactively adjust the panels on your model. Click on an edge to **add** a seam and separate large panels into smaller ones. Alternatively, you may **remove** seams to create larger panels for your project.

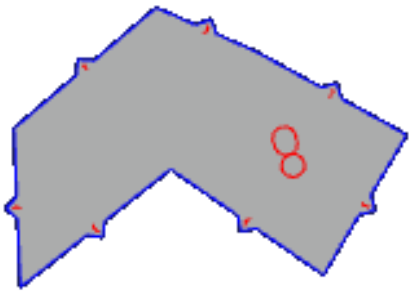


Joint Type



(Only available for folded panel projects.) Select a joint type to determine the method used for connecting folded panels together. If your model shows construction errors, changing the joint type can solve some issues.

Tap  to access the Joint Type toolbar. The options that appear help you to eliminate model assembly errors, change spacing between rivets, set the size of ticks, set the width of seams, and more.

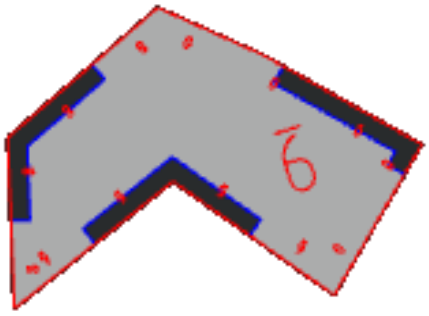


Diamond

Connect panels by folding and affixing (gluing or welding) the triangular ticks. Change **Tick Radius** to set the width of the ticks.

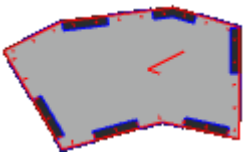
Tick Radius = 0.25

Tick Radius = 0.50

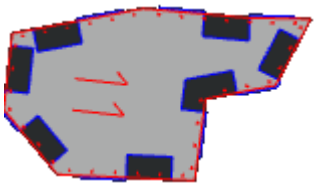


Gear

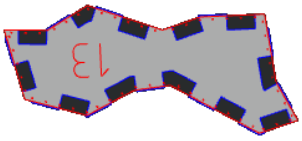
Connect panels by folding and affixing (gluing or welding) the rectangular ticks. The dark areas of the image need to be cut out. Change **Tooth Radius** to set the distance from the edge of the sheet for the cutouts. Change **Tick Spacing** to set the spacing between the cutouts. Change **Tooth Scale** to set the height of the teeth.



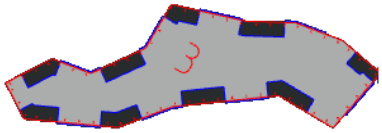
Tooth Radius = 0.20



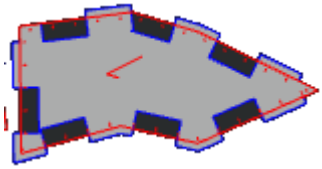
Tooth Radius = 0.50



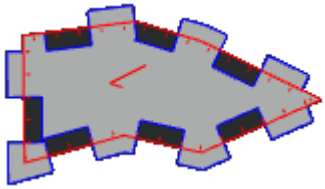
Tick Spacing = 1.00



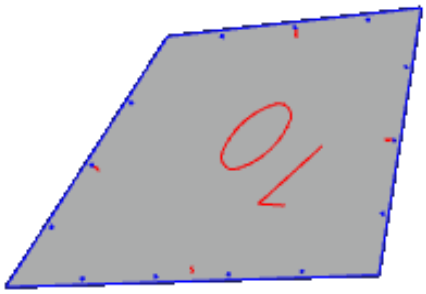
Tick Spacing = 1.50



Tooth Scale = 0.40

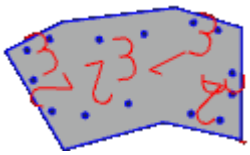


Tooth Scale = 0.90

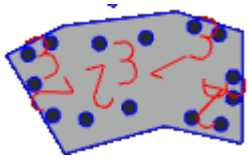


Laced

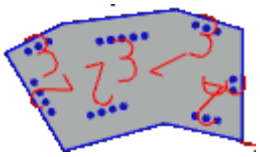
Connect joints by lacing sheets together. Change **Hole Radius** to make the holes bigger. Change **Joint Space** to set the spacing between rivets used to connect the joints. Change **Tick Radius** to set the distance of the rivet holes from the edge of the sheet.



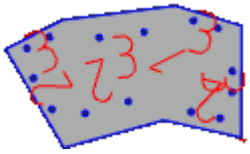
Hole Radius = 0.065



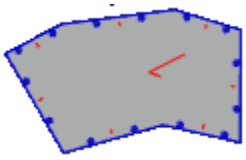
Hole Radius = 1.500



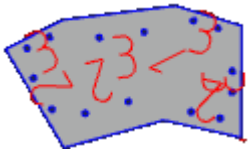
Joint Space = 0.25



Joint Space = 1.00



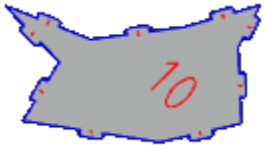
Tick Radius = 0.065

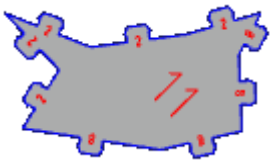


Tick Radius = 0.750

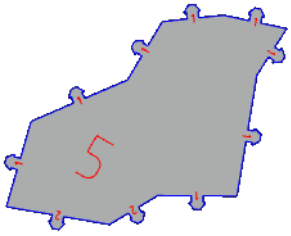
Multitab

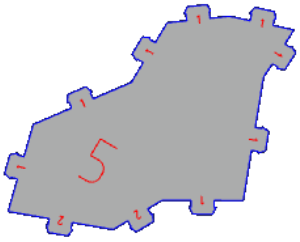
Connect joint by fixing ticks together. Change **Tick Radius** to change the height of the ticks. Change **Tab Fraction** to change the width of the ticks (tabs).



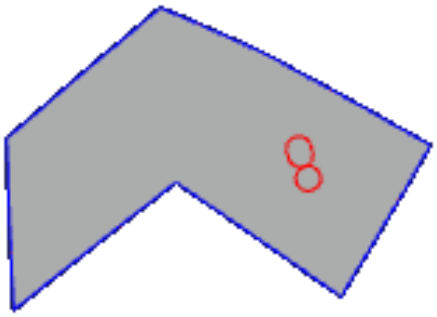


Tick Radius = 0.050 Tick Radius = 0.150





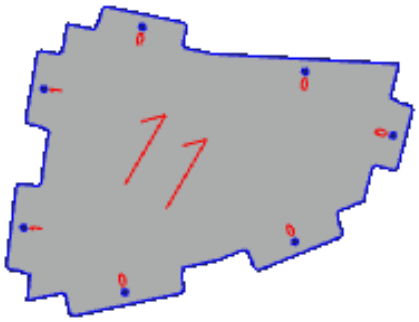
Tab Fraction = 0.25 Tab Fraction = 0.50



Puzzle

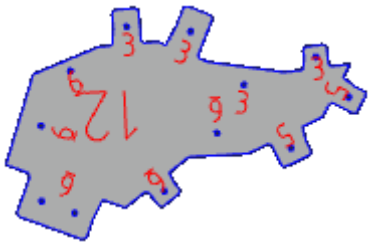
Connect joint by fitting pieces together. Change **Tick Radius** to change the height of the ticks.

Tick Radius = 0.0500 Tick Radius = 0.0955



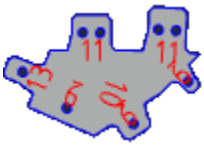
Rivet

Connect joint by riveting ticks to one another. Change **Hole Radius** to make the holes bigger. Change **Joint Space** to increase or decrease the number of rivets used to connect the joints. Change **Tick Radius** to set the distance of the rivet holes from the edge of the sheet.



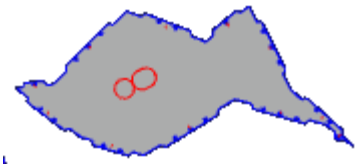
Hole Radius = 0.075

Hole Radius = 0.100



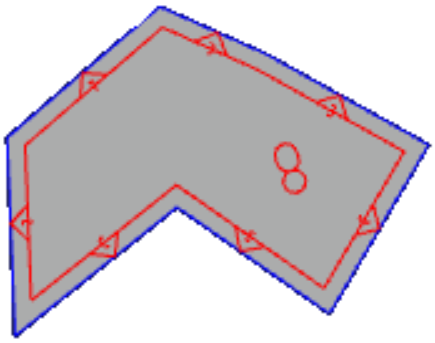
Joint Space = 0.500

Joint Space = 1.500



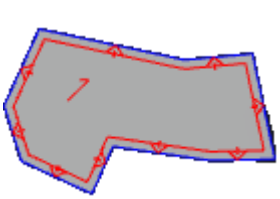
Tick Radius = 0.050

Tick Radius = 0.500

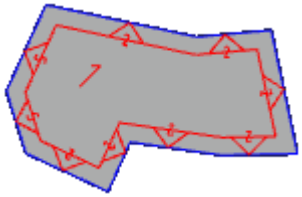


Seam

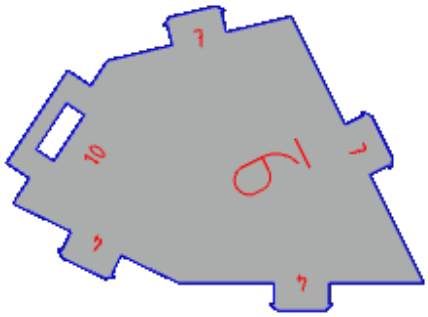
Connect joints by sewing them together (like a sewing pattern). Change the **Seam Radius** to change the width of the seam's panel border.



Seam Radius = 1.25

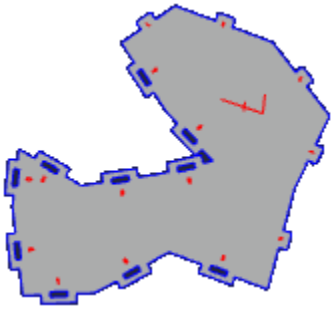


Seam Radius = 1.50



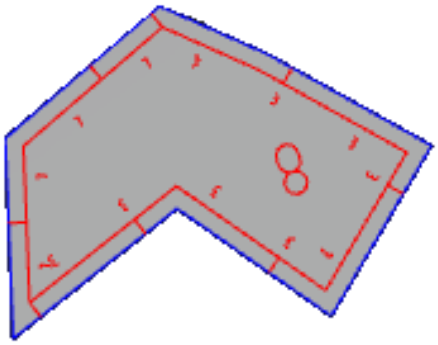
Tab

Connect joints by inserting a tick into slots. Change the **Tick Radius** to change the width and length of the slots and ticks.



Tick Radius = 0.25

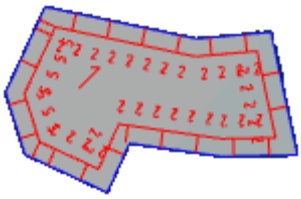
Tick Radius = 0.40



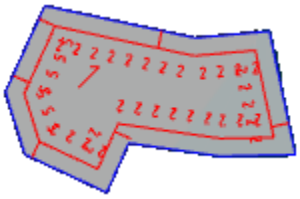
Ticked

Connect joints by connecting the ticks along the seams

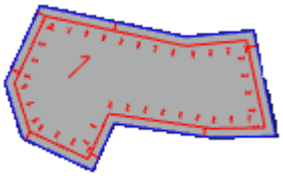
Change **Tick Space** to make the ticks closer to one another or further apart. Change the **Seam Radius** to change the width of the seam's panel border.



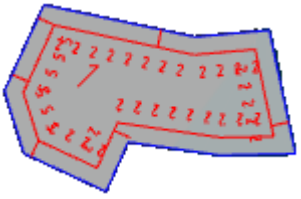
Tick Space = 1.00



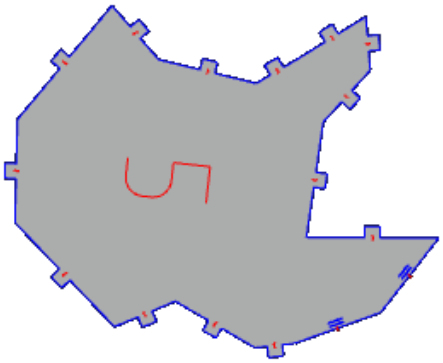
Tick Space = 4.00



Seam Radius = 0.25

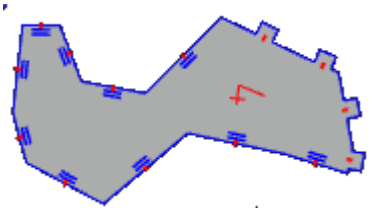


Seam Radius = 0.50

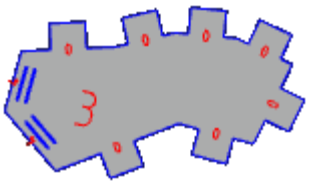


Tongue

Connect joint by inserting a tick into slots. Change **Tick Radius** to change the tick length and width and slot width.



Tick Radius = 0.25



Tick Radius = 0.50

Changing manufacturing settings

Click  to access joint type manufacturing settings.

Creating a custom joint preset

If you plan to use a custom material repeatedly, create your own joint preset. Once created, your new setting will appear in the **Joint Type** drop-down menu.

1. Click  to access **Manufacturing Settings**.
2. Click +, then name the joint preset and enter its manufacturing settings.
3. When finished, click **Done**.

Variations

To create slight variations of a joint type, click the joint type to select it, use  to make a duplicate, and then tweak the settings and click **Done** when finished.

Modify Form

To help solve "Output" issues, click  to access **Modify Form** tools. Select a tool, adjust its slider, and click **done**.

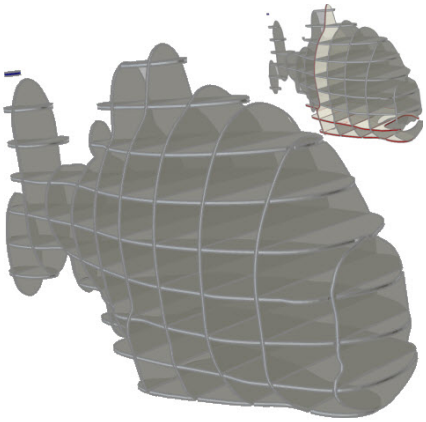
Click  to see the changes made to the mesh.

Important: Keep in mind; changes made with these tools are applied to the whole model, not just the selected slice.

Hollow

For models where you want to reduce the weight and amount of materials used, but stay true to the shape, use **Hollow**. It creates a shell of the model.

Thicken



For cutouts that are too thin and would fail during printing, use **Thicken**. It widens every piece to eliminate the problem, slightly modifying your model.

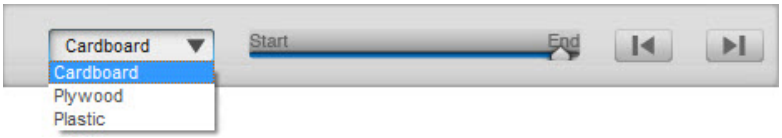
Shrinkwrap

For models that have details too fine to cut out, use **Shrinkwrap**. It approximates and smoothes the shape enough to create an object that still resembles your model. With sharp T-rex teeth, **Shrinkwrap** rounds and softens them. The end result is a softer T-rex with teeth that more reliably show up in the physical object you build.

- To make watertight models, use **Shrinkwrap** to close up any holes.

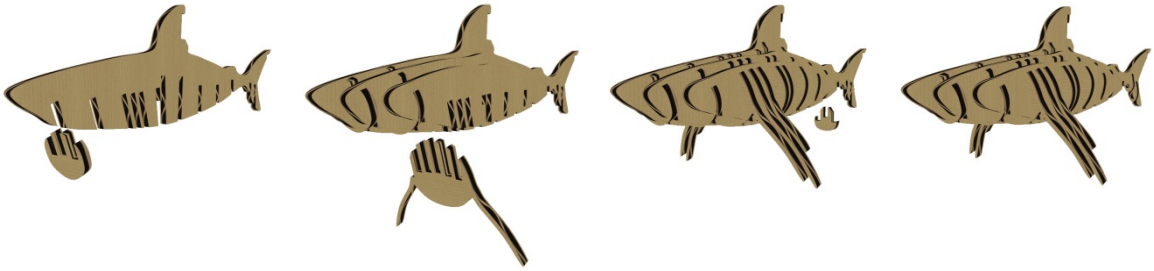
Assembly Steps

To view an animation of how to assemble your model, click  . Along the bottom of the screen, select a material.

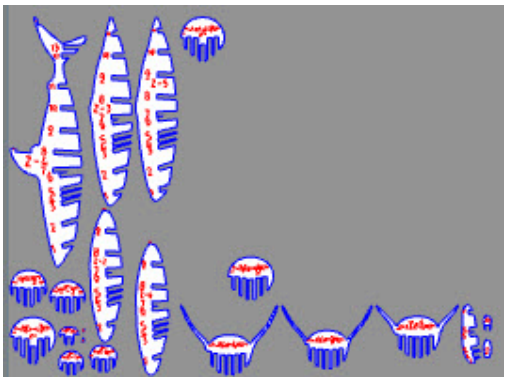


Cardboard Plywood Plastic

Drag the slider left and right or use  and  . Use the animation as the assembly instructions when building your model.



To view an Assembly Reference sheet, click one in the panel to the right.



To move the sheet around, click  and drag the sheet.

To zoom in on a part, click  .

To fit the sheet to view, click  .

To return to your model, click  .

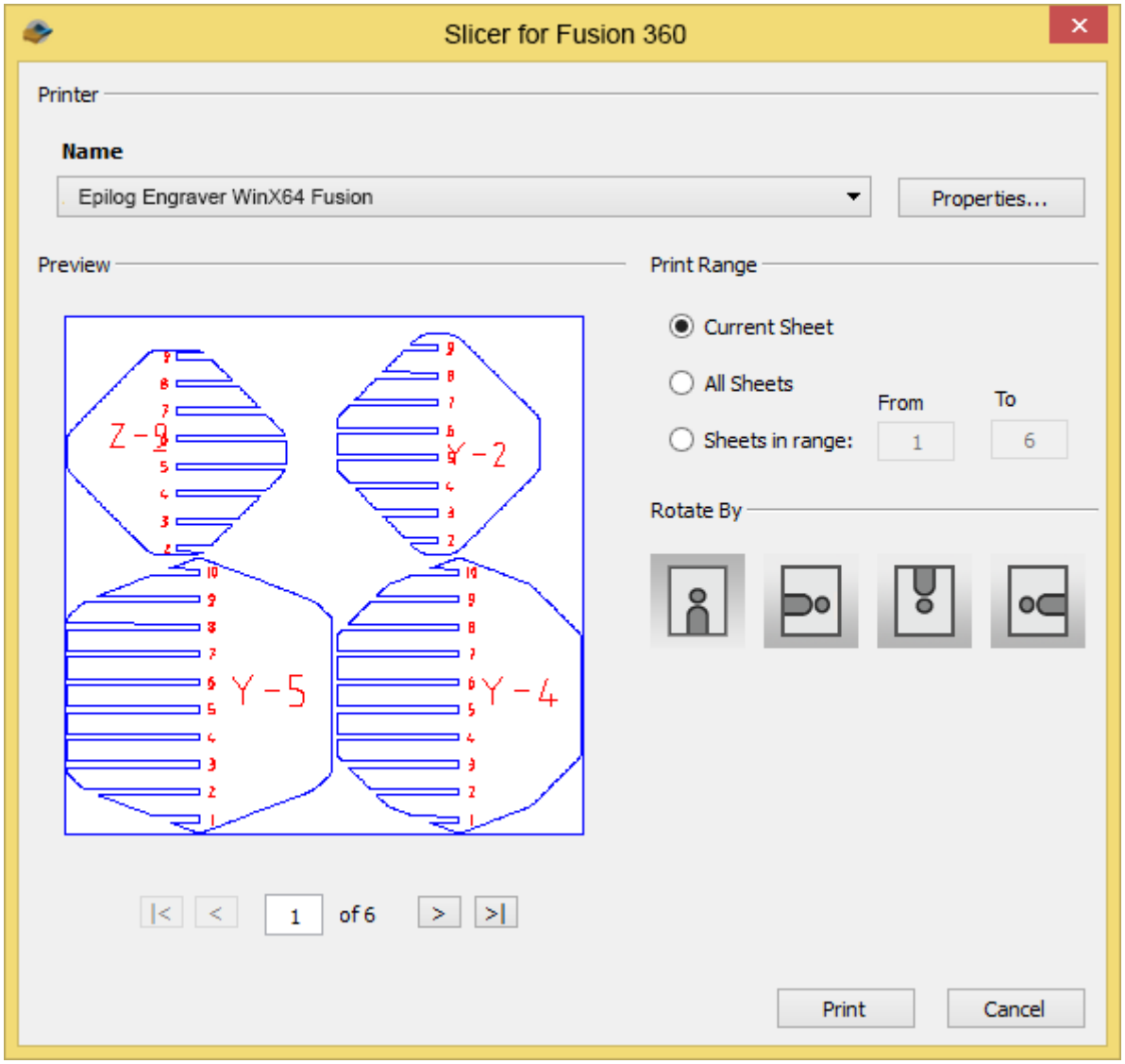
To print out instructions, see Output – Get Plans.

Output – Get Plans

You may either print your 2D plans to a system laser cutter/printer or export them to a file.

Print

Click the Print button to send the sheets to a system laser cutter or printer. You may preview how the sheets look for the chosen printer or laser cutter.



Print Dialog

Save

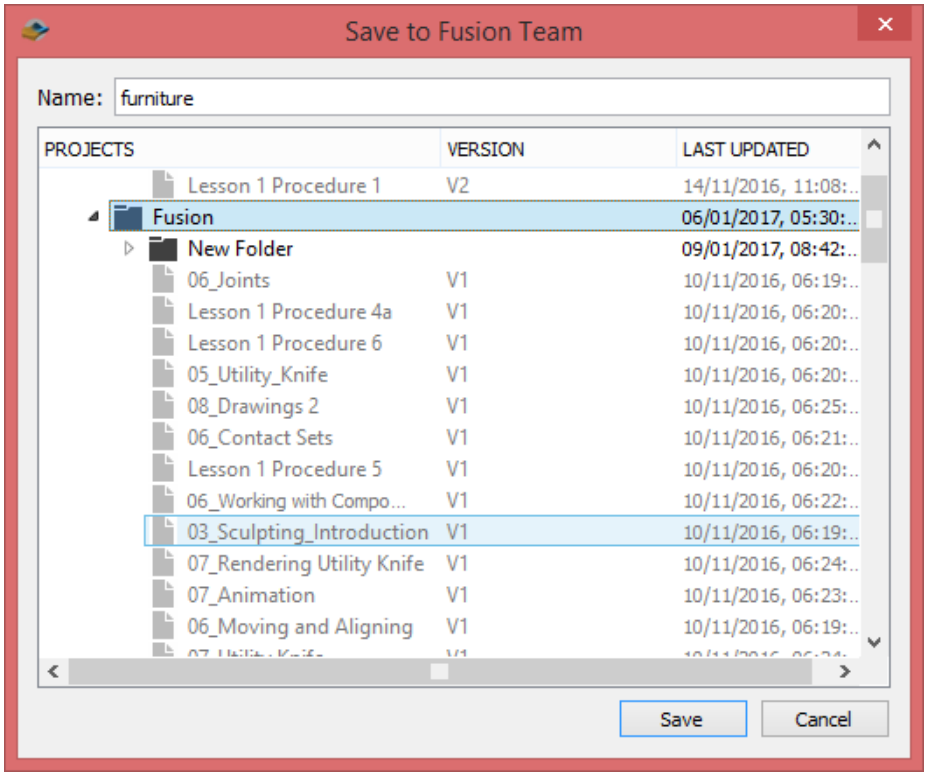
Save to my computer

To save your work to Fusion Team select '**Save to My Computer**' from the app menu.

To send an STL or OBJ file of the sliced mesh model (not the pre-sliced original), use **Export Mesh** in the Appmenu.

Save to Fusion Team

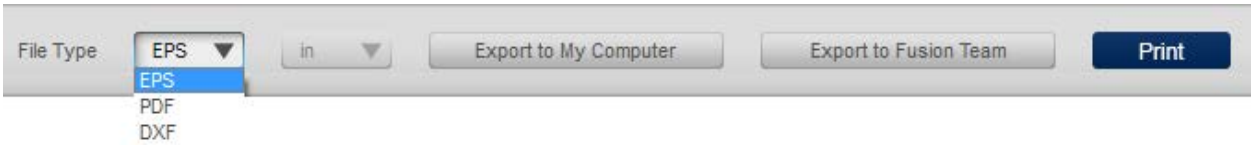
1. To save your work on Fusion Team select '**Save to Fusion Team**' from the app menu.
2. Provide a name and then save the file under the Project/Folder that you want to save. Remember that you need to provide the file name in English, otherwise the application may not work properly.
3. We have a versioning system in place for the files. So if by chance, you overwrite a file, you can still retrieve the previous version, on Fusion team.



Export

Export to My Computer

To export your 2D plans, click , select one of the following file types, then click **Export to my computer** to save the file to your hard drive.



1. **EPS** Choosing EPS will create a zip folder with separate files for each sheet.
2. **PDF** Is a good choice for people who don't possess an application that reads EPS files. Choosing PDF will create a single PDF file containing everything on separate pages.
3. **DXF** and **EPS** have the advantage of separating text on one layer and profiles on another. This separation can be quite useful when laser cutting.

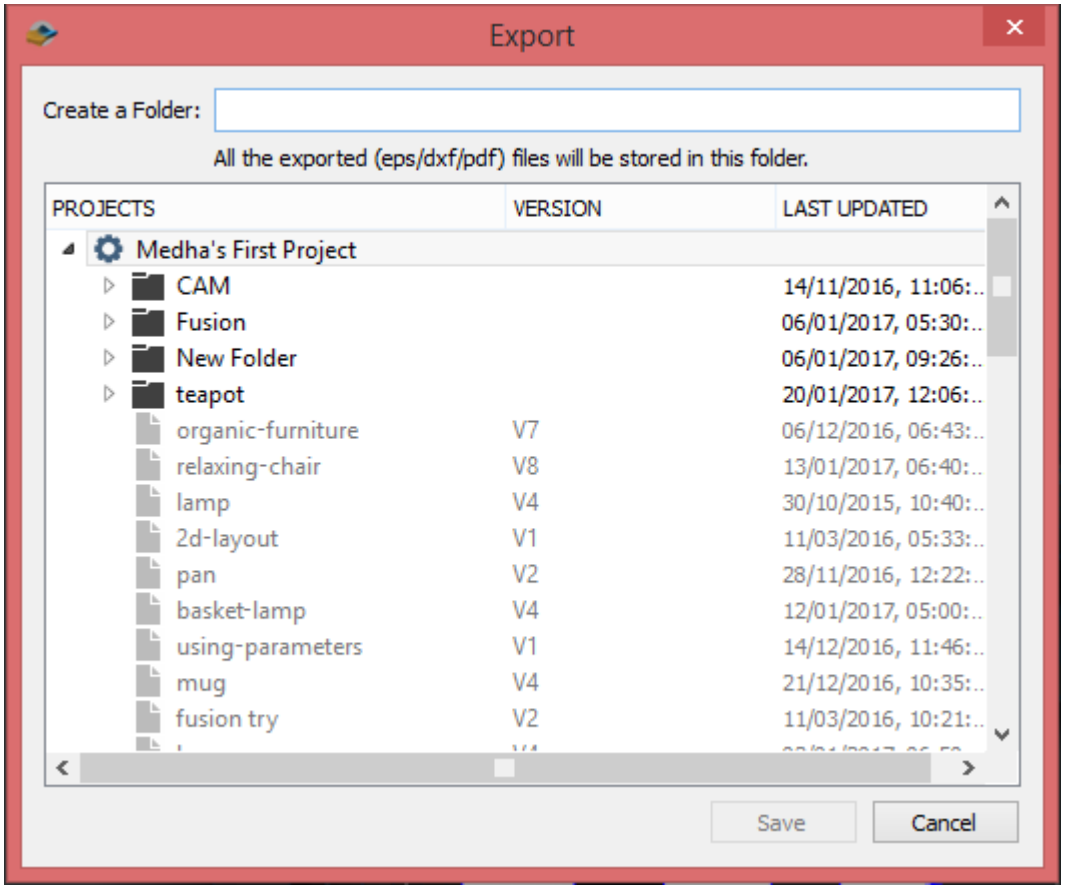
Use these files to print the slices you need for assembling your 3D model.

Export to Fusion Team

To export your plans to **Fusion team**, select one of the following types and then click on **Export to Fusion Team.**

Remember, that your plans are always exported in a new folder each time. And you need to name that folder. For eg: if you select **EPS**, create a folder by an appropriate name and all the eps files would be saved in this new folder.

In case, if you want to delete a folder, you need to go to Fusion Team website and delete from there. Folders cannot be deleted from Slicer for Fusion 360.

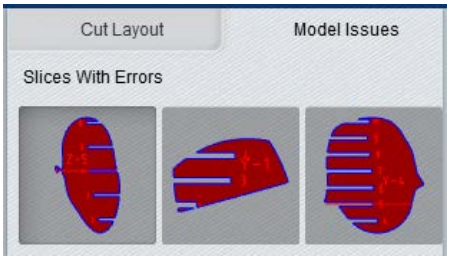


DXF files can be opened in Fusion 360. At present, though, we cannot directly open the DXF file, which has been exported to Fusion Team from Slicer for Fusion 360. There is a work around that can be used in that case.

1. Export the files locally from Slicer.
2. Upload them to Fusion Team from the website.
3. Now open Fusion 360. Right click on the thumbnail in Fusion Data Panel and click on ‘Open as Fusion Design’.

Troubleshooting with Model Issues

If there are errors with your model, use the **Model Issues** tab info to see what they are and how to fix them.

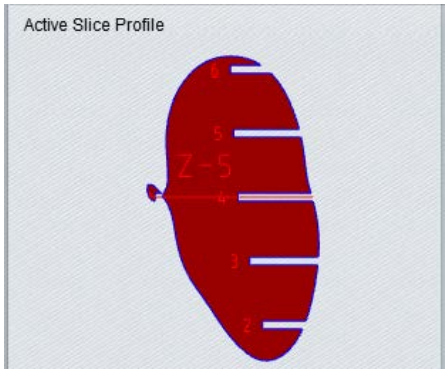


Slices with Errors

If there are slice errors, this section displays a small preview of the errors. Click any of these previews to highlight the slice in your model.

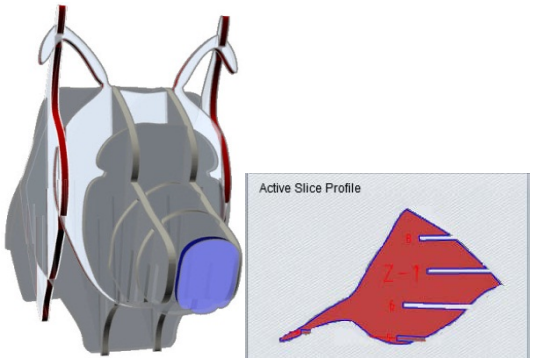
Error Explanation

This section displays explanations of the errors found in the preview selected in **Slices with Errors**.

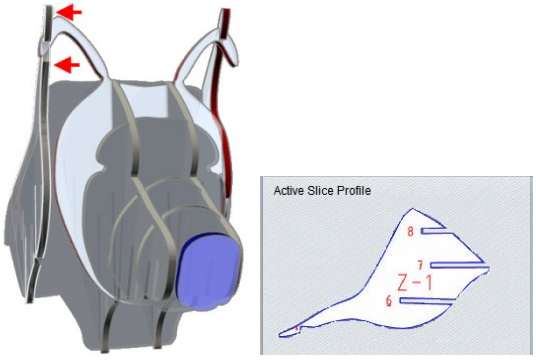


Active Slice Profile

This section displays a cross section of the selected slice. As you move a slice, the cross section updates to help you eliminate the errors.



Original



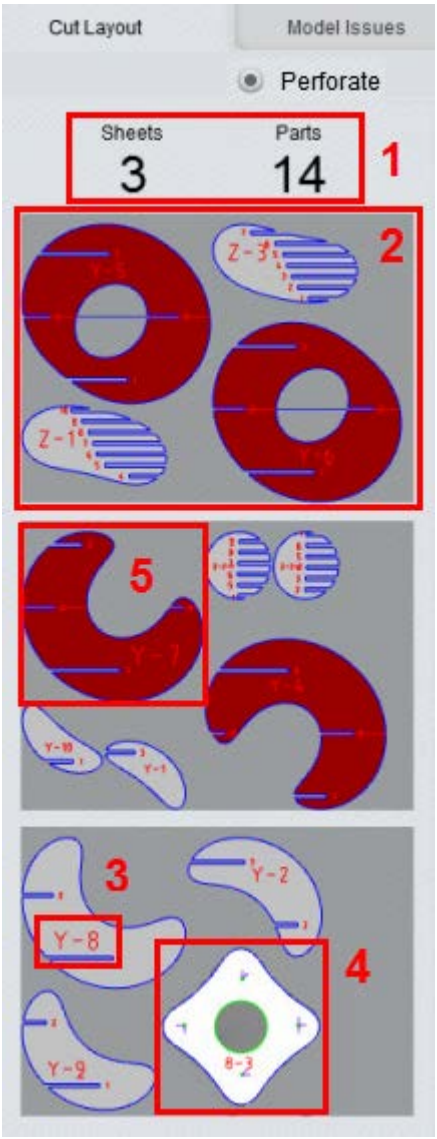
Moved to correct the error

Troubleshooting with Cut Layout

In the **Cut Layout** tab, see a preview of your model output. Use it to troubleshoot, so that in the end, your output is error free. No errors means, you'll be able to assemble your model.

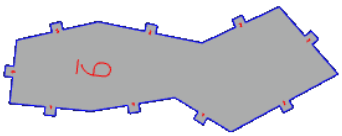
What is Cut Layout for?

Preview your sliced model and troubleshoot. It show where errors are. Then, use **Modify Form** tools and/or change **Slice Direction**, **Slice Distribution**, **Object Size**, or even **Construction Technique** to fix the problem.

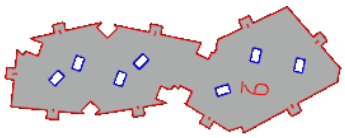


1. This gives a summary of how many sheets of material are needed to make your model, as well as the number of slices and parts.
2. **Cut Sheet**: This shows the printed sheets with the slices and parts spread across them. You cannot directly change where your model's parts are placed. Click a sheet to see a magnified view of it.
3. **Labels**: The hyphenated combos, such as Z-6-2, indicate the axis of the piece (Z), slice number (6), and part number (2 - if a slice has multiple pieces). The elements on a sheet are not in any order. Parts are automatically fitted to use as much of the sheet as possible.
4. **Colored Outlines**: *Blue* outlines are the model's *outside* edge. *Green* outlines are cuts made *inside* of your model to make hollow areas. *Yellow* outlines are the *scored guides* for placing the next piece during assembly.
5. **Errors**: Red parts have an error making your model difficult or impossible to build.

Perforate: *(Only available for folded panel projects.)* It provides ticks to indicate fold lines. However, at first, the perforations may look like holes.

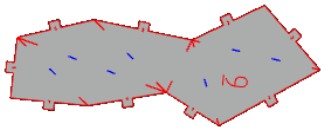


Perforate = OFF



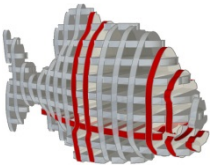
Perforate = ON

To see the fold lines, in **Manufacturing Settings**, tap  . In the toolbar along the bottom, decrease **Thickness** to a value around 0.005.

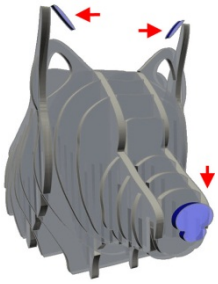


Thickness = 0.005

Troubleshooting Print Errors



Any sliced part in your model that are **red** have errors and can't be assembled.



Parts that are **blue** are detached from the model and will fall off.

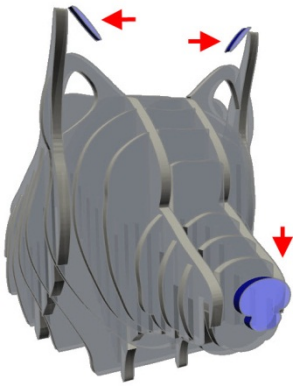
General Tips

- Resize your model slightly.
- Change the angle you slice your model at.
- Shrink your model to fit the material.
- Change your material to a larger sheet size (this may affect cost).
- Reposition the highlighted slice that is causing the error.
- Remove the highlighted slice that is causing the error.

Specific Print Errors

Unconnected pieces

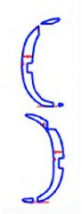
Chunks of your model are not attached.



These detached parts are blue. Typically, this happens to the first or last *interlocked* slice. There is no supporting structure to keep the slices attached. For a more complete structure, you can:

- Change the slicing angle.
- Change the number of cut slices.
- Change the construction technique to Stacked.
- Delete the disconnected slice. Be warned, if a slice has multiple parts, they will all be deleted.

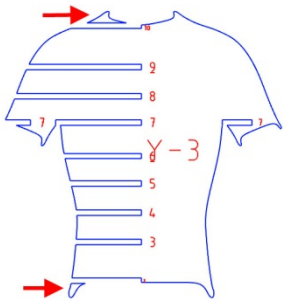
Part too small



Part of the model is so tiny it will be difficult to cut out or assemble.

- Enlarge your model.
- Change the slicing angle.
- Drag the affected slice over.
- Use **Thicken**.

Notches split part



Part of the model was cut off from the main section by a slice.

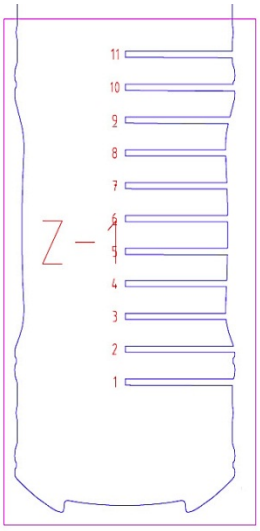
- Move the affected slice in small amounts until the separated pieces join together to form a whole part.
- Change the slicing angle.

Part too narrow

Part of the model is too thin and may break when printed. You may choose to print with this error, at your own risk. Stronger materials like plastic probably wont break.

- Use **Thicken**.
- Enlarge your model.
- Change the slicing angle.

Parts too large for sheet

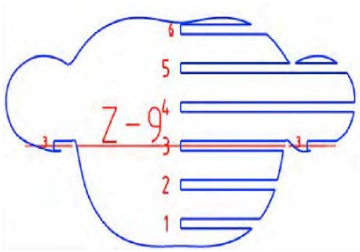


Parts of the model do not fit within the boundaries of the sheet.

If you are using *stacked* slices, Slicer for Fusion 360 will try to break the large part(s) into two pieces, with puzzle piece tabs to connect them. This does not happen with *interlocked* slices.

- Make your model smaller in **Object Size**.
- Make your sheet size larger in **Manufacturing Settings**.
- Change the slicing angle until the part fits.

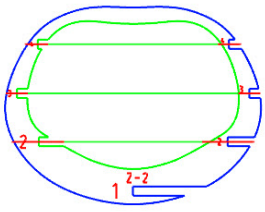
Multiple Notches



Part of an interlocked model has extra, unnecessary notches. In some cases, a notch become blocked (see left). When printed, you can't lock the two slices together.

- Change the slicing angle.
- Drag the slice slightly in one direction.
- Delete the affected slice.
- Change the construction technique to **Stacked Slices**.
- Resize your model slightly.

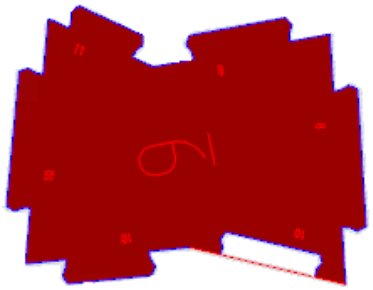
Notch intersects hole



A hollow area of your model has had a notch cut into it. The notch is not required for assembly.

- Change the construction technique to **Stacked Slices**.
- Change the slicing angle.

Joinery didn't fit



For Puzzle joint types, the edge length is too small and it fails because it can’t create a puzzle joint.

- Reduce the Tick Radius.

Border self-intersections

For Gear joint types, the adjacent gear tooth intersect with one another.

- Reduce the Tooth Radius.

Hole intersects

For Rivet joint types, the boundary is cut by the holes, due to them being too close to the sheet edge and their radius being too large.

- Reduce the Hole Radius.
- Reduce the Tick Radius.

Reinstalling Slicer add-in in Fusion 360

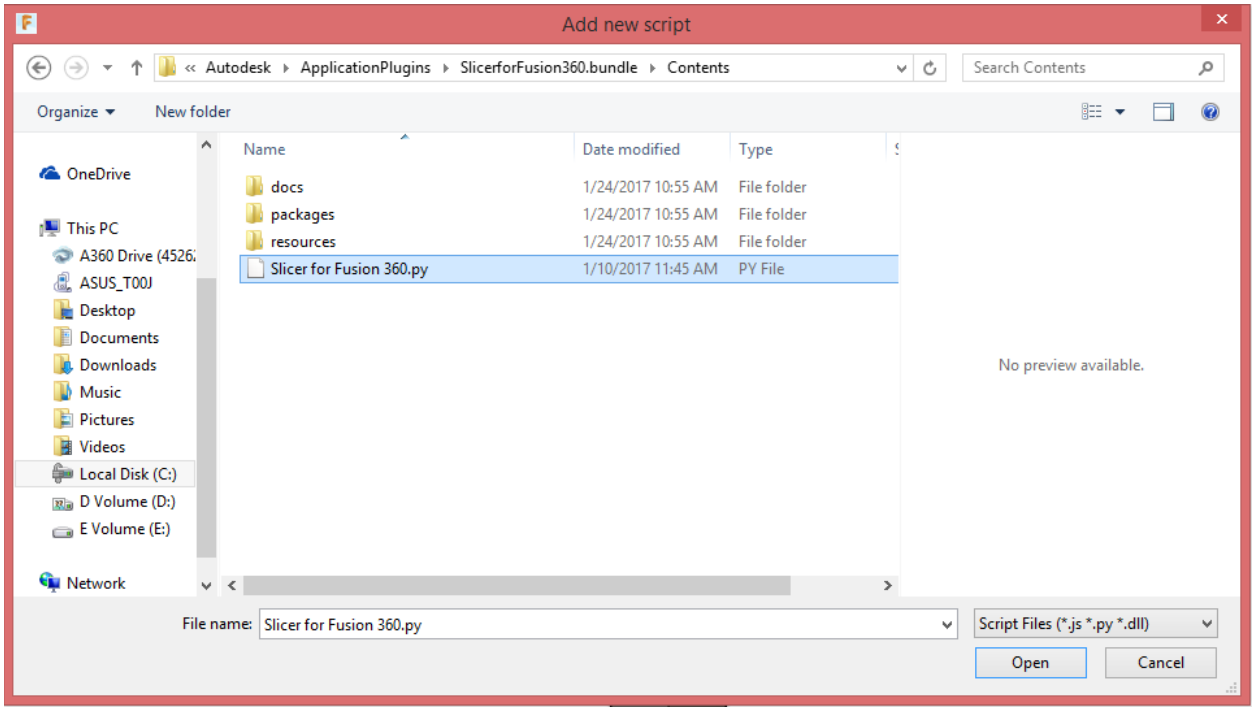
For Mac

If, for some reason, you remove the add-in from Fusion 360 and you want to add it again, then you first need to '**delete**' a file which resides in this path. **/Users/<'Your Account'>/Library/Application Support/Autodesk/Autodesk Fusion 360/<'Your A360AccountID'> .** Then try re-installing it again.

Note: 'A360AccountID' is either a 15 digit number or a 12 digit string. JSLoadedScriptsInfo is the file that records the add-in status.

For Win

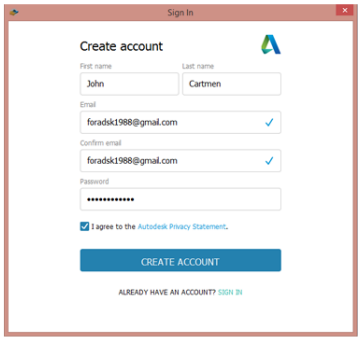
If, for some reason, you remove the add-in from Fusion 360 and you want to add it again, then you open add-ins options and click on + in add ins tab. Now follow the path **C:\ProgramData\Autodesk\ApplicationPlugins\SlicerforFusion360.bundle\Contents\Slicer for Fusion 360.py** and double click on the file. It should get added under 'Make' option.



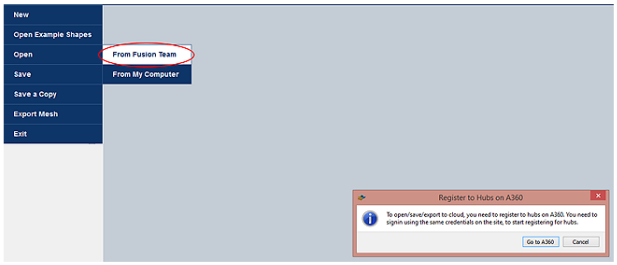
Unable to access Fusion Cloud from Slicer

If you create a new account in Slicer, then you need to take an additional step of registering on A360 hubs. Until you register to hubs, you won't be able to open/save/export any files to A360 (cloud) from Slicer application.

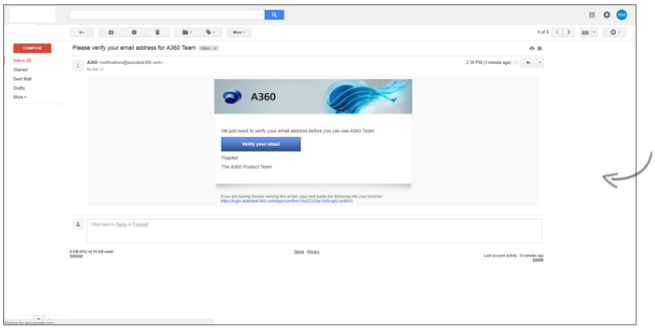
Create account in Slicer



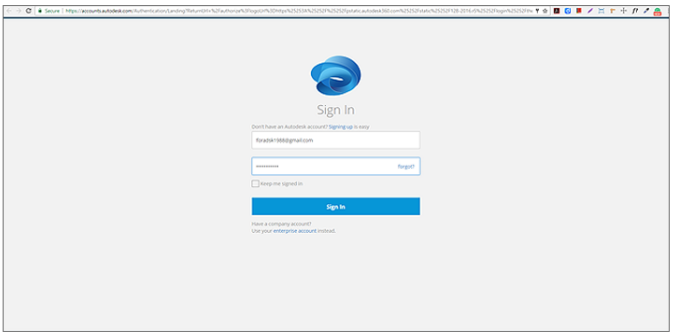
Cannot open cloud (Fusion Team)



Verify your account



Go to A360 website and follow the steps



1. Create a new account in Slicer for Fusion 360.
2. Now if you try to open/save/export something from/to Fusion Team, you won't be able to access it. You will get a message mentioning the same.
3. Click on the button 'go to A360'.
4. Signin with your newly created credentials.
5. You will be asked to register for hubs. Please proceed.
6. Check the verification mail in your inbox. And click on 'Verify' to verify your account.
7. You will be directed to A360 website and would be able to access the hubs now.

8. Now, in Slicer, sign in again.
9. You will be able to access the cloud now, if you try to open, using ‘Open from Fusion Team’.

If you are a Tinkercad user or 123D user, you would still need to register to hubs on A360. You can follow the above steps to do the same.

If you have an existing account in Fusion 360, you will have registered to on A360 hubs automatically, so you won’t face any problem.