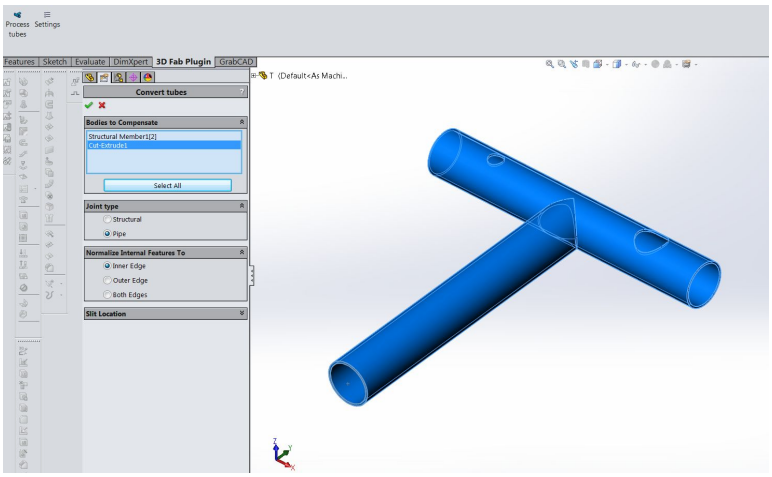
# 3D Fab Plugin Manual

Version 2.0 — March 19, 2019




# Table of Contents

# 1. Plugin Purpose

The 3D Fab Plugin for Solidworks serves two purposes:

1. It converts solidworks weldment components into geometry that can be cut on a 4-axis laser cutter.
2. It unwraps those weldment components and automatically saves a .dxf file that can be fed into FabCreator to create a file so the tubes can be cut.

It works similarly to Solidworks' built in "Convert to Sheet Metal" function, but it allows the user more control over how features get compensated so that all of your laser cut tubes fit together exactly as expected.

## 1.1. About Weldments

Weldments are a way of building a structure in a Solidworks Part that will provide the user with separate bodies that can be output to a file. The user can take those parts and make a cutlist for producing the tubes, which takes most of the guesswork out of the fabrication.

It is convenient to use weldments, because as the user changes the base sketch (the frame of the weldment), each structural member will update with it. This allows for easy iteration of designs and revisions. The purpose of this plugin is to convert a weldment into a file that the laser can interpret. The nuances of the plugin are described in this document, but first, the workflow will be explained. This will give the user an idea of how to best design their parts for cutting on the FabLight.

## 1.2. About Solidworks

Solidworks is independent commercial software that is upgraded regularly. Unfortunately, files are often not compatible between versions, and instructions for how to do things in Solidworks can change. This manual is written for Solidworks 2015. If something is not working in your version of Solidworks, you can try searching online for an answer to your issue. You should also feel free to contact 3D Fab Light service (service@3dfablight.com) for help resolving the problem (or if you solved it, to let us know what you did). We aim to improve our documentation.

## 1.3. Weldment Tutorial

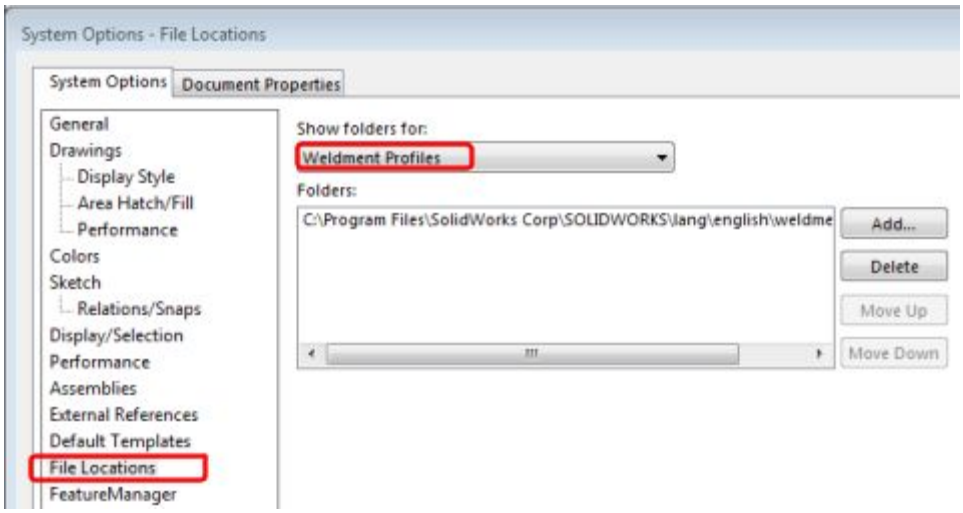
Describing how to use Solidworks weldments is beyond the scope of this document. There is a tutorial built into Solidworks by clicking Help > SOLIDWORKS Tutorials. Other tutorials can be found online.

## 1.4. Weldment Library

There is a FabLight weldment library that includes a number of square and round tube profiles. This library is not required, but might be helpful, especially for square tubes.

**NOTE**: These profiles were made with Solidworks 2015, and may not open properly in newer or older versions due to Solidworks incompatibility issues.

After downloading and unzipping the library, you can install the weldments into your weldment folder manually. To locate the folder, go to Tools > Options, then pick “File Locations” in the list and “Weldment Profiles” from the drop-down menu. The two folders in the library, FabLight Round and FabLight Square, can be moved into that location. Restart Solidworks to load the new libraries.




Creating your own custom profiles is easy -- they are just 2D sketches showing the outline of the tube that will be created, that are saved into the Weldment Profiles location. If our library is giving you problems, you can recreate any weldment easily.

**NOTE**: If creating a custom Weldment library folder, it needs to have at least two levels of folders for Solidworks to recognize it. For example, even though the round tube profile is the same for all materials, an extra “Any Material” folder must be in the path (e.g. FabLight Round > Any Material > 1 inch round, .035 inch wall.SLDLFP).

## 1.5. Corner Radius

The most important part of our weldment profiles is the corner radius, which affects the overall unwrapped perimeter of a square tube, so it is important to match it to the tube size you're using. The corner radius value depends on the wall thickness of the square tube, but is independent of outer diameter and material type. The following are standard values we have set in our Weldment profiles:

| Wall Thickness (in) | Corner Radius (in) |
|---|---|
| 0.065 | 0.07 |
| 0.120 | 0.13 |
| 0.180 | 0.25 |
| 0.250 | 0.38 |

Note that if you create your own sketch for a square or rectangular tube, the inside and outside corner radii must be concentric for Solidworks to convert the tube to sheet metal for flattening.

## 1.6. Multi-Profile Weldments Best Practice

One thing to note is that if you have different size weldment profiles within your structure (e.g. different diameters, mixed round and square, etc), you can create separate structural member groups. Note that the separate groups will not terminate (cope) into each other. The user must do this.

Normally the user will use trim/extend feature found in the weldments tab, HOWEVER, the plugin does not work with this feature -- the weldment will not process with this feature. Therefore, **the user needs to use cut/extrude with an "up to body" end condition**. Be sure to not merge results and select the correct body. If the user would like to make a pipe for fluid flow, etc., now is the time to extrude/cut the orifices. From here you can run the plugin, as described in this document.

This idea will be revisited later in context. For now, it is something to keep in mind when creating weldments.

# 2. Installation

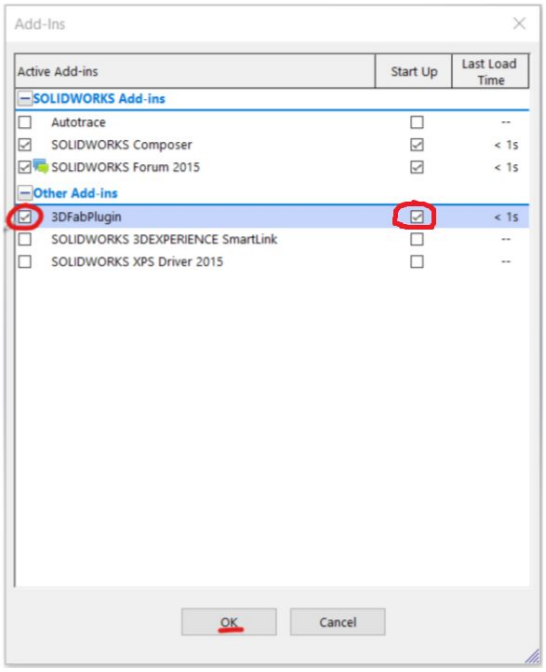
## 2.1. Download and Install

Download the plugin installer and install it. Also download the Weldment Library and install it in your Weldment Profile directory (see Weldment Library in section 1 for more info). The library includes common sizes of round and square tubes up to 0.120” wall and 2” diameter round and square.

As with most design, it is important to keep in mind the capacity of the machine used to make your parts. The FabLight rotary chucks can cut up to 2” square and round tubes, up to 54” long (standard) or up to 80” long with the extension.

### 2.1.1. Activate Plugin Add In

After installing the plugin, you will need to go to Solidworks Tools > Add Ins… and under Other Add-ins, check both boxes for 3DFabPlugin, then hit the OK button. The box on the left is to activate the plugin, and the one on the right is to start the plugin on Start Up of Solidworks.



### 2.1.2. First View of the Plugin

The plugin now appears in the Toolbar, much like Weldments. If the tab is not visible after activating, try right-clicking on another tab and then choosing 3D Fab Plugin from the list that appears. Once the tab is visible, selecting it reveals two options: Process tubes and Settings.


### 2.1.3. Licensing

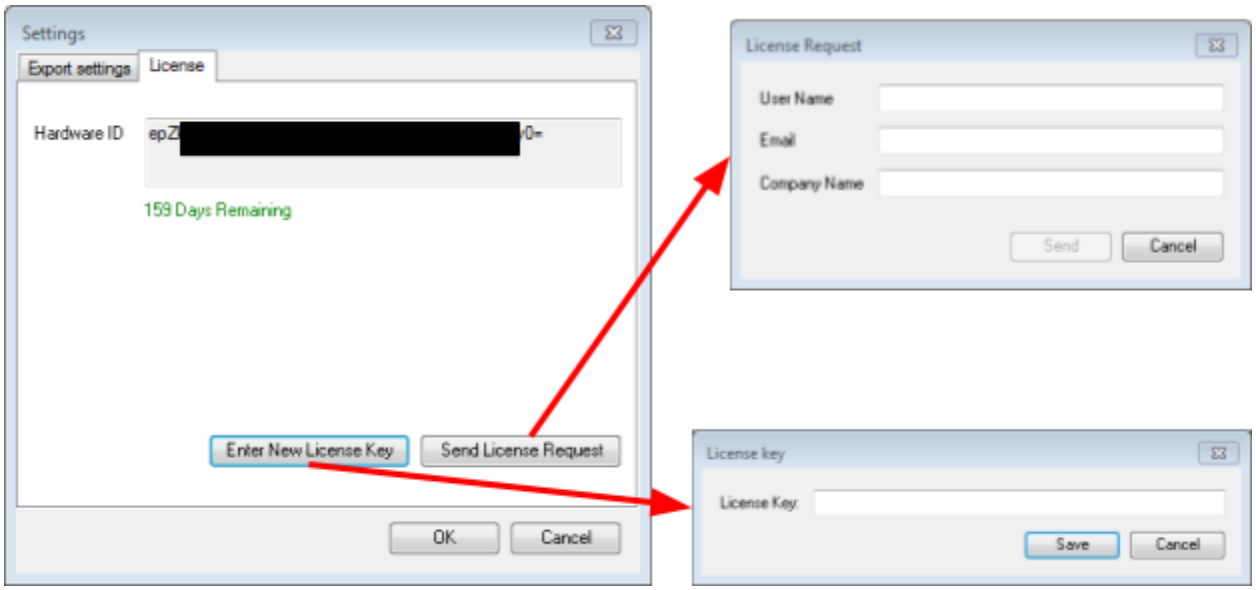
The 3D Fab Plugin is designed to be free to use, but requires registration. The plugin will work for up to 7 days without requiring registration.

Registration is done by clicking Settings in the 3D Fab Plugin toolbar. This brings up the Settings window, as seen on the image below. Clicking on the the License Tab and pressing the “Send License Request” button opens another window asking for name, email, and company name. Customers can install it on as many copies of Solidworks as you’d like. We are just trying to monitor the number of licenses, and non-customer users.

> **NOTE**: If you get a warning, “Please connect to the internet” after clicking “Send License Request,” you will have to email the HardwareID directly to swplugin@3dfablight.com using another email method.

You will receive a license key sent to the email provided. Currently this is manually generated, so it might take a few hours to receive a reply. Please plan ahead for the 7 day trial period.

When you receive the key, copy and paste it into the window that opens when the click the “Enter New License Key” button (see image below). This license key is good for 1 year, then the user needs to repeat the same process.




## 2.2. Plugin Inputs

The plugin can handle inputs from:

1.) Multibody Solidworks weldment generated from a skeleton sketch.
2.) Single or multibody tube structures generated using SW extrusions.

**NOTE**: The plugin cannot currently handle STEP files or other types of “featureless” inputs. You will have to recreate the geometry in Solidworks weldments.

## 2.3. Plugin Outputs

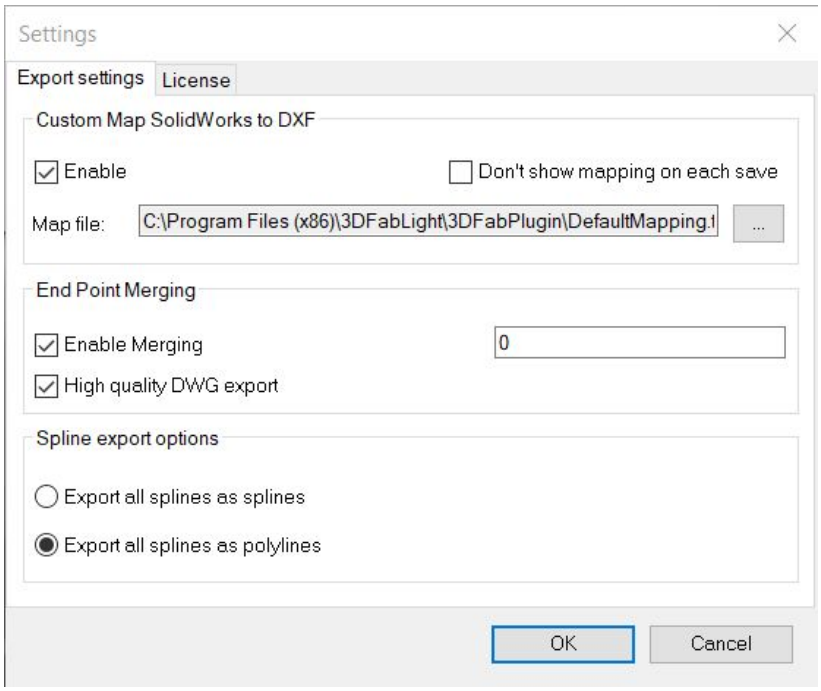
The plugin outputs the following for each tube in the input file (more specifically, for each tube selected):

1.) Compensated Geometry in a SW part file
2.) Unwrapped tube DXF file, showing the flat pattern for the unwrapped tube
3.) SW Assembly file showing how all the compensated parts fit together

Even if you only run one tube, the plugin will still generate an assembly with only that part in it. See section “3.2.1. Output Files” for more details on the exported folder and files.

## 2.4. Plugin Export Settings

Clicking on the Settings button on the 3D Fab Plugin toolbar brings up the Settings Window. Besides the License tab (see 2.1.2 Licensing above) there is the Export settings tab. When you select your settings and click “Ok”, your selections will be remembered every time you open Solidworks until you manually change them again. When you first install the plugin, it is recommended you go into its settings and set them up as shown below. When you accept the settings by clicking “OK”, the system will remember them until you manually change them again.




By default, the plugin enables Custom map Solidworks to DXF. When the plugin is installed, it points to a map file by default that places the cut geometry on Layer 0 and the Bend Lines on Layer “bend_lines”. This file can be ignored by unchecking the enable box, or can be modified by pointing to a different map file.

End Point Merging can help clean up the exported .dxf file and is recommended to be enabled.

It is also recommended to select “Export all splines as polylines,” as the FabLight works better with curves that are designed using arcs rather than splines.

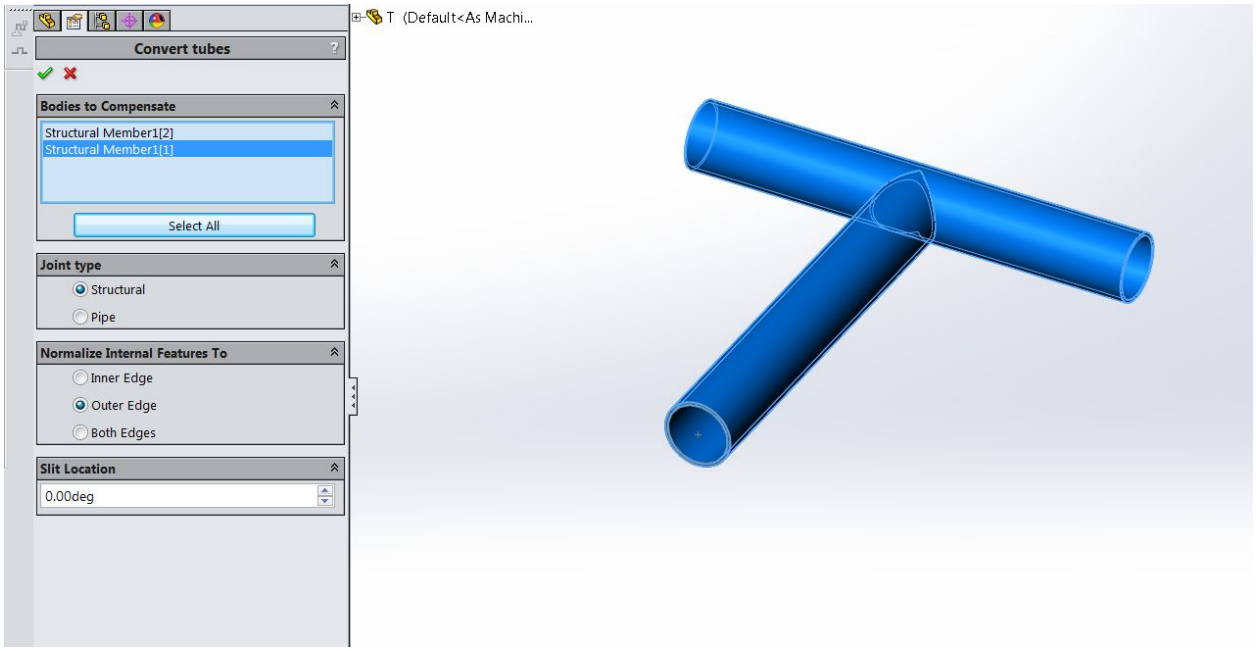
# 3. Using the Plugin

## 3.1. Plugin Options

Now that the plugin is setup correctly and licensed, we will discuss the available compensation options and how to select them.

When you click the “Process Tubes” button from the 3D Fab Plugin toolbar, it launches a sidebar. First select the bodies to compensate. You can select them individually or press the “Select All” button to automatically select every tube.

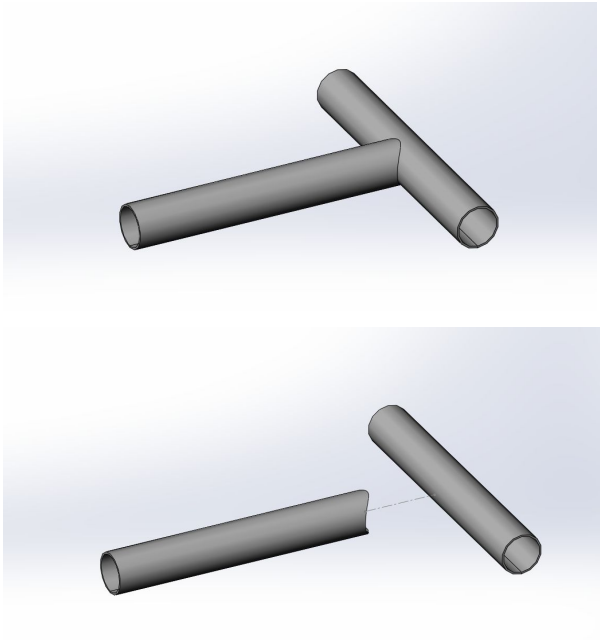
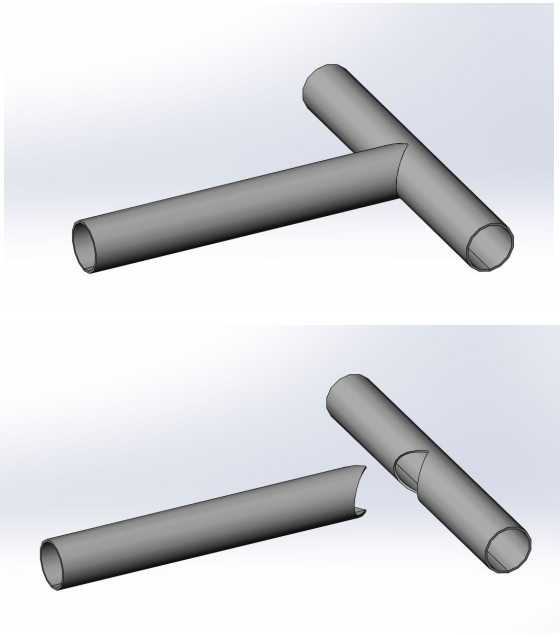
The two main options to be aware of here are how the features on the tubes will be joined and normalized.




### 3.1.1. Joint Type

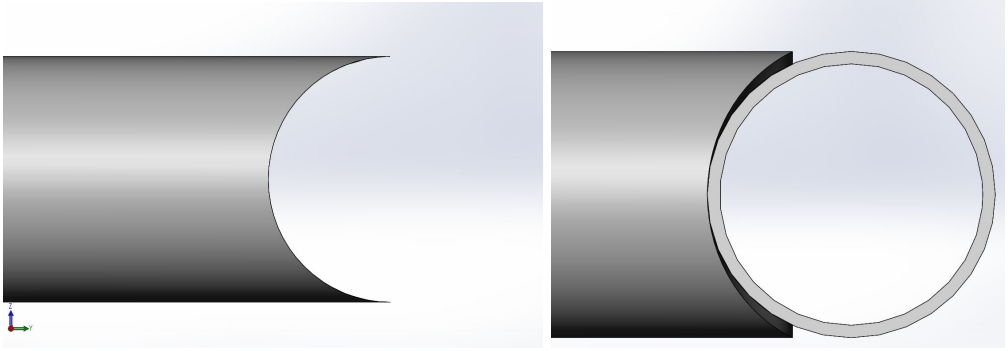
This determines how the ends of the tubes will be compensated, either “Structural” or “Pipe”.

| | |
|---|---|
| **Structural joints** compensate the edges of the tubes to the inner diameter (ID). This means the tube can only contact the other | **Pipe joints** only compensate the edges of the tubes to the outer diameter (OD). This means the tube can only contact the other tubes on it’s outer edge. In order for this type |

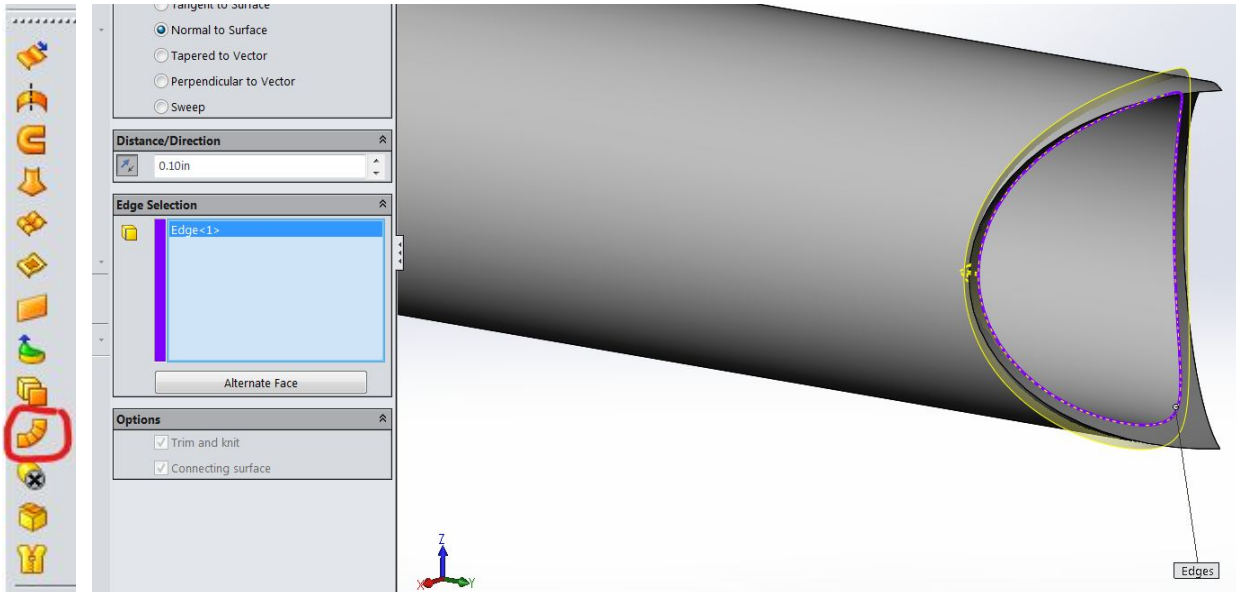
| | |
|---|---|
| tubes on it's inner edge. It also maintains the section of all tubes (no added cuts). | of compensation to mate up correctly, this requires a corresponding cut to be placed into the wall of the mating tube. This leads to a "fish mouth" type joint. It might be desirable if you need fluid to flow through your joint or for aesthetic reasons. |
|  |  |

If the plugin fails to process, or out of curiosity for how it works, you can replicate the results manually by following the example for a Structural joint below:

We are going to focus here on the tube that has the "fishmouth" end cut, to show how to compensate that cut manually. In the left image below, you can see that the tube comes to an infinitely sharp point at both the top and bottom. This feature is impossible to cut on the FabLight, because the cutting head doesn't rotate, and therefore can't cut the tube at an angle. It can only cut perpendicular to the tube (right image showing compensation to the ID).



The first step in compensating this end is to create a ruled surface to correspond to the ID edge. From the surfaces toolbar, select the ruled surface icon, shown circled in red in the left image below.

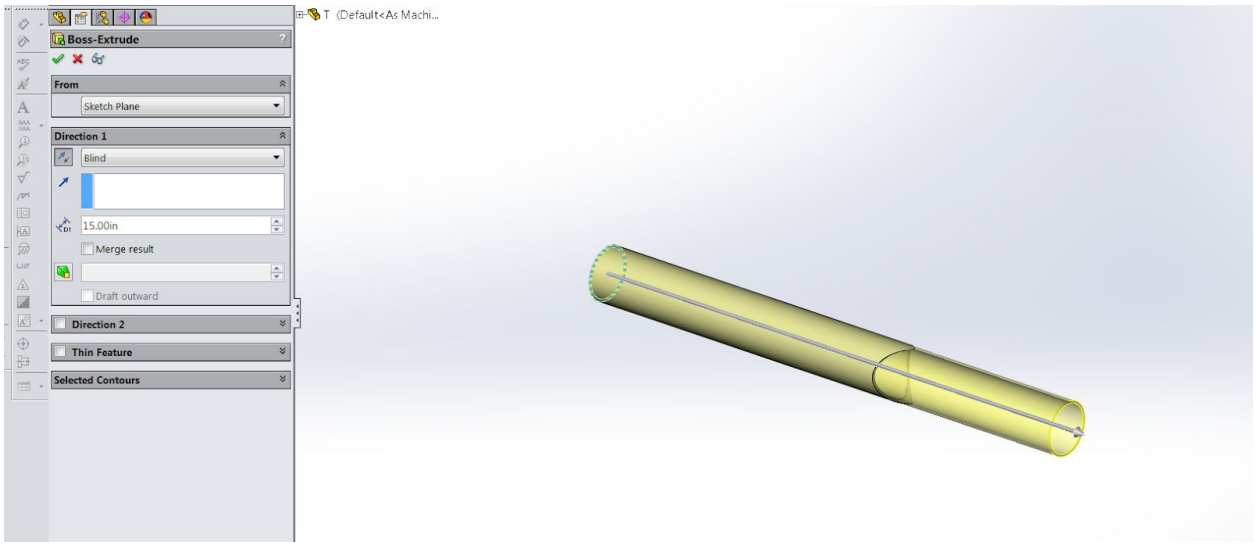



When the screen launches, select the inner and toggle the distance, direction, and face so that it matches the image above.

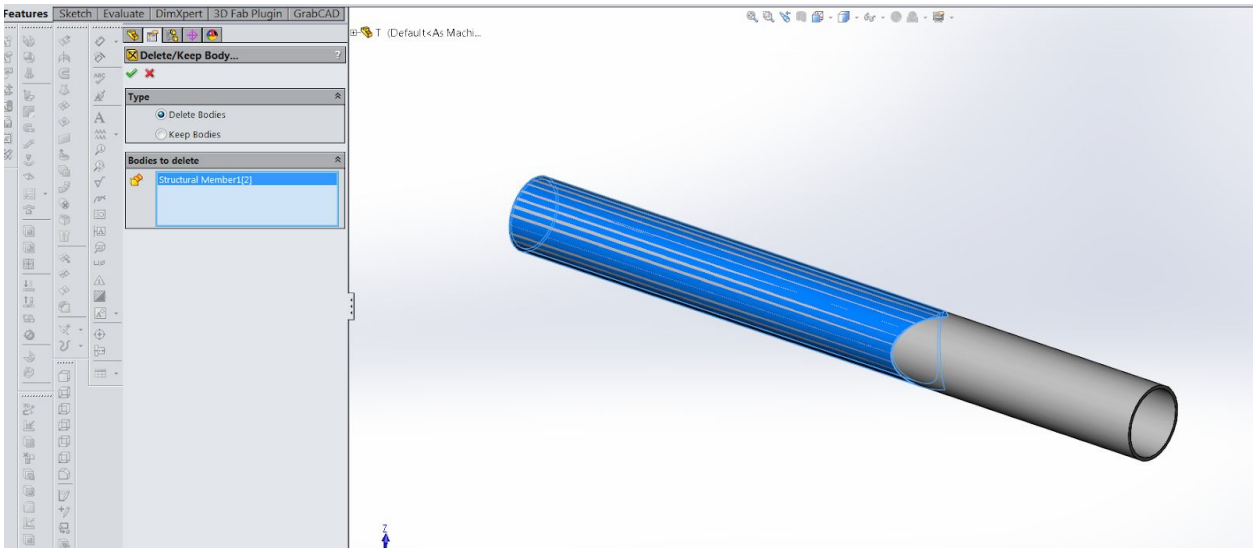
| **NOTE:** The distance should be greater than or equal to the wall thickness of the tube. |
|---|

Click the green arrow to accept the surface.

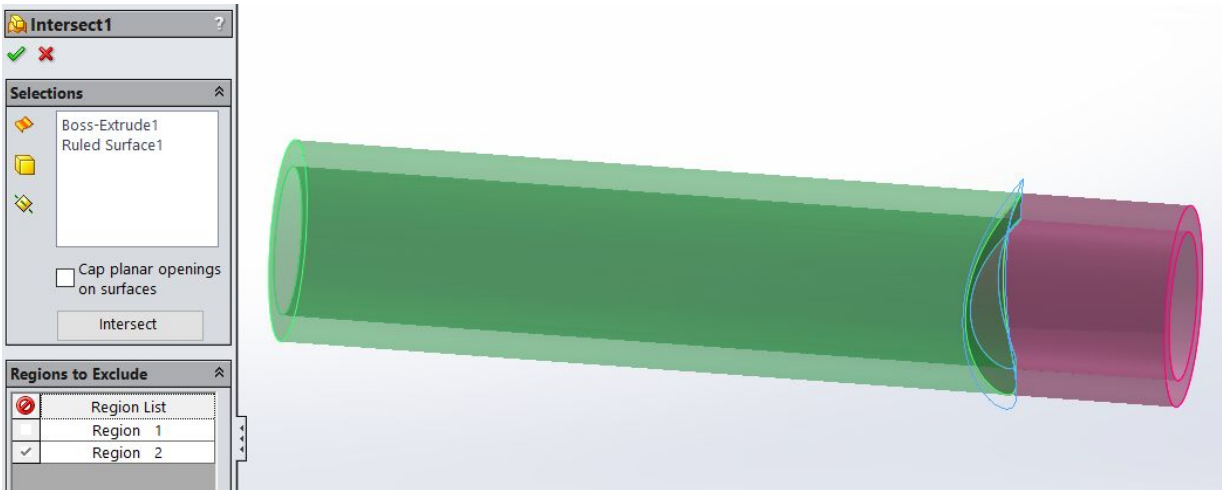
Now, for ease of modeling, select the flat face on the opposite end of the tube and sketch two circles that match the ID and OD of the tube. Extrude this sketch so that it encompasses the entire tube, plus a little extra, as shown below. Be sure that the “Merge Result” box is not checked.



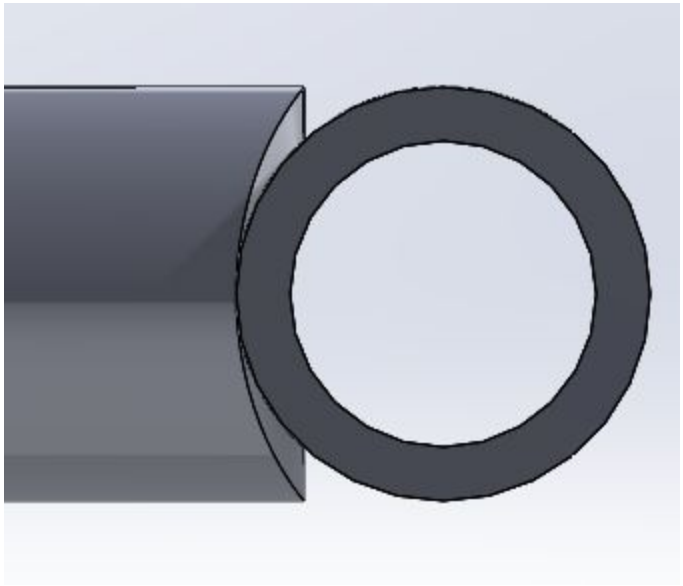

Now go to Insert → Features → Delete/Keep Body. Select the original weldment body and delete it, as shown below.



Now go to Insert → Features → Intersect. Select the extruded body that is left on your screen and the ruled surface from the ID edge. Check the box to delete Region 2, as shown below, and check Merge Results at the bottom.



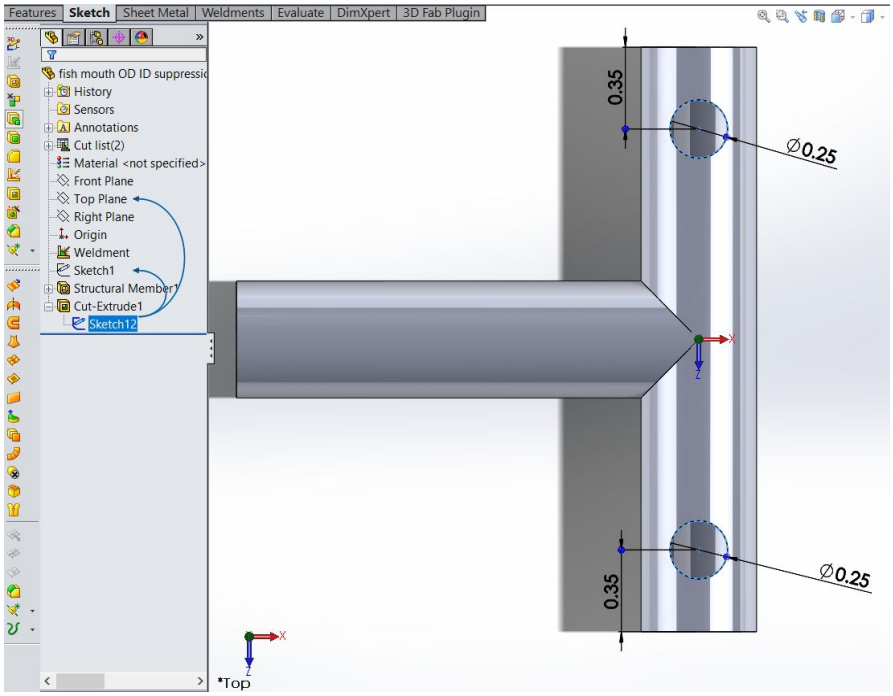

Now the tube is normalized to only the ID.



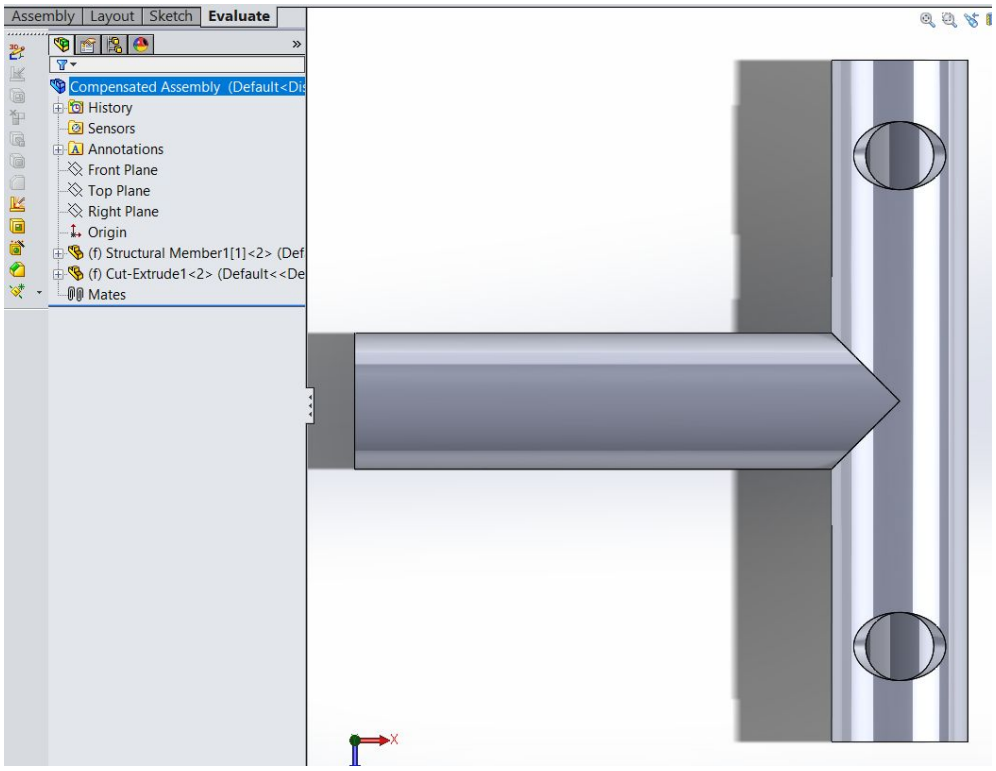
## 3.1.2. Normalize Internal Features

The second drop down menu is labeled “Normalize Internal Features To” and it is used to control how all cuts that are not on the edges (e.g. “internal” to the tube) are compensated. The options are compensation to Inner Edge, Outer Edge, or Both Edges. The example below illustrates the difference between the options.

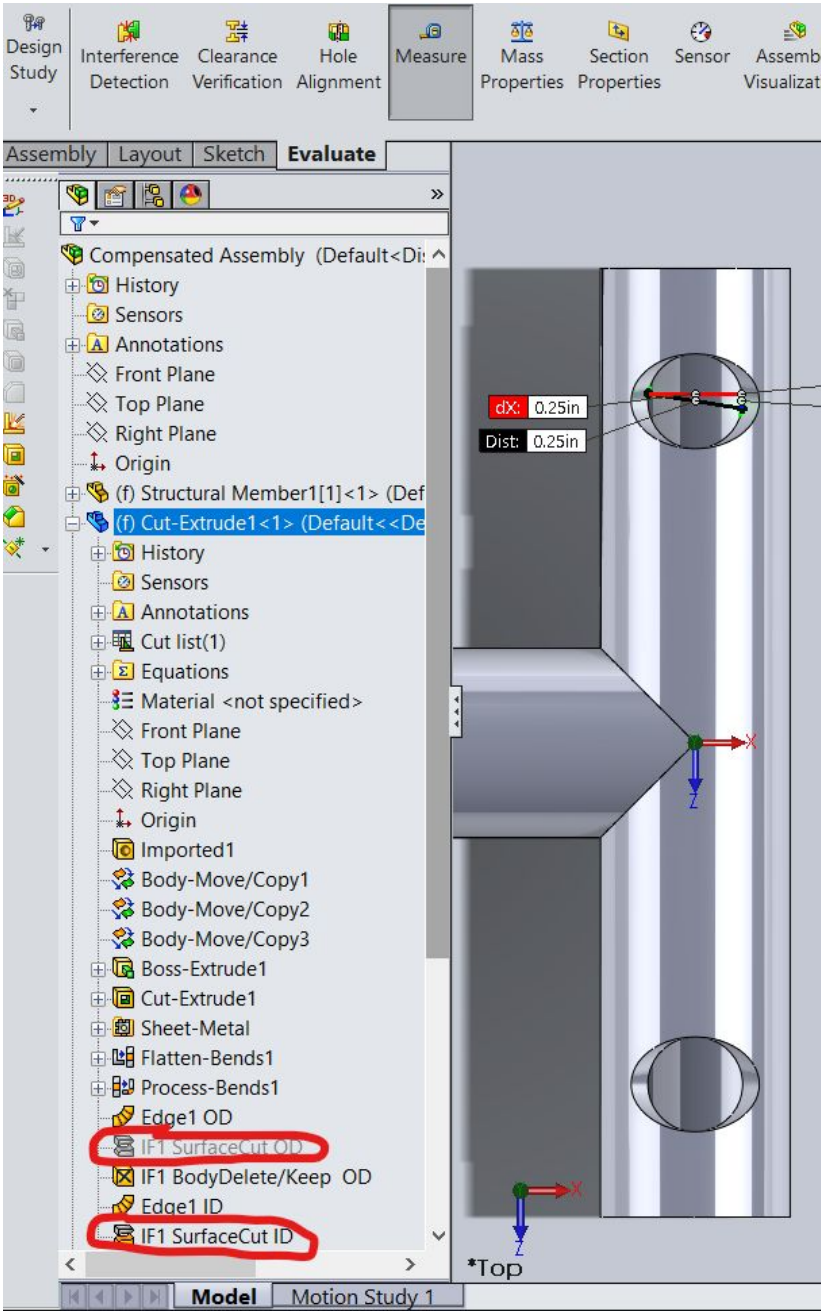
Two identical 0.25” diameter circles have been cut-extruded in one of the structural members.




After processing these tubes as Joint Type: Pipe, and Normalize Internal Features To: Both Edges, we get the resulting Compensated Assembly below.



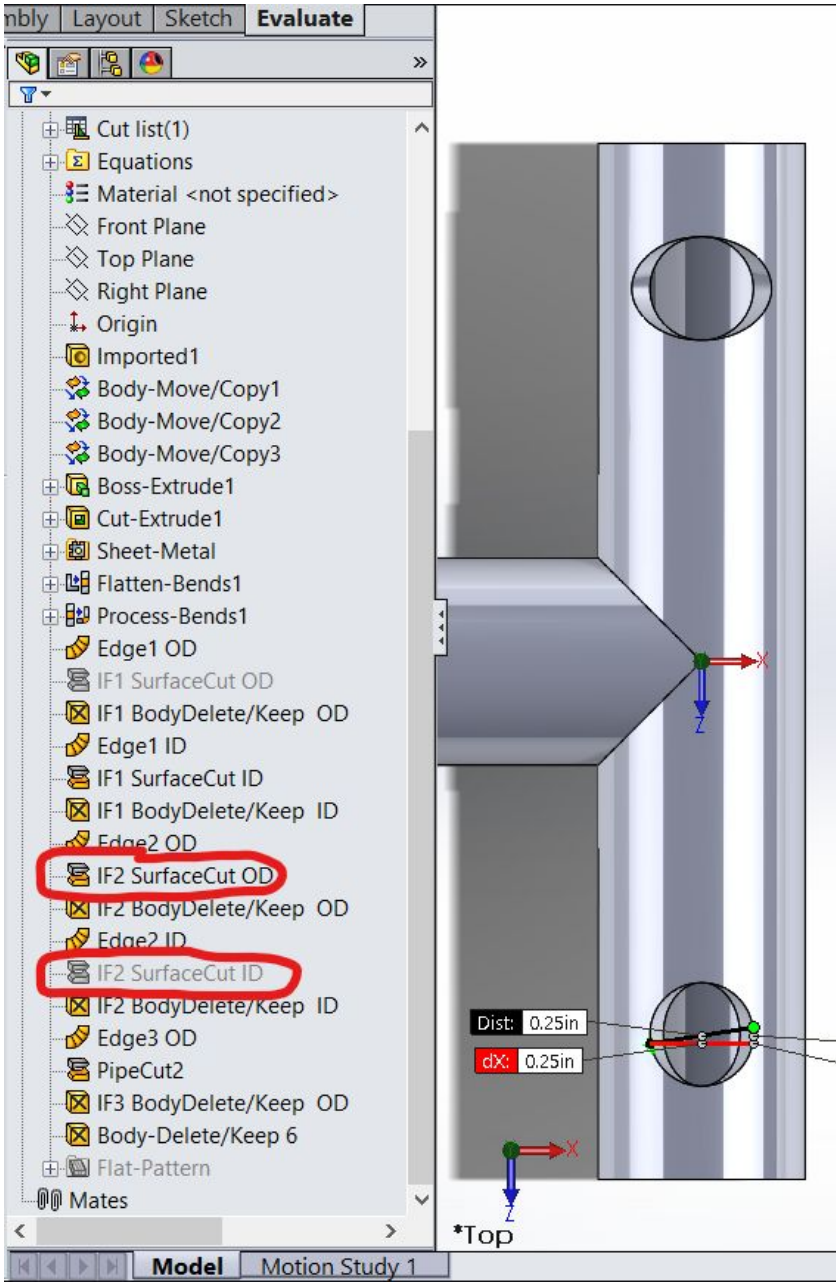
Note the cut-extrusions show different shapes — circle for the inner diameter (ID), and elliptical for the outer diameter (OD). This is because the plugin outputs two Surface Cuts in the Compensated Assembly for each internal feature, one for the OD and one for the ID. When you Normalize to either Inner Edge or Both Edges, the OD Surface Cut is suppressed by default, meaning the ID is the edge that retains the original 0.25” measurement as shown below.



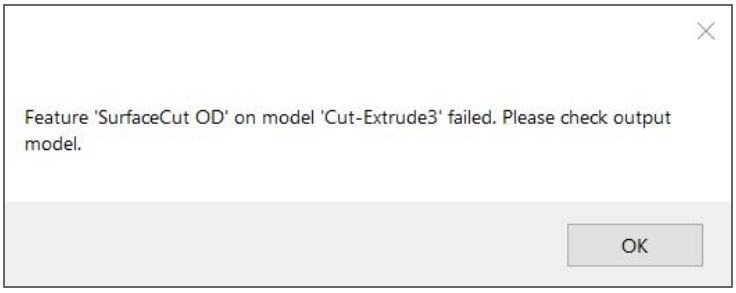

In order to make it so that the OD retains the original 0.25” instead, you have to switch the “IF SurfaceCut ID” to be suppressed and make the “IF SurfaceCut OD” unsuppressed, as applied

to the bottom extrusion below. Notice how switching the suppression also switches the circle/elliptical shaping, such that the circle-shaped edge is the one that is unsuppressed and retains the expected dimension. The elliptical-shaped edge is the one that is suppressed and does not match the original extrusion dimension.

**IMPORTANT NOTE**: If you make manual adjustments like this to the processed part file, you need to Flatten the part in the Sheet Metal toolbar and resave the DXF manually.



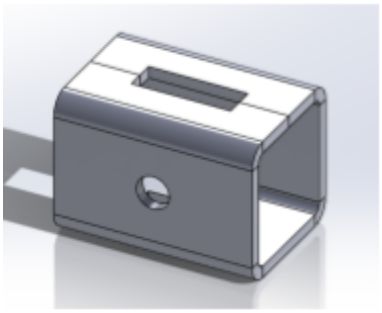
**NOTE:** If you encounter the error below, the plugin is simply stating that for some reason it could not create SurfaceCuts for both the OD and ID, so you will not have a choice of which one to suppress. Click “Ok” and check the output file for any irregularities. If the error does not appear to have affected your desired design, you can ignore it.




### 3.1.3. Slit Location

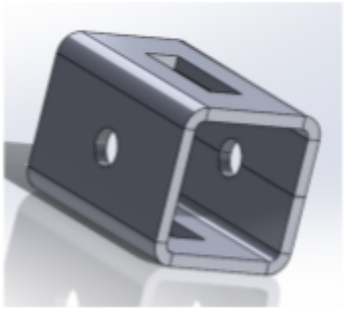
The final option in the setup is the slit location. This is an angular position between 0 and 360 degrees that determines where the tube is cut before it is unwrapped. By default, this value is set to 0. This works in most cases, but if it cuts through features awkwardly or if the plugin does not run correctly, you can try adjust the slit location and running again. This is important because any features running along the slit will be split both in the DXF and in cutting and engraving. The slit should be moved away from any critical features to avoid any interruptions in the cut.

**SQUARE TUBE NOTE**: You should only rotate the slit location on a square tube by factors of 90°. When you use FabCreator to process a DXF into a FAB file for cutting, FabCreator will assume the bottom of the unwrapped flat pattern is in the middle of a square tube. So if you rotate the slit by a value other than a factor of 90°, it will shift your design feature locations in an unintended manner. The example screenshots on the next page illustrate how a 0° (default), 20°, and 90° slit location value impacts the FabCreator wrapping and job preview of a square tube with cut features on all faces.

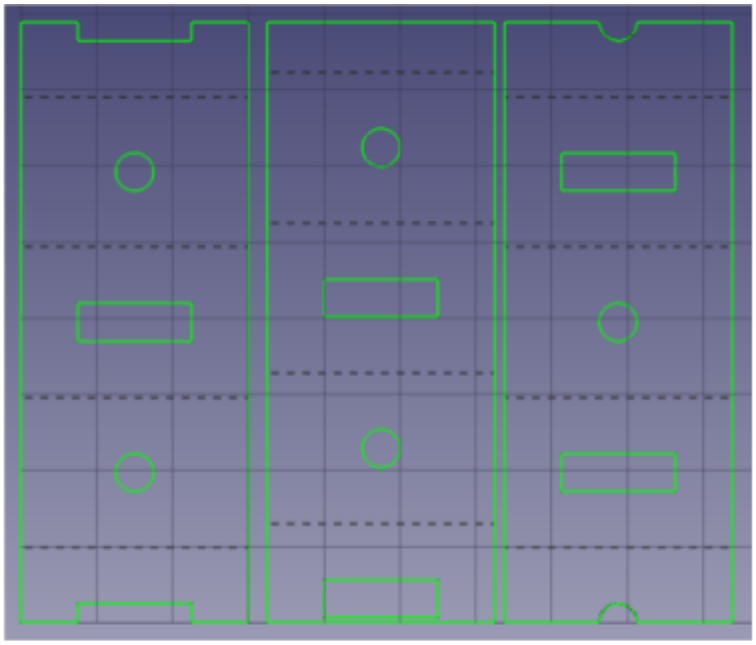


0° - slit intersects rectangle

20° - slit shifted off of rectangle



90° - slit intersects circle on adjacent face

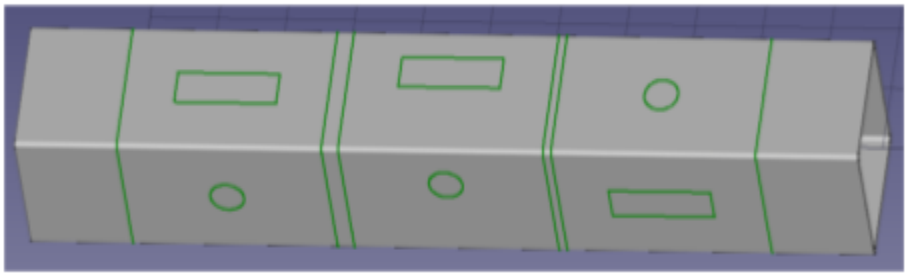


0° - bottom of unwrap starts at rectangle intersection

20° - bottom of unwrap starts close to rectangle

90° - bottom of unwrap starts at circle intersection

wrap starts with bottom at center of tube in final job preview



0°

20° - features are shifted off center

90°

## 3.2. Running the Plugin

### 3.2.1. Output Files

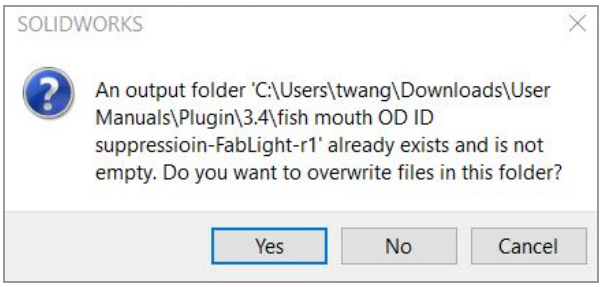
After you select tubes, compensation options, and slit location, click the green Check Mark to run the compensation routine. When it's finished, a new assembly called "Compensated Assembly.sldasm" will be open, made up of the processed part files that are automatically created.

These files are in a folder that is automatically created in same directory as the part being processed. This folder uses the part file's name to create "PARTNAME-FabLight-r1". Inside this folder are a number of files. For a two-member weldment, there would be:

- Compensated Assembly.sldasm
- StructuralMember[1].dxf
- StructuralMember[1].sldprt
- StructuralMember[2].dxf
- StructuralMember[2].sldprt

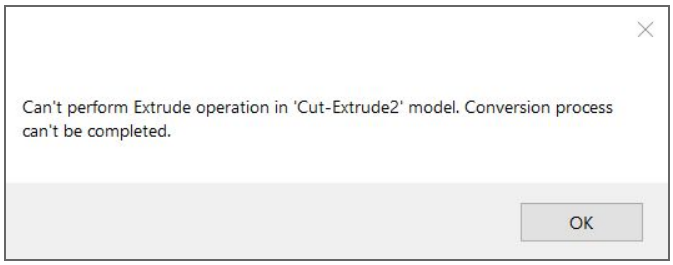
The "Compensated Assembly.sldasm" is the file that is open. The two DXF files in that folder are the flattened output that will then go to FabCreator to make the FAB files.

If you process the same file through the plugin more than once, you will be asked if you want to overwrite the existing output folder. If you select "No", the plugin will continue to process but save the output to a folder with the end suffix "-r2", and so forth for any future iterations.

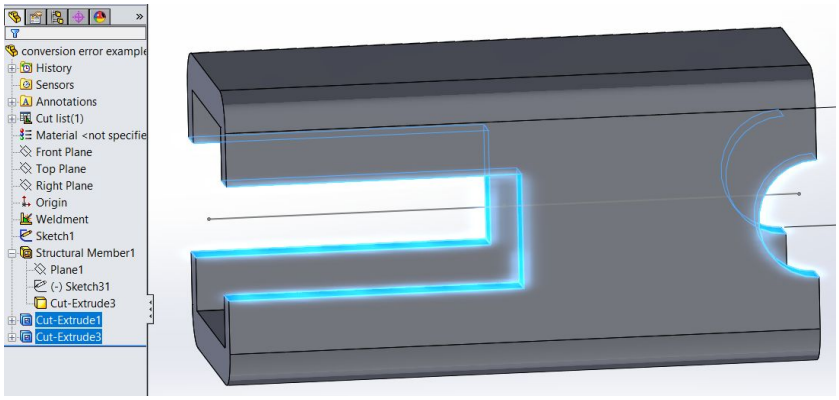


## 3.2.2. Conversion Process Error

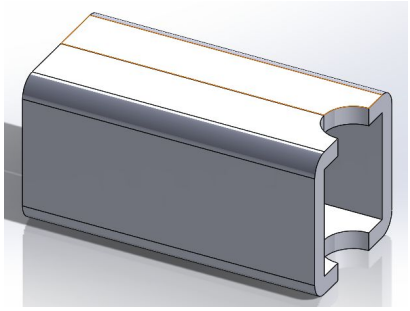
There is currently a known bug with tubes that have cut-extrusions through their base end. If you see an error like the one below, use the following steps for our temporary workaround.



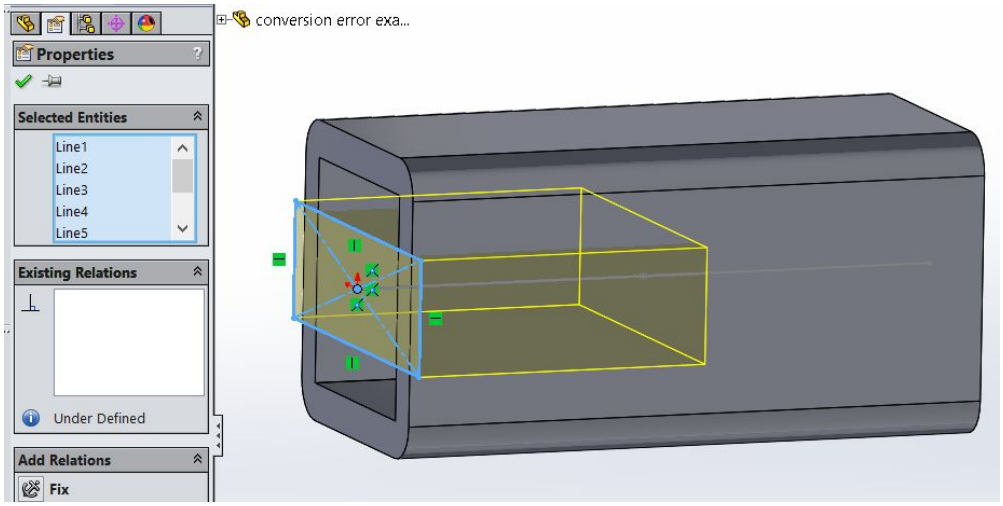

In this example below, we have a square tube structural member made from a Factory weldment profile. There are cut-extrusions on both ends of the tube, a rectangle sketched on the end face of the tube, and a circle sketched on a side face.



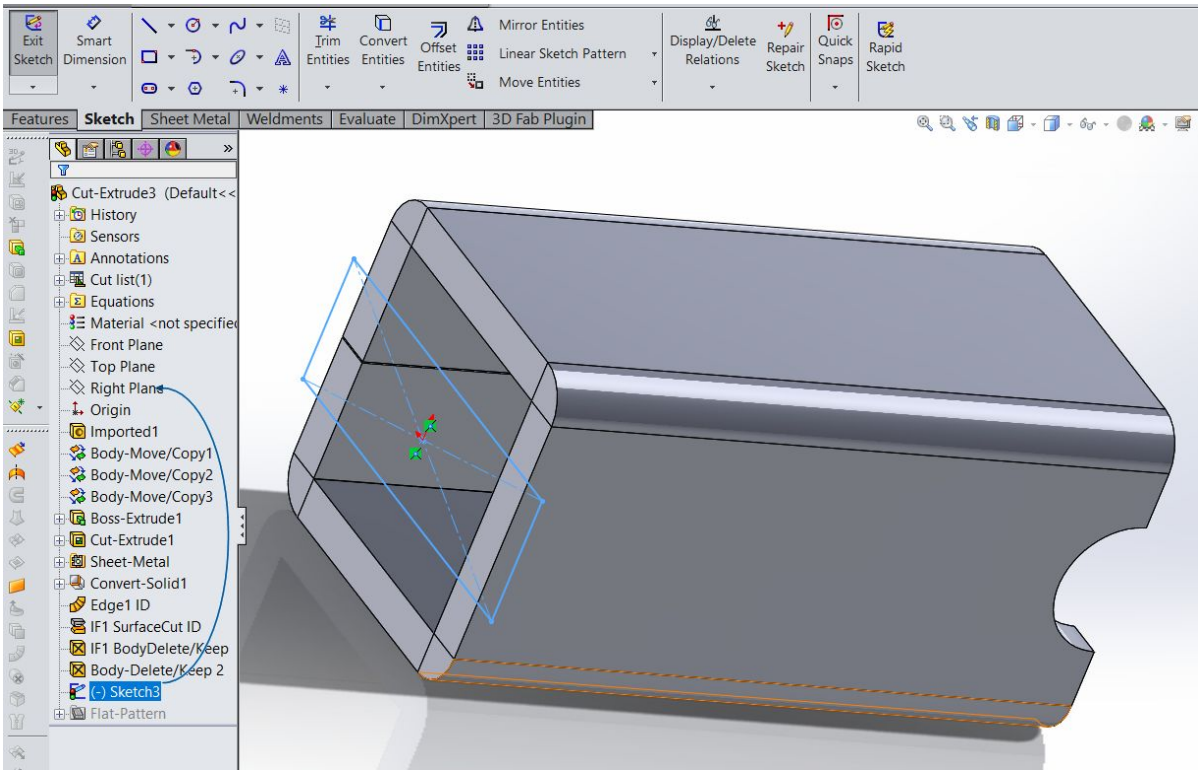

Only the rectangle extrusion will cause the conversion process error, because it is at the base end of the tube - the starting end of the sketch that the structural member is made from. If just the circle extrusion is suppressed, the same error will occur in the plugin. The cut-extrusion at the base of the tube, the rectangle, must be suppressed to properly process the plugin. Doing so results in the compensated assembly below that is missed the original rectangular feature we want.



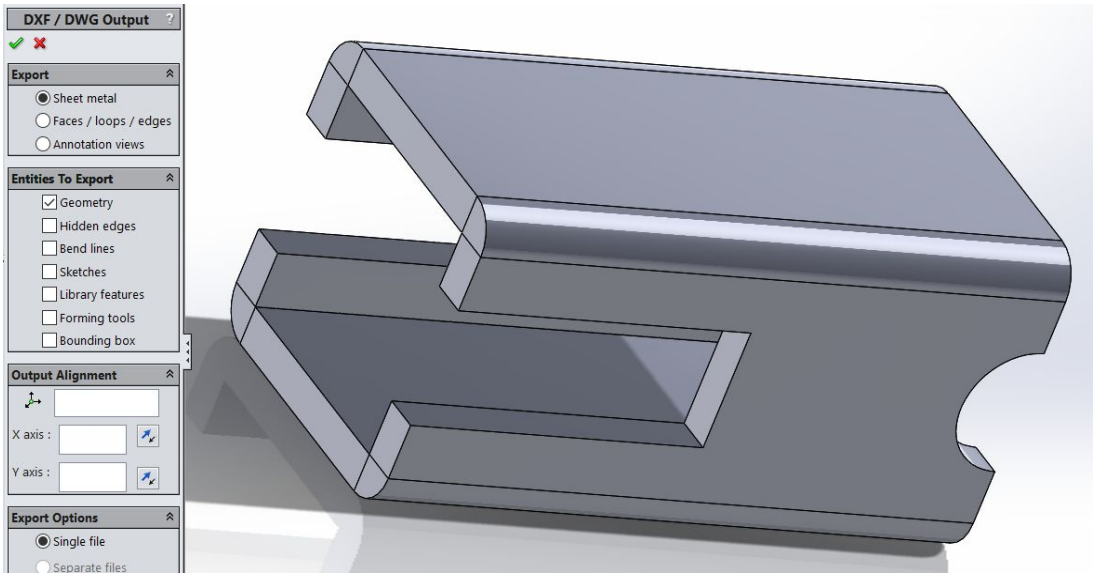
The trick is to manually apply the erroneous feature post-processing. First you have to open the part from the compensated assembly. Then, from the original pre-processing file, enter the sketch for the erroneous extrusion at the base end of the tube and copy all the entities.




Go back to the post-processing file that is missing the erroneous feature and paste the copied sketch onto the matching face, making sure to correct orientation and any other details not pasted.



Apply the cut-extrusion to the pasted sketch in the post-processed file to bring in the missing feature. Once the post-processed part file matches the pre-processed file, you must manually save it as a DXF to import into FabCreator. Select the “Sheet metal” Export option to get a flat unwrap of the tube.



# 4. Notes and Best Practices on the Plugin

- Larger models tend to slow the plugin down, but if the tubes are relatively simple geometry (not too many cuts), the plugin usually completes. Note that it can take minutes, so be patient.

- Internal features seem to really slow the plugin down and often cause it to crash. If you have a tube with 100 holes in it, it might be pretty challenging.

- Internal features tend to behave better when you select “inner edge” or “outer edge” rather than “both edges”.

- Adding tabs (for slots) to the end geometry in the weldment file currently crashes the plugin. However, using the Unfold feature, you can add tabs to the compensated sheet metal flat pattern. Draw on a plane that intersects the axis of the receiving tube, extrude cut the slot profile. Because the part is already a sheet metal feature, it will add the compensation to the bend around the cut profile. The DXF will have to be manually re-saved afterwards.

- Currently, there is no support for extruded aluminum square tubes with sharp corners. This is because a sharp corner does not have a bend radius and therefore cannot be converted to sheet metal. If the ID and OD of the weldment are radiused, the plugin will process the tube, but will output an erroneous geometry.

- Sometimes if you re-run the plugin after modifying or adding a weldment, the original part will remain in the assembly. Avoid this by deleting the old files in the output location (the same location as the weldment file) before re-running the plugin.

- Plug-in does not seem to support Trim/Extend Weldment Feature. This feature would normally be used to cope a tube of one structure group into another. We are not certain of the functional difference, but a Cut/Extrude up to the coping surface (Up to Face Option) of the intersecting Structure group seems to work. You can also try a backwards approach by running the plug-in on the pre-coped tubes, then doing an extrude-cut on the compensated assembly that is created of the pre-coped tubes.

- Plug-in does not support Hole-Wizard features. All holes will have to be extrude cut into a tube.

- Plug-in can output a square tube file with a rotated slit location, but FabCAM will only assume the bottom of the flat pattern is in the middle of the profile.

- The “Pipe” feature in the options does not seem to work on square tube. It only outputs structural joints. Only round tube is affected. This can be worked around if the intersection hole is modeled in the weldment.

# 5. Solidworks References

Here are two references we used in creating the plugin, and which might help if you ever need to convert a tube to sheet metal or create ruled surfaces without the plugin.

- Using ruled surfaces to create a normal cut path: https://forum.solidworks.com/thread/85466
- Cut a slit and flatten tube like sheet metal: https://www.youtube.com/watch?v=dTR9FBEGzy8

# 6. Non-Solidworks Users

Here are a couple of links that might be helpful for users of non-Solidworks CAD software:

- Inventor: https://forums.autodesk.com/autodesk/attachments/autodesk/78/413984/1/Creating%20Coping%20Patterns%20in%20Inventor.doc
- Solid Edge: https://www.youtube.com/watch?v=2PjtV-nakd0

# 7. Service Through Freshdesk

If a file is giving you trouble, or you need help processing it, you can attach it to a ticket on Freshdesk. Please include the Solidworks file and any output DXF files that were created. You may also submit tickets about plugin bugs or feature requests through Freshdesk.

You can create an account here: https://3dfablight.freshdesk.com