VR3 Engineering
Student 3D Modeling and Document Guidelines
2023

Contents

Preface:

VR3 Engineering provides services mainly in the form of CNC notched/profiled and formed/bent tubing. Each tube is precision notched on both ends and each end is 'clocked' or orientated to each other accurately to match 3D CAD part files supplied to VR3. Holes, slots, and other notched features can be added to a tube during this process. Forming/bending of tubing is done after this notching process and while there are few constraints on notching, the bending process has many more stringent constraints.

Other services available include welding, laser cut sheet metal, sheet metal bending, and general machining. These services are provided alongside a tube kit but can be much more costly in comparison to notching. The notching process is streamlined, efficient, and accurate providing the best value to the customer.

Support is also available in many areas such as help with modeling your tube frame, designing for the manufacturing process/constraints, document preparation, and so much more. Please reach out if there are any questions or confusion and VR3 will help you along the way.

Request for Quote:

In order to provide an accurate and timely quote, we require a specific set of documents/information about your project or design. If you do not currently have a 3D model, we are able to provide technical support.

1. **Send to VR3:**
   - □ Entire 3D CAD Model (weldment / tube frame).
   - □ PDF file of Weldment Balloon Drawing.
   - □ Excel Bill of Materials (BOM) with all required tube information.
2. **VR3 reviews model and provides a quote**:
   - □ Trims, bent/formed tubes, documents, and all other features will be reviewed to ensure they are ready for the manufacturing process.
3. **Once the model is approved**:
   - □ Send individual part files (numbered per balloon and Excel BOM).
   - □ Send shipping and payment information (a form will be provided to fill out). We <u>cannot</u> ship the tube kit until we have this information. For welded frames, we will not begin manufacturing without it.

The required files are described in detail in the following sections; examples of each are available on our website.

Required Documents for Quote:

1. **3D CAD Solid Body Model of the Tube Structure with Properly Defined Geometry**
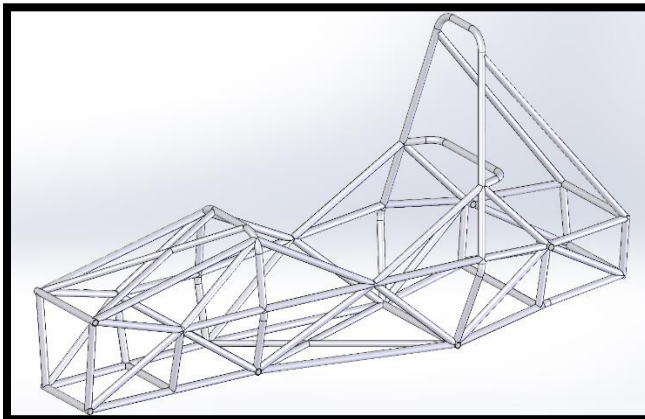
- This document is reviewed to verify: that all trims are done correctly such that there are no interferences or gaps between tubes ensuring proper fit up during weld assembly, bend radii and bending constraints are adhered to, and any other details or features of consequence.

- On it's own, this document conveys little information and is mainly used for discussion. Bundled with a drawing and a BOM, all of the details and features can be fully defined and will allow for an accurate quote and more in-depth discussion if needed.

2. PDF Drawing with Balloons and Cutlist/Material List
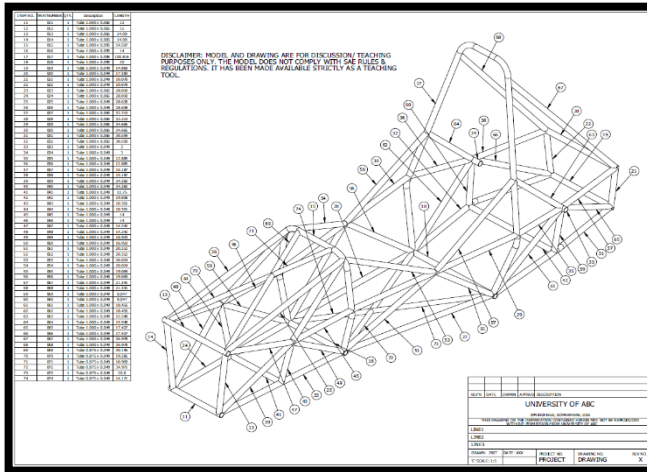
- This drawing shows an isometric view of the frame with balloons that clearly indicate each part relating to the item or part numbers on the cutlist/BOM.

- Create this drawing from the 3D model and before sending, save as a PDF file. A PDF file can not change while be sent or being opened on another computer. It 'freezes' the drawing in time and can be related to the date it was sent easily.

- Each part should have a unique part/item number. Be careful about left and right parts, they are rarely identical in end trims and tool path.

3. Excel Bill of Materials (BOM)

- This document should match the cutlist on the drawing. You can use the drawing to create this file and that will be explained later.

- The following information needs to be supplied for each part: quantity, outside dimension, tube type (rd/sq), wall thickness, approx. length, and any features (bends, holes, etc).

- You can fill out a sample version of this file that is provided on our website.

Preparing Solidworks for Modeling:

Solidworks is a very useful tool for designing tube kits and weldments. Spending time up front to optimally setup the software for weldments will save time downstream and result in a quality and easy to use/update model.

Weldment Setup: Properties File and Weldment Profiles:

The Weldment Property file is a text file which determines what options will be available in the dropdown menu for a cut list item. This information is used when creating automatic cut lists and allows information about tubes to be applied on a drawing BOM very easily when configured properly.

1. Find your weldment properties file > Tools\System Options\File Locations\Show Folders for: Weldment Property File (This will show the folder in which your weldmentproperties.txt file should be located. Typically, will be in C:\SolidWorks Data\lang\English)
2. If there is no folder shown, then you will need to add a location and create your own weldmentproperties.txt file
3. Go to this location and open the text file. It should have the same content and format as the side bar to the right.

> **weldmentproperties.txt**
> DESCRIPTION
> PARTNUMBER
> REVISION
> MATERIAL
> WEIGHT
> LENGTH
> COST
> SOURCE
> SW-Part Number

Weldment Profiles are sketches of a 2D cross-section of a tube. The weldment Profile is extruded along a sketch line in your model to create a 'structural member' which is a tube member in a weldment. A structural member has many tools and advantages when it comes to trimming and having automatically generated information to be used on the drawing later. A Weldment profile must be saved for each size of tubing to be used.

1. Create a new part in Solidworks.
2. Create a 2D sketch on the front plane.
3. Create the outer diameter (OD) of the tube. Use the 'Offset' command on the OD to offset it inward by the wall thickness to create the inside diameter (ID) of the tube.
4. Without leaving the sketch, File\Save as
5. File name: 1.000 x 0.035 (tube size)
6. Save as a Lib Feat Part (*.sldlfp)
7. Description: Tube 1.000 x 0.035 (tube size)
8. Repeat these steps for all tube sizes required to complete the weldment profile library.

This PC
Windows (C:)
data (\\VR3-Ser

1.000 x 0.028.sldlfp
1.000 x 0.035.sldlfp
1.000 x 0.049.sldlfp
1.000 x 0.058.sldlfp

File name: 1.000 x 0.035

Save as type: Lib Feat Part (*.sldlfp)

Description: Tube 1.000 x 0.035

We have a large selection of Chromoly 4130N in stock. See our website or the Appendix for a chart of sizes available. We recommend using round tubing because it is more readily available and significantly cheaper than square/rectangular tubing. However, sometimes square tubing is required. To ensure the square tubing is modeled properly and accurately, the inside and outside corners of the square tube should have radii on them. The outside radius can be found in a chart on the VR3 website and the inside radius is the outside radius minus the wall thickness such that the radii are concentric. See the below image for some example square tube profiles.

Square Tube Weldment Profiles

* corner radii must be 'concentric'
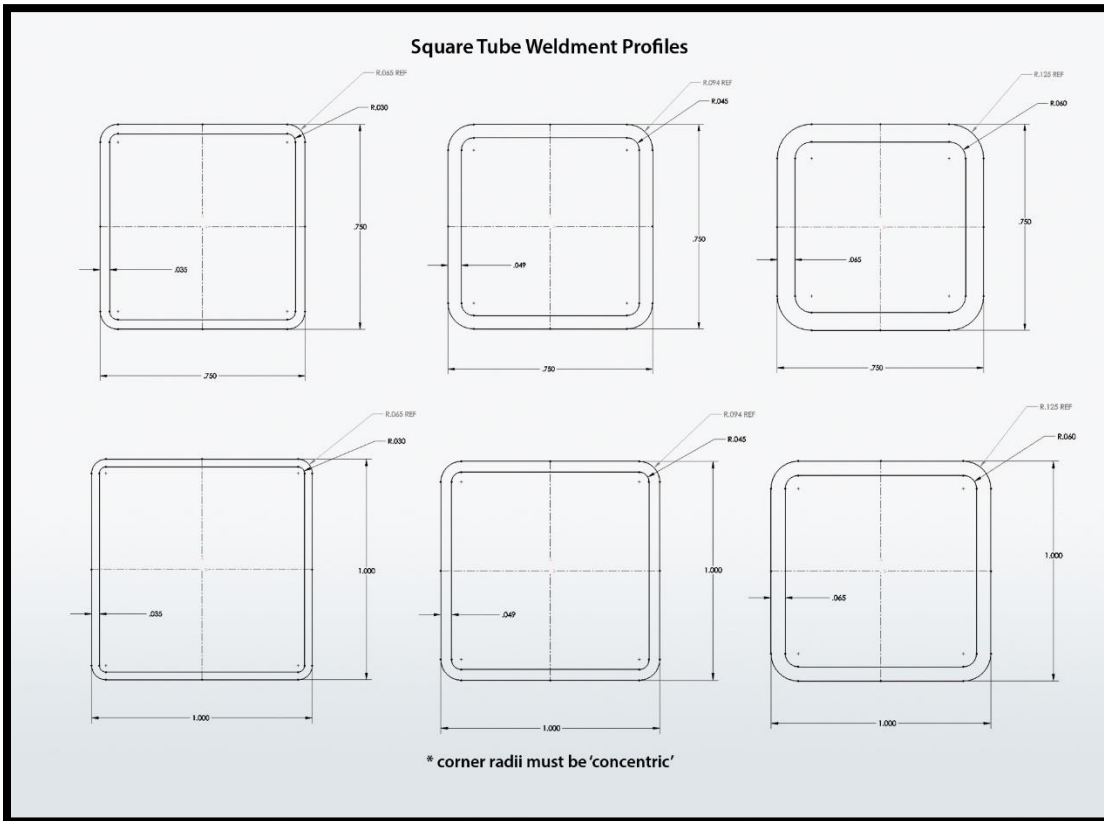
# Weldment Modeling: Step-By-Step Instructions:

Weldment Modeling Procedure:

This guide is to assist designers in preparing information suitable for the tube profiling process. These methods and suggestions are specifically for Solidworks but should be applicable to other 3D CAD Software. Tubing structures are best suited to be built as a 'weldment' part. Some advantages are:

- One file contains the complete parametric information and therefore can be sent without related attachment files or libraries.
- Trim and extend features are quick and easy within the weldment environment.
- Profile library of tube sizes is convenient for creating automatic BOM information.
- One file to update with any changes to tube sizes or geometry.

This model is reviewed to ensure all components are ready for and are applicable to the manufacturing process. Trims, bend features, basic tube geometry, and many more are carefully checked once it is sent. However, the supporting documents of a PDF drawing and an Excel BOM are needed to efficiently and effectively review the package as a whole and not miss the small details/features. There is a sample model, PDF drawing, and Excel BOM for your use and reference located on our website. You can follow along the design tree of the model. This will match with the following information below.

Before starting modeling make sure you have your weldment profile library updated and your weldment properties file in the correct location. See Weldment Setup: Properties File and Weldment Profiles on page 5.
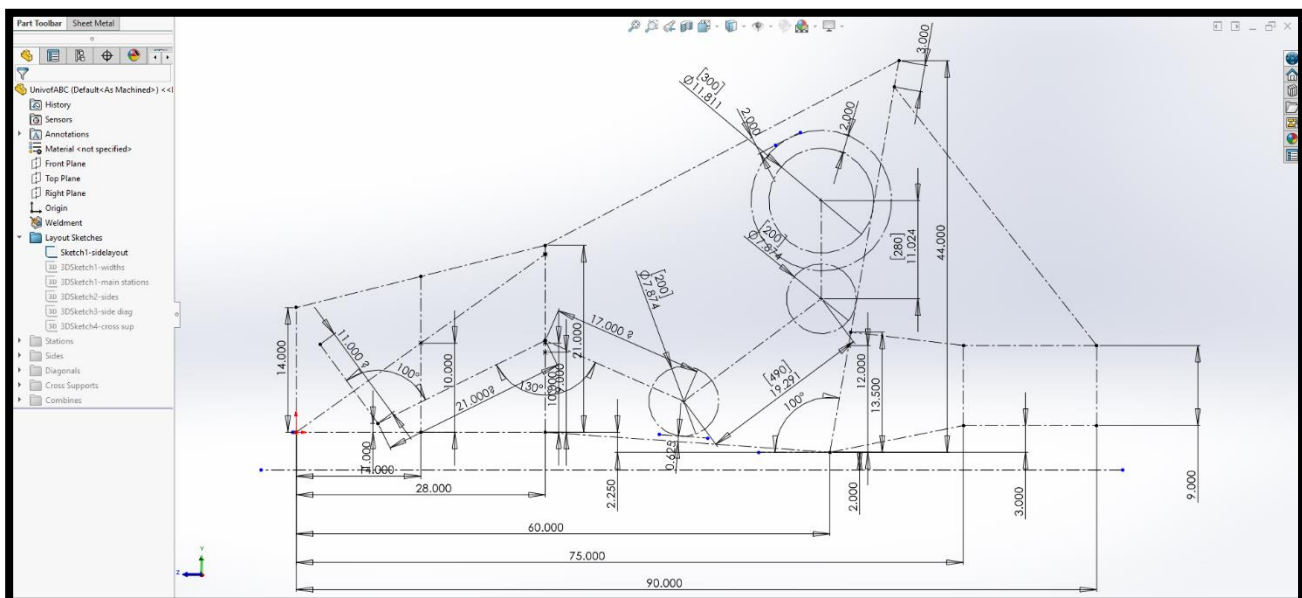
Profile sketch on the initial reference plane of the structure.

1. Each structure starts by establishing an initial reference plane. This plane is usually through the centerline of the structure. This establishes the profile of the structure. This is typically a 2D sketch. Avoid using 3D sketches initially. Use construction lines for these 'design' or 'reference' sketches.

   NOTE: Ensure all sketches are fully defined using dimensions and constraints. This is conveyed by the colour of the lines/points in the sketch (blue is underdefined, black is fully defined). Beside the sketch name in the design tree, a '(-)' will show up on sketches not fully defined. Do not use the 'Fix' constraint to fix or to define underdefined features.

3D sketch widening the frame from the initial 2D sketch.

2. From the 2D reference sketch, create a 3D sketch to widen the 2D profile of the frame and bring it into 3D space. Tubes centerlines that run from left to right/perpendicular to the first sketch, any important cross lines like the top centerline of a roll hoop, or useful intersection points can be included here.
3. Define formed/bent tubes in unique sketches/planes. 2D sketches are easier to work with for bends/arc lines. More detailed info about proper bent tube modeling will be provided in the Bending Guidelines section.
4. Continue to build the tube frame in the solid model generally in the same order as one would build the structure on the shop floor with additional 3D sketches. Primary members, then secondary members, then tertiary/support members.

Additional tube centerlines are added to grow the frame.

5. Once all layout sketches are complete using construction lines, add 3D sketches to define the paths for Structural Members using solid lines. This is should be done in groups (i.e. main members, bracing, etc.) to ensure clarity.
6. Add tubes as 'Structural Members' with the correct OD and wall thickness. Do not 'merge arc segment bodies' when creating bent tubes. Use the combine tool as the last step for bent tube segments to create one continuous bent member. Trimming features are generally more reliable when the bodies are not combined. Use a unique structural member for each tube. This allows for ease of changes to tube sizes or trims and is less confusing.

7. Use 'Trim/Extend' to trim each tube **as it is added** to the structure. This mimics the process of building a tube structure on the shop floor and ensures no trims are missed. This is also less confusing than determining the trim sequence at the end once all the tubes are added. See the design tree sequence in the next image for an example. In depth trim guidance and issues to watch out for are provided in the Tube Trimming section.
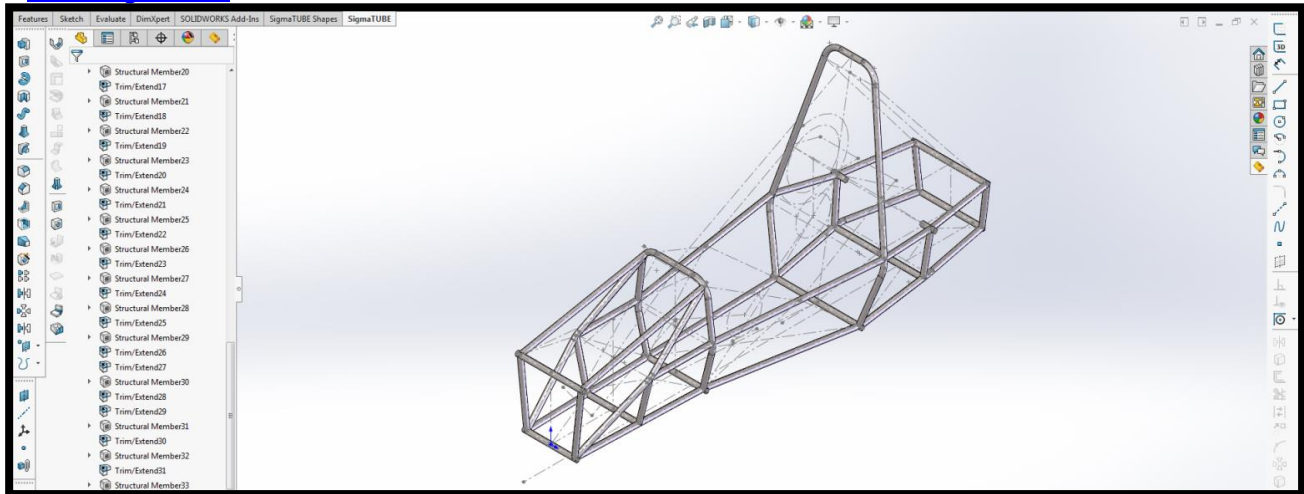
Structural Members are added and trimmed in the order that the structure is to be built (notice the sequence in the design tree on the left).
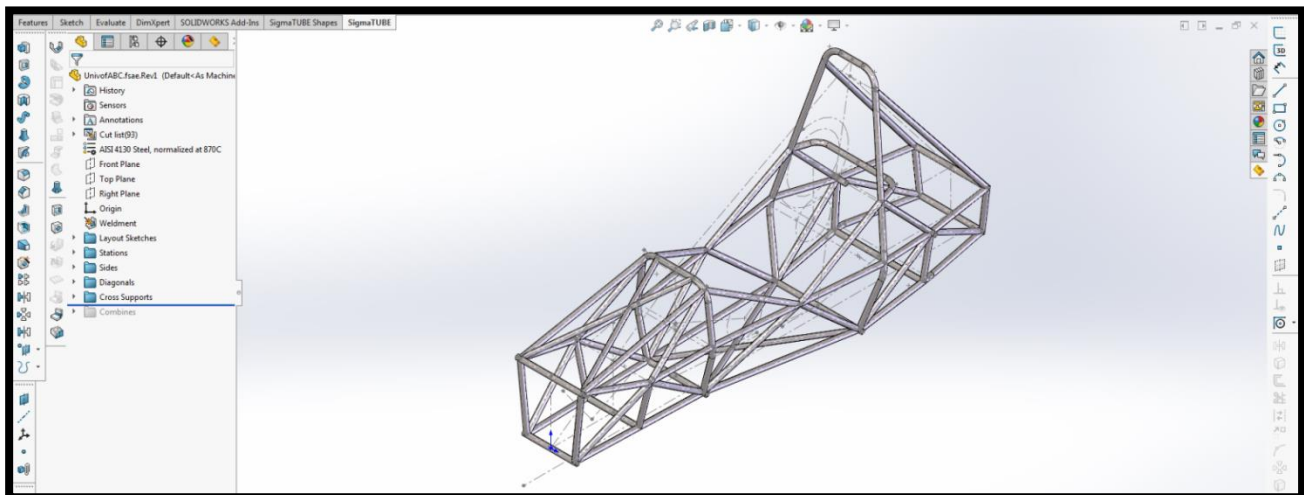

Complete chassis with main, formed, and supporting tubes as structural members.

8. Use folders to organize the design tree. This allows for features to be found much more easily and keeps everything clean and organized.

9. Avoid mirroring tubes. Mirrored tubes generally have unique profile paths and require separate profile programs to produce. Trim features are more robust with non-mirrored tubes and changes are easier to apply downstream.

10. Once all the tubes in the solid model are added and trimmed, combine the bent tube segments as required to produce single tube components.

    NOTE: Our CNC profiling process cuts tubes 'perpendicular to the surface'. Excess 'feathered edges' are cut away resulting in the part looking slightly different after profiling vs in the model. This perpendicular cut to the surface is ideal for welding and results in a complete tube to tube contact around the perimeter.
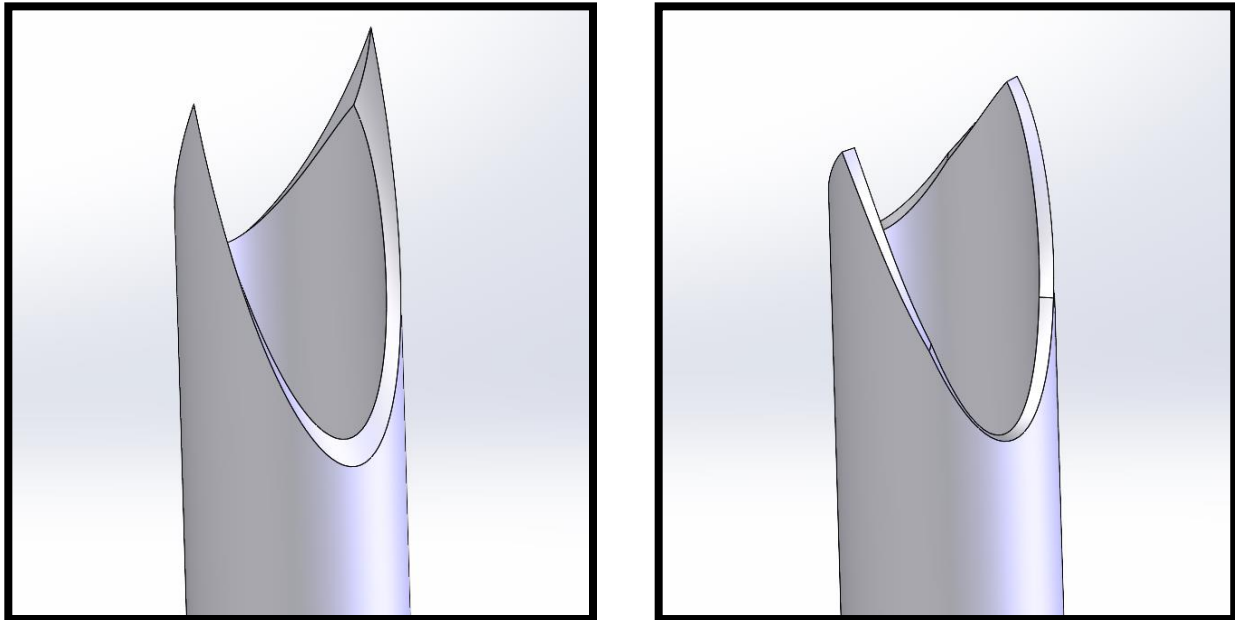
The as modeled profile created by trimming in SOLIDWORKS (left) and what the cut part will look like after cutting perpendicular to the surface (right).

Cut list Setup:

When you have completed your model, you will have all of your bodies in your Cut list() folder at the top of your design tree. They will most likely be put into Cut List Item folders. The body of each tube is found within each folder named after their structural member or the last feature which affected them. Notice that the number of bodies in your model will be shown in brackets after the 'Cut list' title.

1. Right click on the Cut list(xx) folder. 'Create Cut Lists Automatically' will likely be checked. Toggle this off by selecting it in the menu to ensure the list does not change unnecessarily. If this has been on until this point, all tube bodies will be within Cut List Item folders. If they are not, simply turn on 'Create Cut Lists Automatically' and this will place them in folders. Turn this back off right after.
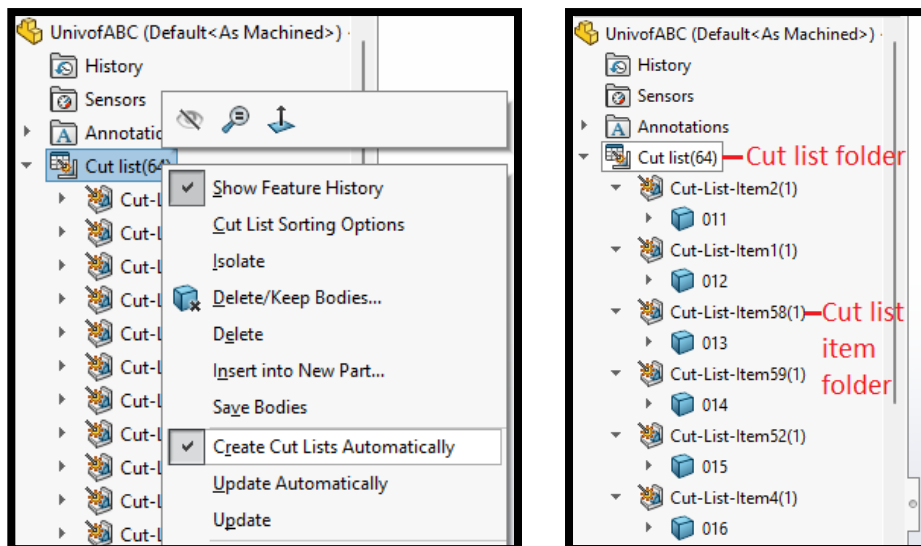
Cut List 'Create Cut Lists Automatically' option and proper numbering of parts.

2. In two places you should assign a part number. Give each Cut List Item and the body within the Cut List Item folder a part number (001, 002, 003, etc.). The part number you assign to the Cut List Item is what will appear on the drawing cut list later. The part number for the body itself will help speed up saving of individual parts at the end of the process and reduce human error.

3. Right click on a Cut List Item folder and go to properties. This will show all of the properties of the tube(s) within the folder. Things like description (tube size) and length will automatically be generated from the structural members and the weldment library that was set up previously. Under the Property Name column on the last entry where <Type a New Property> is, click on that cell and then click drop down menu and select PARTNUMBER. This comes from the Weldment Properties File set up earlier. Type a part number under 'the Value / Text Expression' column. You must click ok in the bottom left to enter these values. You cannot click on the next Cut List Item as this won't save the previous values you entered.
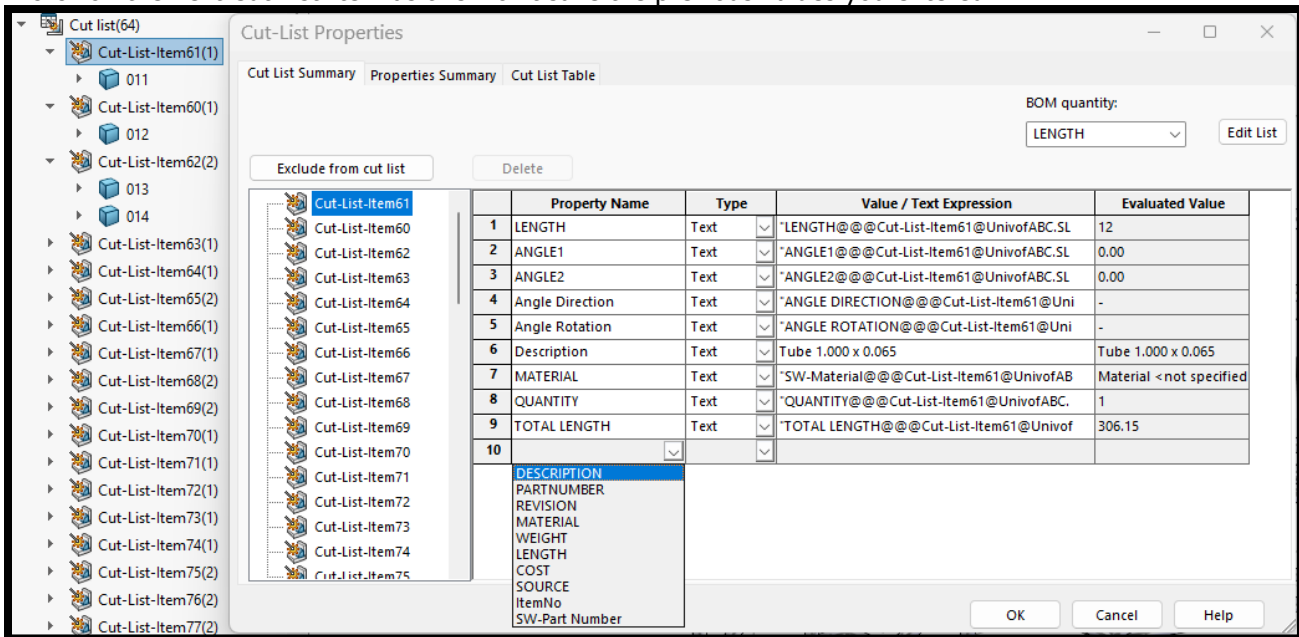

Cut List Item properties window

4. Sometimes there will be two or more parts in a single Cut List Item folder. Solidworks believes these parts are identical and can have the same part number with a quantity of more than 1. However, Solidworks consistently gets this wrong with left and right parts. If this happens, simply delete the Cut List Item folder containing the parts. This will place the parts in the overall Cut List folder at the end. Next, right click on each part individually and manually add them to a Cut List Item folder by selecting 'Create Cut List Item'. If you don't see this option that means 'Create Cut Lists Automatically' is still on. While 'Create Cut Lists Automatically' is off, any new parts added will also need to be manually added to a Cut List Item folder.

Drawing Setup and Procedure:

1. Create a Drawing file from your weldment. File>Make Drawing from Part. This drawing should include a sheet with an Isometric ballooned view as well as a second sheet containing three main views (side, front, and top) as well as any relevant section views and detail views. Add overall length, width, height dimensions, as well as any pertinent design dimensions. These dimensions are the only way to verify that your model is designed correctly and will help you during weld assembly.
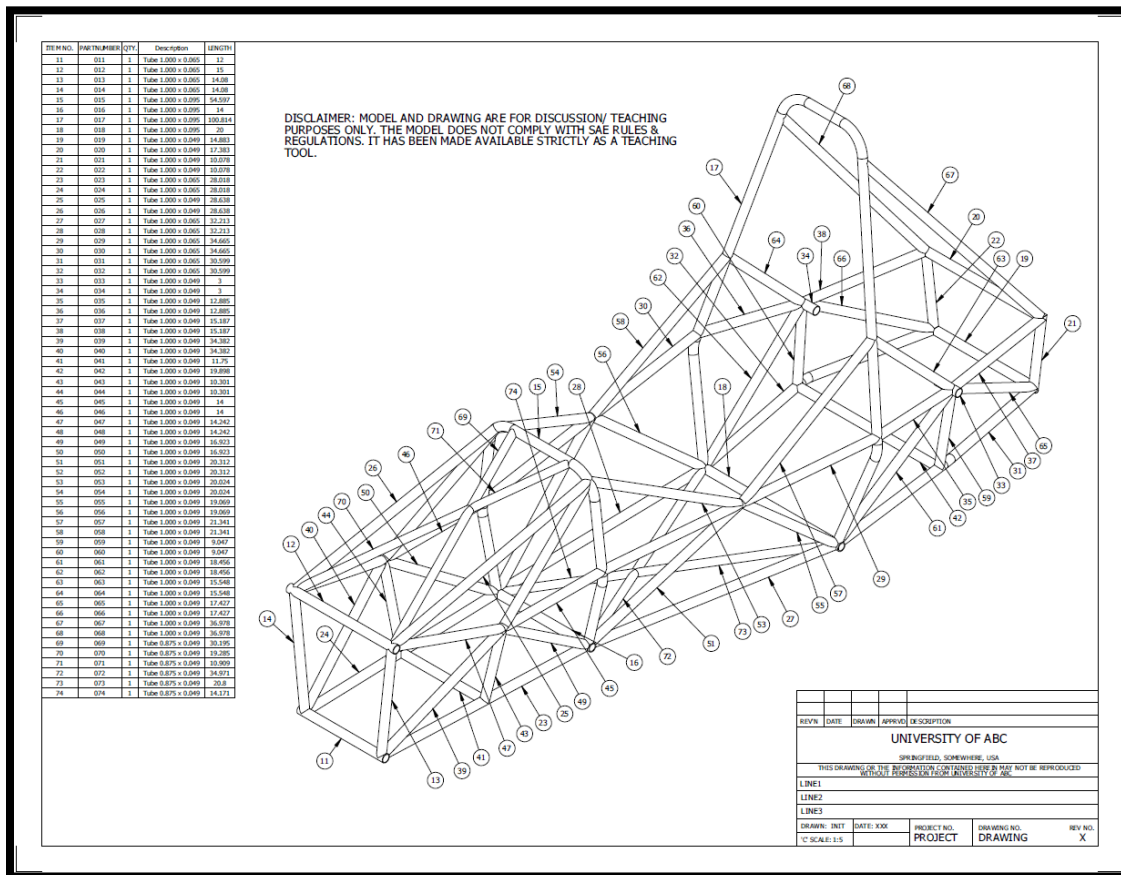
Example of the first sheet of the drawing.

2. A screenshot of sheet one of a sample drawing is shown above and this drawing in PDF format is available on our website which will also include the second sheet. The cut list is shown on the left and the information on it stems from the Cut List Item folder's properties. Each tube has a balloon pointing to it calling out the part number. Item numbers can be used but part numbers are preferable. The arrows for the balloons should point to the middle of the tube away from clusters so that it cannot be confused for another tube. Scale and proportion your drawing well. Do not make the isometric view very small when you have space to make it bigger. Drawings are meant to be printed out and read/marked up, not zoomed in on a computer screen. Try your best to mimic our sample.

3. Review and check the design:
   - Confirm the design dimensions with your drawing
   - Confirm the tube sizes
   - Confirm all tubes are shown

4. NOTE: The length on the drawing and within the cut list item properties is the overall length of the tube from the furthest points on either end after trimming. As stated just before this section, we cut the tubes 'perpendicular to the surface' and so the length on the label of the tube we provide you will differ slightly from this.

Excel Bill of Materials (BOM):

A sample Excel Bill of Materials is available on our website. You can fill this out directly and the information can be extracted from the drawing you have created. The weldment cut list on the drawing can be saved out as an Excel file. Right click anywhere on the table and click 'Save As'. Chose an Excel file format for the file and save it. This is the cut list information but not organized in an efficient way. You can rearrange the data to fill out the sample Excel BOM available on our website. Note any features (holes, cutouts, slots, bends, etc.) or out of the ordinary information here.

Bending Guidelines:

VR3 tube kits can include formed/bent round tube components. The value and convenience of our combined profiled and formed tube kits is exceptional. However, there are restrictions and limitations. First the tube is profiled as a straight tube, then it is bent with the correct orientation to end trims and with bend allowances. The critical step is setting up the procedure. There are many factors and variables in the bending process that require concise procedures and a knowledgeable operator. However, once it is set up, consistency and repeatability are very good. We can produce formed tubes with multiple bends in different bend planes subject to restrictions. However, simpler is always better.

For an example, a tube with 2 bends can be defined with only 4 points. The start and end points, and the intersection between each straight section. It's that simple. Connecting these points with lines, each bend plane can be defined using the line before and after the bend intersection. Arc lines that are defined in each bend plane can then define the radius of the bend. This modeling progression is show below.
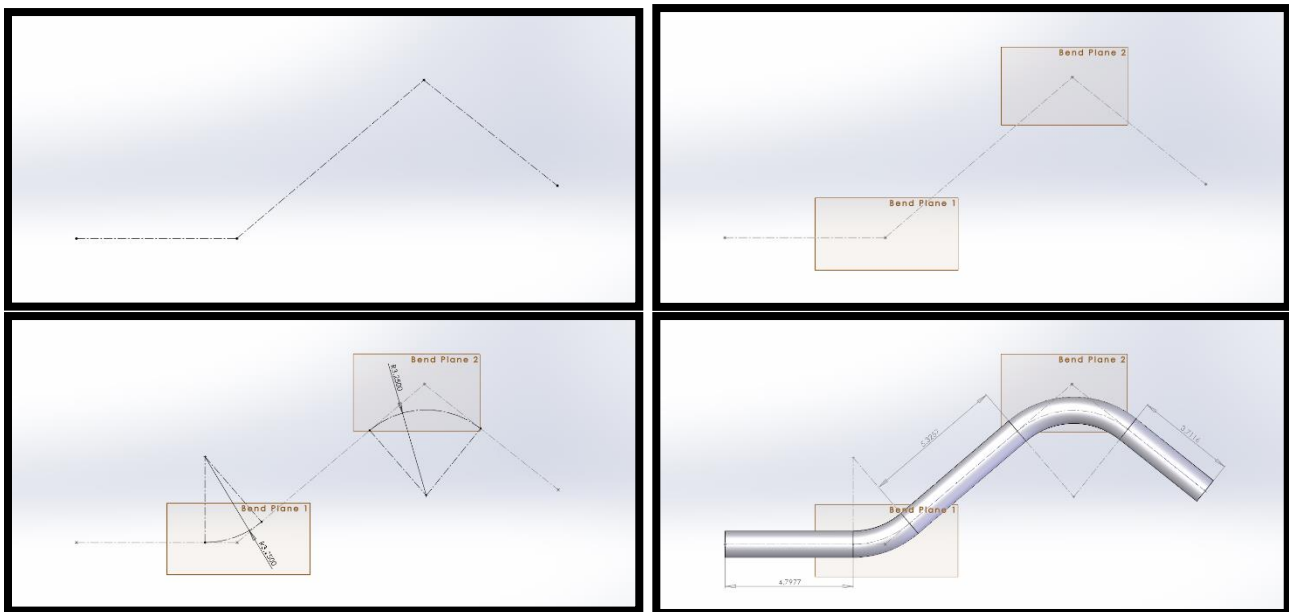
The progression of modeling a formed/bent tube.

Bending Constraints and Modeling Considerations:

- The arc of a bend radius MUST be tangent to the straight line before AND after the bend.
- The above modeling procedure applies to both 2D (flat) and 3D bent tubes (out of plane bending).
- We cannot bend over 180° in a single bend. If the sum of bends in a single direction adds to above 180°, or if the tube bends in a different plane after 180°, usually there are issues geometrically with hitting the machine during bending. Reach out to VR3 to discuss.
- All bending dimensions refer to the tube centerline. The radius of the bend die corresponds to the centerline radius ('clr') of the bend. Spec all bends by centerline radius in your design.
- Bend radii must match the tooling/bend dies that VR3 has available. The tighter the radius of a bend, the thicker the wall thickness of the tube must be in order to withstand the stresses of the bending process without failing/deforming along the surface of the bend. Flattening and wrinkling are the most common issues.
- A bend die/radii chart can be found here in the appendix or on our website.
- There needs to be a minimum straight length before and after a bend for the bender to hold onto the tube. If an end leg is too short, VR3 can 'scribe' the desired end profile on the surface of the tube with extensions provided beyond this. The tube would then need to be trimmed at assembly by the student

team following the scribe lines as a guide. Below is a sample image of a scribed tube.

- The distance between subsequent bends must be a minimum length to bend. There is no work around for this. If both bends are in the same plane, we can get away with a smaller straight length than if the bends are in different planes. See the Bend Die Chart in the Appendix for values.
- Our process does not allow for trims, features, or scribes that are within the bend feature of a tube. In the case of a trim within the bend, VR3 will provide a bent tube with extra end length after the bend. The trim within the bend will have to be trimmed to suit at weld assembly.
- Adjusting tube end lengths to adhere to our process can be done by adjusting the first 4 points. The tube should update and change accordingly if done right. Radii can be updated easily too.

Tube with a short leg and a 'Scribe' line.

Tube with 'Flattening' due to a tight radius and thin wall.

Next Steps for the Quote Process:

1. VR3 needs 3 files to review and produce a quote.
   Checklist:
   - A 3D model (.sldprt file or other applicable file if not using Solidworks)
   - A PDF isometric ballooned drawing
   - An Excel bill of materials (BOM)
2. Samples of all of the above can be found on VR3's website. Send these 3 files and VR3 will review. The model will be reviewed for trim issues, bending constraints, tube sizes, and another other special features. A quote will be generated using the Excel BOM. All three files should match up with each other (i.e., the Excel BOM and the cut list on the drawing) and be the most recent revision or update.
3. When the quote is sent, any comments or concerns about the model or documents will be brought up. Communication back and forth will start where you clean up issues and adjust as per our manufacturing constraints, send in updated files, and VR3 reviews again.
4. Once this back and forth is complete and no issues are left, produce a part file for each tube in the weldment while maintaining the parametric link to the solid model weldment. See the Individual Part File Creation section for more information.
5. Visually inspect each tube when saving. Check the end trims. Save this file in its native (.sldprt) format. Save the part as its 3-digit part number. If you are using a CAD program other than Solidworks, please see the Using Other CAD Software section.
6. Ensure bent tubes are a single body and conform to all of the above bending guidelines to be compatible with our bending process and tooling.
7. From these individual part files VR3 extracts the profile data to generate the CNC tool path for the tube profiler. We will profile, bend, label, package, and then ship the set of tubes ready for weld assembly.
8. If your team is interested in having VR3 weld the assembly, please let us know ahead of time. A different quote must be generated including this cost and the model might be subject to different requirements. Additionally, weld scheduling/timeline is very dependent on work load at the time of the order.

Tube Trimming Guidelines:

General Trimming Guidelines:

Whenever one tube intersects or interferes with another, one of the two tubes should be trimmed to the other. Trimming is done in a layering fashion. Generally, there is one main member or intersection that all other tubes trim onto. A tube running through, the bend of a bent tube, a mitre intersection between two tubes, etc. Then, the primary members trim on top of this. Usually, the largest outer diameter tubes are these members. Next, secondary tubes trim onto the primary members and the main member or intersection. Then, tertiary members trim on the secondary, primary, and the main member or intersection and so on and so on. Two tubes should not trim to each other, only one overtop of the other.

For some visual help with this, you can see our website for a sample FSAE chassis. Take a look at the trims in the design tree and inspect each tube cluster in the model. Visually inspect all of your tube clusters by selecting each tube (highlighting it in blue) and then inspect the end trim lines shown.

Sometimes when it comes to trims, Solidworks just doesn't want to do what you want it to for seemingly no reason. In these cases, play around with the trim options and see if you can get anything to work or get creative with solutions outside of the trim command. Worst case scenario, highlight/explain the issue when sending files to VR3 and we can often manage it on our end.

By default, turn both 'Allow Extension' off in the trim command settings and select 'Bodies' as the trimming boundary. 95% of the time these settings will be what you need. This is shown in the image below. Other trim issues where these settings need to be changed will be explained in detail below. If you are having particular trouble with a trim not being created (sometimes this is just Solidworks' fault) you can try turning on 'Allow Extension' or you can use 'Face/Plane' for the trimming boundary to try and get it to work. Sometimes this fixes the issue, but other times it can introduce another issue. Solidworks can be finicky and workarounds are sometimes needed.
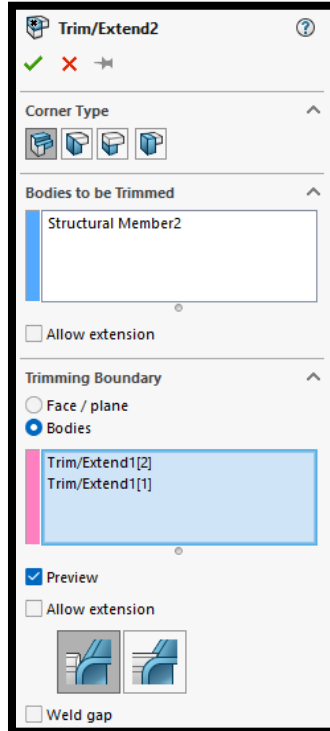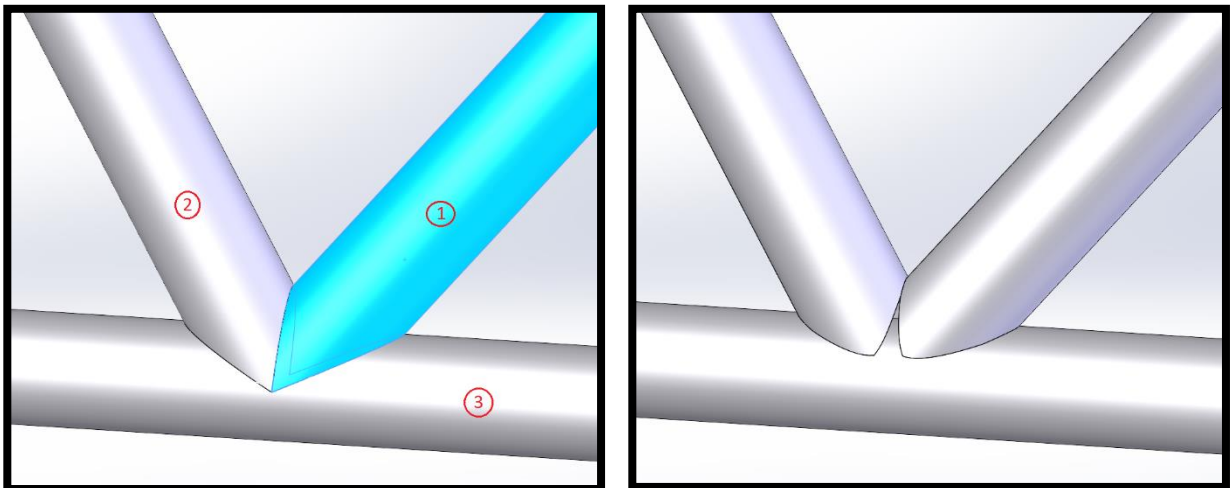
Hollow Nodes:

Hollow Nodes occur when two tubes trim to each other. One tube should trim overtop of the other, not each tube to each tube. If two tubes are trimmed to each other it looks proper in the model at first glance. However, as shown before about the CNC process cutting 'perpendicular to the surface', in reality these two tubes will have a gap between them. This makes fit up at weld assembly more difficult, welding much more difficult, and the node structurally weak (edge weld versus a stronger groove weld). To avoid this, trim tubes in a layering fashion. A way to check for Hollow Nodes is to select both tubes in question and right click to select 'Isolate'. This shows only these two tubes and you can look more closely at the end trims between them. Below are some images to show how to avoid Hollow Nodes.
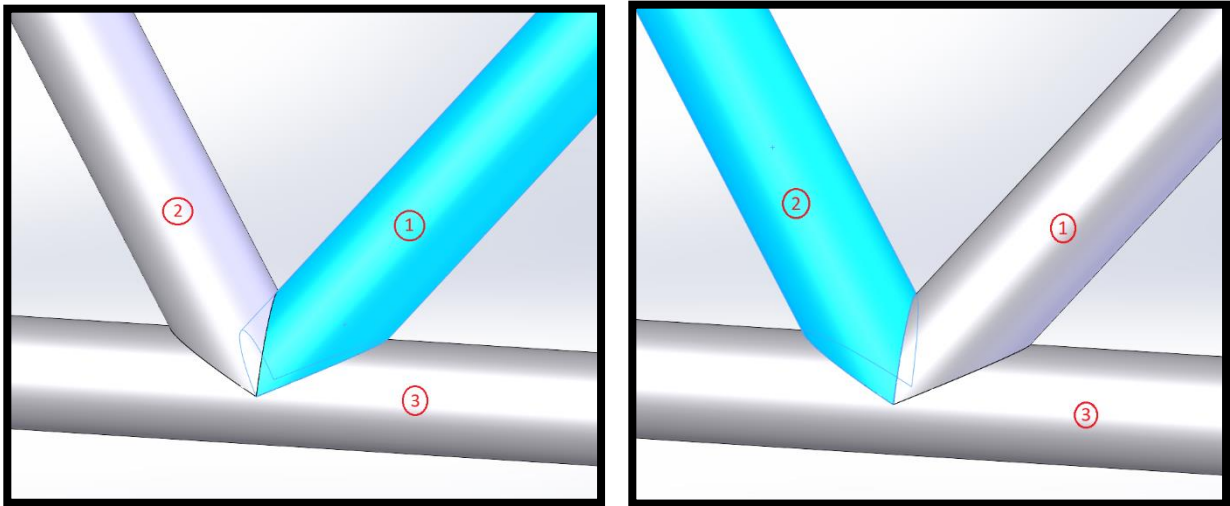
Example 1: The left image shows Tube 1 trimming onto Tube 3 only. The right image shows Tube 2 trimming onto both Tubes 1 and 3. This is the correct trim sequence to avoid a hollow node.


Example 2: The left image shows Tube 1 now trimming onto Tube 2 as well as Tube 3. Tube 2 is still trimming onto Tubes 1 and 3, the same as above. This causes a hollow node. The resultant profiled tubes are shown on the right causing a gap.
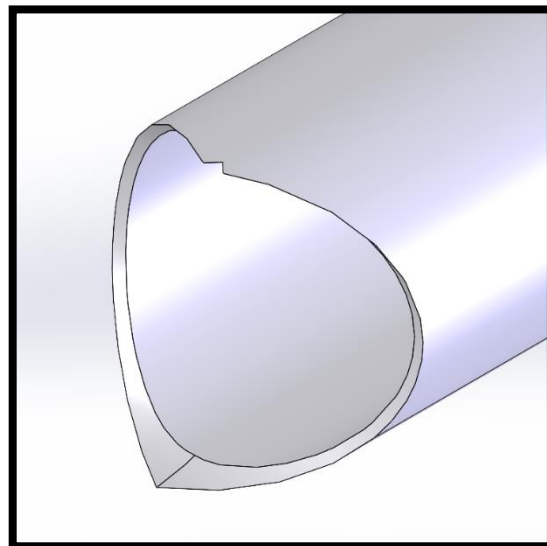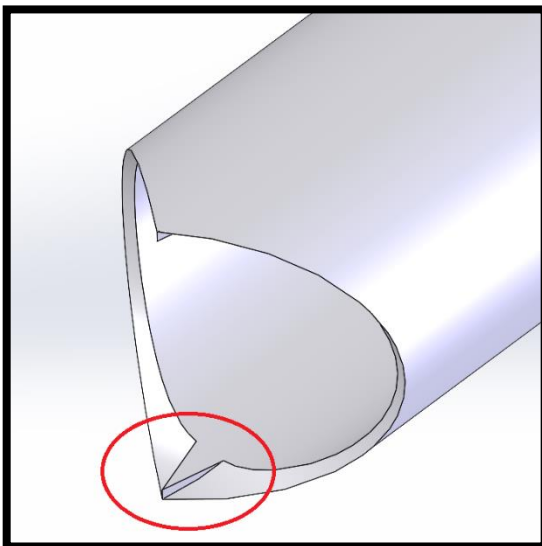
Trim to Mitre/Toothy Trim:

A 'toothy trim' occurs when a tube is trimmed to two other tubes that are mitred together as a mitre joint. Solidworks can incorrectly trim this tube when certain settings are used. For this specific case use 'Face/Plane' as the 'Trimming Boundary' and select each tube that make up the mitre joint. Ensure that all bodies within the trim are 'discarded'. You can do this by clicking on the 'keep' tag to toggle it to 'discarded'. These are the 'toothy' trims that should be removed. Check all tubes that trim onto a mitre by right clicking on the tube and selecting 'Isolate'. This way, you can look at the end trim of one tube at a time and visually inspect it for any toothy trims.
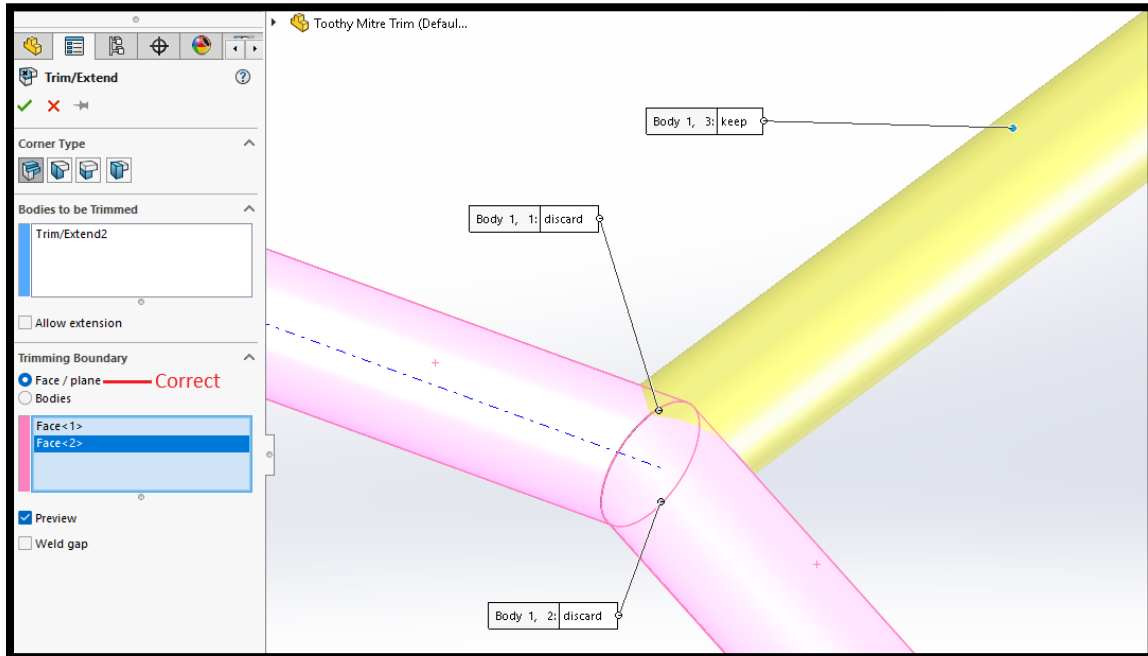
The options and selections to properly trim a tube to a mitre joint.

The left image shows a toothy trim. A 'chunk' of extra material that is as thick as the wall thickness of the tube. The right image shows a properly trim to a mitre. Notice the top 'V' cut out. This generally means the trim to the mitre is good.

Allow Extension:

There are two 'Allow Extension' options in the trim command when selecting 'Bodies' as the trim boundary. The first 'Allow Extension' within the 'Bodies to be Trimmed' section allows the body that is being trimmed to be extended to reach the tubes that are selected to be trimmed to if it is already not doing so. This is very rarely needed so only turn this on if your tube does not fully reach the tubes you wish to trim it to. The second 'Allow Extension' extends the trimming boundary tubes in both directions, and causes tubes to trim to an infinitely long version of the trimming boundary tubes instead of the body that is actually there. This can cause gaps that shouldn't be there and is shown below.
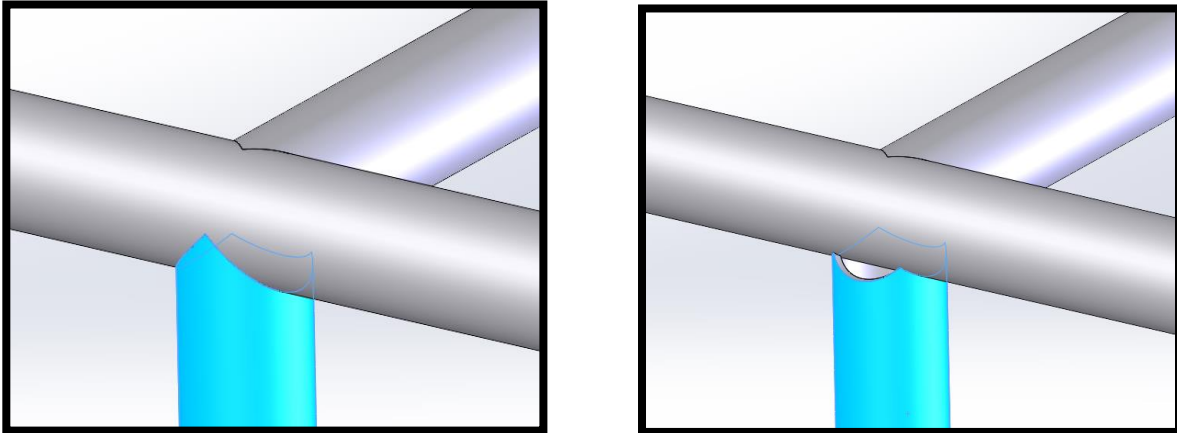
The left image is the correct way to trim the bottom tube to the other two. The right image shows when 'Allow Extension' in the trimming boundary is left on and a gap is created.

Trim Bodies:

A 'Trim body' is when the part of a tube that is trimmed away during the trim command is not actually removed, but is instead made into it's own body. This throws off the total tube count by adding an additional tube to your drawing cut list and BOM that shouldn't be there. The 'Trim Body' is usually hidden within a tube cluster and is difficult to find. It comes from the trim command when a body that should be removed has a 'Keep' tag pointing to it. If you click on the word 'Keep' it will turn to 'Discard' and will remove the body appropriately.

A way of checking for trim bodies is to look at the cutlist you've created on your drawing. If there are any tubes with a length less than or around a single inch, they are probably a trim body. If a tube is this short by design, then you can ignore it. To find the body which then can help find which trim command created it, follow the steps below.

1. Right click any cut list folder and go to properties
2. Click the 'Properties Summary' tab at the top
3. Select the Length Property
4. Scroll through until you find the length in question
5. The name of the cut list item folder is shown in the property name column
6. Open this cut list item folder and right click on the body (blue cube icon) and select 'Isolate'
7. Find the body and select the outer surfaces of the body until a 'Trim/Extend' command shows in the top left of the view window. Click on it. This is the trim command causing the trim body issue.

Square Tube/Bracket Trimming and Guidelines:

Along with round tubing, we have the ability to profile square tubes with our 4-axis setup. Square tubes can be very beneficial as design features for mounting components on the chassis structure. However, we suggest using round tubes wherever possible. Consider the following for square tubes:

- Our preferred sizes are 1.000" square x 0.035", 0.049", or 0.065" wall thickness.
- See the Weldment Setup: Properties File and Weldment Profiles: section of this document for sketch profiles with proper corner radii.
- Square tubing material is significantly more expensive than round tubing material. It is also more expensive to set up and profile square tubes.
- Simplify the trim sequences and resulting trims on square tubes. Avoid 'mitering' square tubes wherever possible.
- Square tubes cannot be formed/bent by VR3.
- Square tubes 'nest' onto a single round tube very well but are difficult to nest and trim into complex clusters with round tubes and multiple sizes. Simpler designs require more effort but are more economical to manufacture and much easier to locate, align and assemble downstream.

Square tubes are often used for suspension brackets on an FSAE chassis. This is a good use of our services and there is very little issue with doing this. However, be careful about the way the cutouts in a square bracket are modeled. Our process cuts 'perpendicular to the surface' and on a flat face of a square tube the cut will be perpendicular to said flat face. The corner radii however, is more like a round tube, cutting from the centerline of the radius outwards.
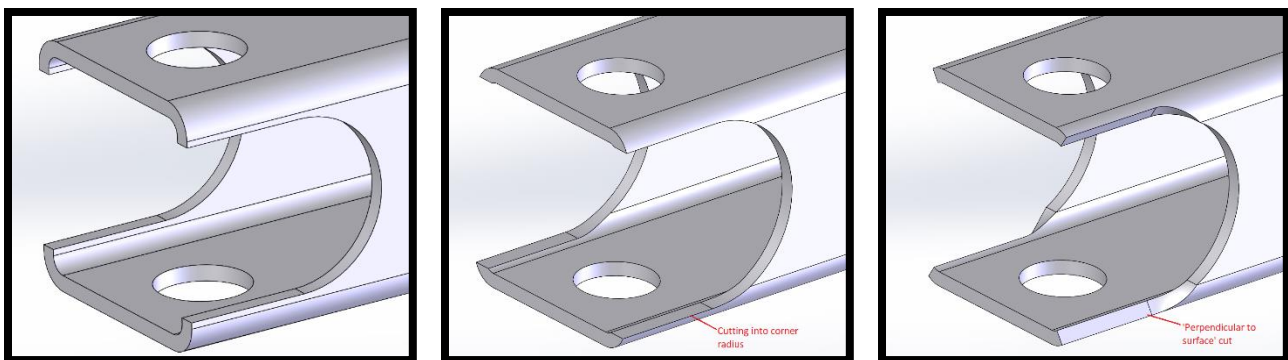
The left image below shows a square tube that will be cut exactly as is shown and is the best way to model such a tube. The middle image shows where the cutout has intruded into the corner radius. The right image shows what this will actually look like after the profiling process. Avoid cutting into the corner radius with cutouts.

Using Other CAD Software - Best Practices:

Our preferred CAD Software is Solidworks as it provides quality tools for weldments and tube structures. However, there are a large number of additional CAD programs that can be used. A neutral file format can be used, which VR3 will import into Solidworks in order to extract a toolpath.
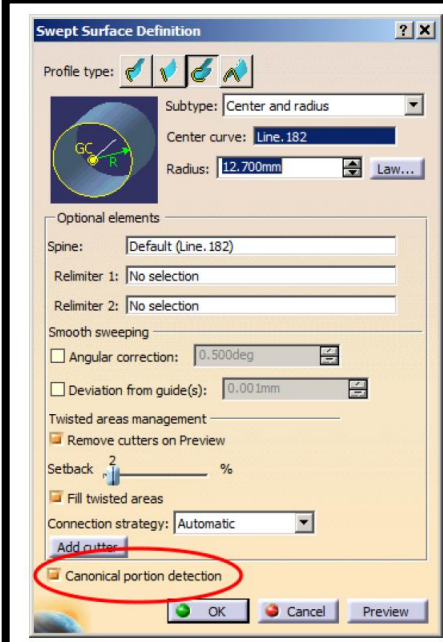
There are some common problems when creating files in other CAD programs and importing them into Solidworks:

- 'Sweeps' are used to extrude profiles along a line to make a tube member. If the profile sketch plane is not exactly perpendicular to the line in which the profile is being swept the resultant tube will be elliptical.
- Lines before and after a bend need to be tangent to the bend radius. See the Bending Guidelines section. This set of bending guidelines applies to any software used.
- Broken surface tubes: caused by a variety of factors, some unknown. Typically caused by incorrect 'sweeping' of the tubes but also caused by saving the file as a 'surface-body'

Sending your files in a neutral file format is best. STEP (.stp), IGES (.igs), or Parasolid (.x_t) are three that are compatible. It is best to check the suitability of the model prior to sending to VR3. Ideally this would be done by accessing Solidworks and opening the files to check that there are no errors when importing. After this, the most important thing is to turn on 'View Temporary Axes'. There should be blue axes along the centerline of each tube. Also, measuring the outer or inner diameter of the tube should give a diameter measurement. If either of these don't work, the tube is not cylindrical and will cause issues with our toolpath generating software.

If you do not have access to Solidworks to double check the above, you can always send VR3 a test version of your model or a test part. We can open it on our end and let you know if there are issues and if it is a common issue we have encountered before, we can help guide you along to modeling it properly.

An issue specific to CATIA:

A specific issue has been realized when exporting *.stp files from CATIA, to be opened in SOLIDWORKS. In CATIA there is a check box (circled below) in the round sweep feature that is unchecked by default. Ensure that this check box is checked for **ALL** tubes to ensure the individual parts export correctly. If any of the tubes do not have this checked it will affect the entire group of solids.

Individual Part File Creation:

Once your design has been approved for manufacture, we require each part be supplied as an individual part file. This forces you to review each tube to ensure the design intent is achieved and also speed up order processing on VR3's end. Ensure the part numbering is consistent between your drawing and model to avoid confusion at weld assembly. We directly extract the 3D CAD data and resulting tool path data to profile tubes from these individual part files. Ensure they are correct and match your design intent.

To create individual part files, right click on a tube within your model and select 'Insert into New Part'. You can also do this through the cut list item folders by right clicking on the body (blue cube icon) within the cut list item folder.

Tube bodies must be numbered in their cut list item folders as explained in the Cut list Setup Section. This is the file name each body (tube) will receive when saved.

'Save Bodies' is a command that can speed up this process, but with inconsistent results. Sometimes the part files saved out are empty or broken. If using this method, ensure you check the individual part files afterwards without the main model open at the same time. If the parts have any issues, this method likely wont work.

Appendix:

Roll Hoop Examples:

Here are some examples of what main roll hoops look like for Formula SAE projects:

It is preferred that roll hoops have all of their bends in a single plane for easier shipping and lower shipping costs to customers. However, we are fully capable of bending multi-plane bent tubes as long as our guidelines are followed.
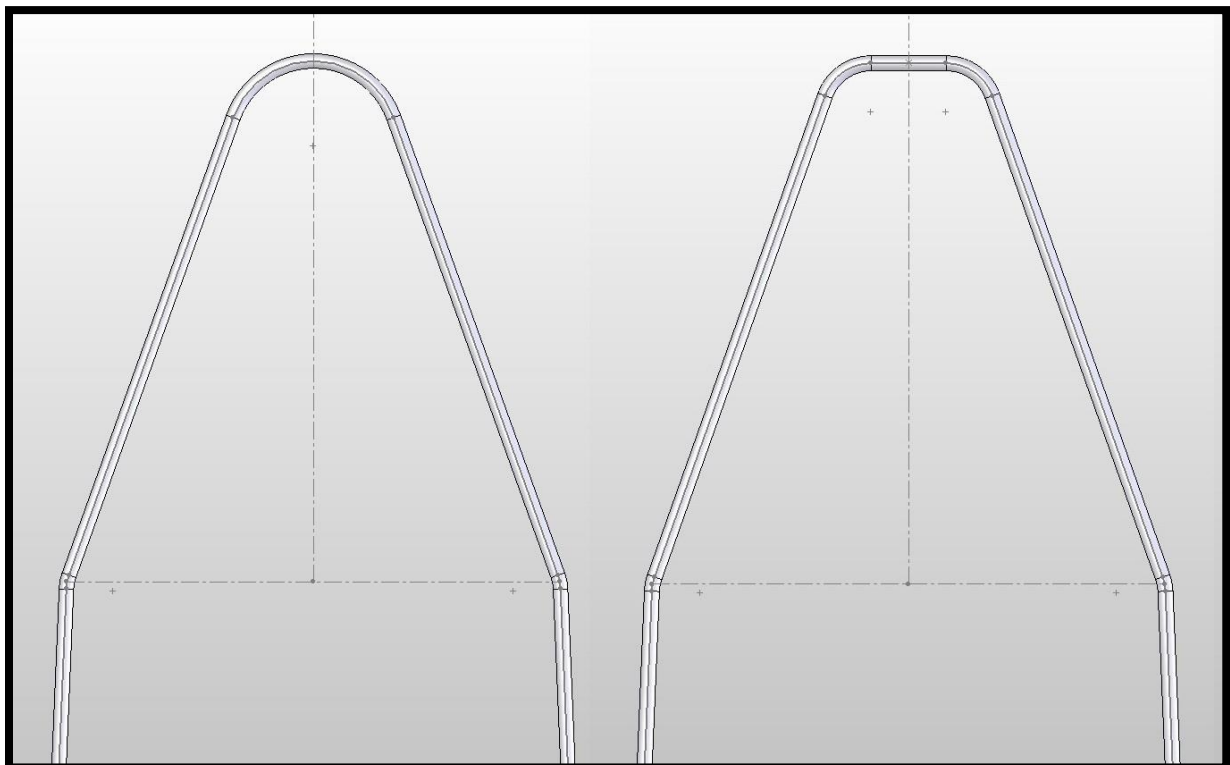
Good practice for the bottom of bent roll bars is to leave the roll bars a bit long and trim all of the other tubes to it. Another good practice is to mitre the bottoms of the roll bars with the cross tubes.

For additional information and a full list of constraints, bend die sizes, and best practices please see the Bending Guidelines section.
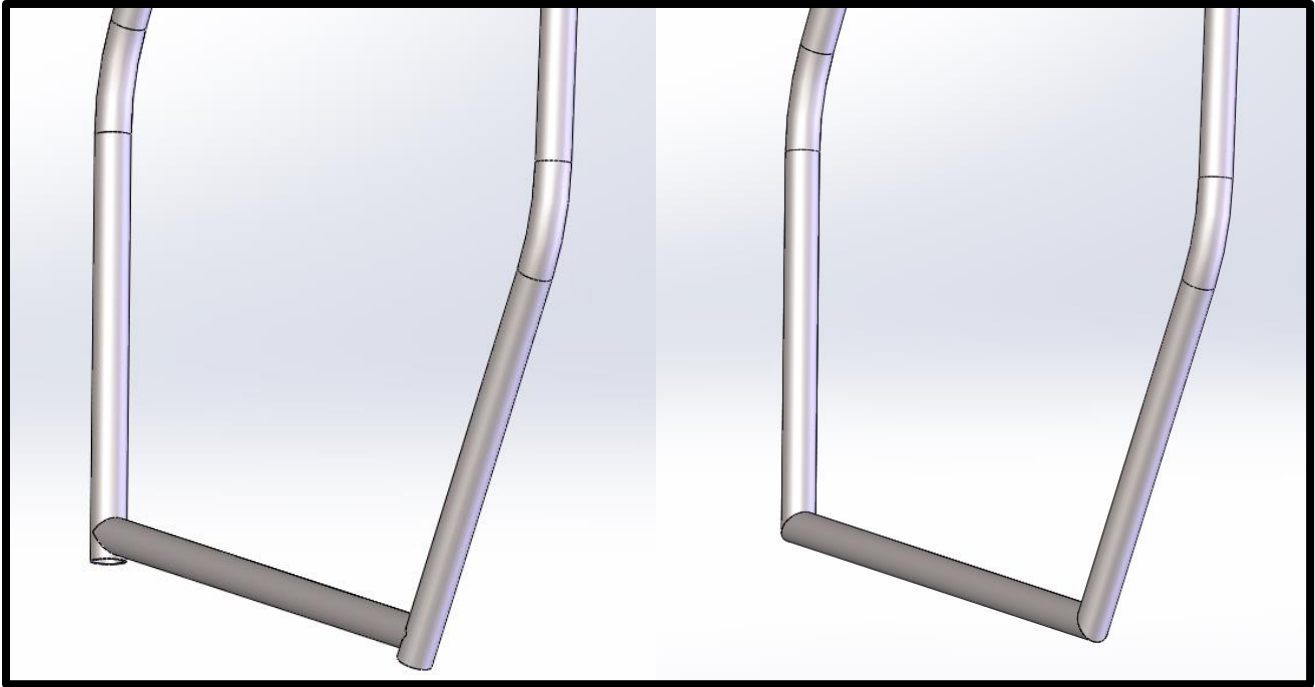
Square Tube Sizes:

VR3 Tube Size Inventory
4130N SQUARE   Aug 2021

Square Tube (inches)	(mm)	Wall Thickness (inches) (equiv mm)	0.028 0.71	0.035 0.89	0.049 1.24	0.065 1.65
0.375	9.53			*		
0.500	12.70			*		
0.625	15.88			*		
0.750	19.05			*	*	*
0.875	22.23				*	
1.000	25.40			*	*	*
1.250	31.75					*
1.500	38.10					*
2.000	50.80					*

Round Tube Sizes:

VR3 Engineering
precision tube fabrication

VR3 Engineering: 4130N Round Tubing Chart
(normally stocked sizes)

VR3 Doc No. WEB4130NRT01   **Last updated:** Mar 2021

Tube OD (inches)	(mm)	(inches) / (equiv mm)	0.028 / 0.71	0.035 / 0.89	0.049 / 1.24	0.058 / 1.47	0.065 / 1.65	0.083 / 2.11	0.095 / 2.41	0.120 / 3.05	0.188 / 4.78
0.188	4.78		*								
0.250	6.35		*	*			*				
0.312	7.92			*		*	*	*			
0.375	9.53		*	*	*	*	*	*			
0.437	11.10			*		*	*	*			
0.500	12.70		*	*	*	*	*	*	*	*	
0.562	14.27			*			*				
0.625	15.88		*	*	*	*	*	*	*	*	
0.750	19.05		*	*	*	*	*	*	*	*	*
0.875	22.23			*	*	*	*	*	*	*	
1.000	25.40		*	*	*	*	*	*	*	*	
1.125	28.58			*	*	*	*	*	*	*	
1.250	31.75			*	*	*	*	*	*	*	
1.375	34.93			*		*	*	*	*	*	
1.500	38.10			*	*	*	*	*	*	*	
1.625	41.28					*	*	*	*	*	
1.750	44.45				*		*		*	*	
2.000	50.80				*		*		*	*	

For more information on your tubing requirements, please contact: **VR3 Engineering / (519) 273 - 6660 / info@vr3.ca**

Bend Die Chart:

	VR3 Engineering precision tube fabrication	**VR3 TUBE BENDING CHART - FSAE and BAJA SAE** Last Updated On: Sept 6, 2021			

Purpose: This document lists the available bend dies for commonly used tube sizes in FSAE/BAJA Projects. Our bend capabilities are not limited to the below listed sizes, and radii - refer to notes/contact us.

Tube OD (in.)	Min. Tube Wall Thickness (in.)	Centerline Bend Radii (in.)	Min. straight length between bends (in.)	Min. end leg length (in.)
0.500"	≥ 0.049"	1.60"	3.00"	2.00"
0.750"	≥ 0.058"	1.81"	3.25"	2.50"
	≥ 0.049"	2.64"	4.00"	2.50"
	≥ 0.049"	3.88"	4.00"	2.50"
	≥ 0.035"	6.00"	4.00"	2.50"
1.000"	= 0.095"	3.25" / 5.88"	4.00"	3.00"
	= 0.065"	3.25" / 5.88"	4.50"	3.00"
	= 0.049"	5.88"	4.50"	3.00"
	= 0.035"	5.88"	4.50"	3.00"
1.250"	= 0.065"	4.41" / 5.96"	4.50"	3.50"
	= 0.049"	5.96"	4.50"	3.50"
	= 0.035"	5.96"	4.50"	3.50"

Notes:

1. Tube bends are made with fixed radius bend dies that are not adjustable.

2. Thinner wall tubes (.035", .049", .065") require larger bend radii / dies to avoid flattening or buckling or other deformation unless, willing to use expensive mandrel bending equipment, and deal with associated setup costs.

3. Refer to our bend guideline, checklist documents for additional information.

4. Our 3D Modeling Guidelines document (available on website - documentation - reference documents), covers every aspect of CAD modeling to ensure manufacturibility of your design.

5.**Avoid notches 'within' bend features if possible! - this increases the steps, and cost of manufacturing the part. Plus, this requires manual layout/fitting/trimming at weld assembly.

6.**Minimum end leg length: Bending requires a minimum straight end length to hold the tube around the die; not having this can lead to extra steps in manufacturing, and additional costs to the process.

7. 'End lengths' less than minimum are okay in many cases. In this case, VR3 would add an end extension(s) and 'scribe' the trim line for the notch to facilitate trimming after bending by the student team or fabricators.

8. Listed above are the common tube sizes, and bend dies typically used for FSAE/BAJA projects.

9. Contact us for other sizes or special features; we can review them, and make necessary suggestions.

10. Refer to our-website-documentation-Manual tube bending tool list, for other available bend dies for one-off parts

11. Automated cnc mandrel bending is available for thin wall tubing, and production bending with additional costs.

12. Our bending capabilities are not confined to the above listed specifications. So, if your project requires other tube sizes, contact us to know if we can make it happen!