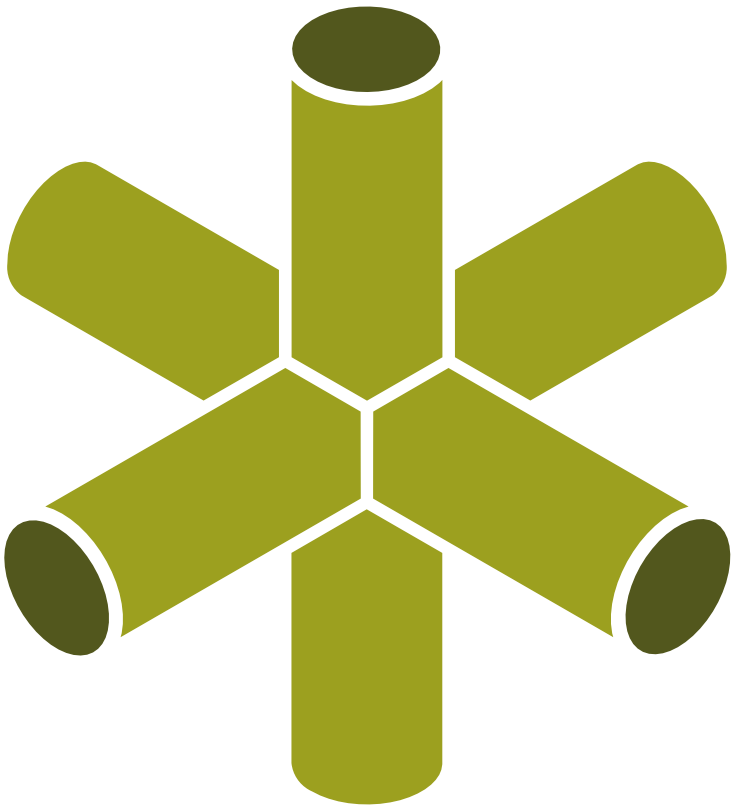
Cartesian Tube Profiling

3D Modeling Guidelines



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Design Checklist

Review this checklist and ensure that your design follows the guidelines to ensure an accurate and timely quote and tube kit. The requirements listed are explained in detail in the following pages.

1. Weldment profiles need to match the material. Ensure wall thickness offset are correct, and geometry is accurate and For square tubes, ensure that corner radii are correct.
2. Ensure that all individual parts have a unique tube number in your cutlist. Match the part number to the balloon number when possible.
3. Please ensure that your assembly drawing is legible. Balloon lines should not cross, and need to be able to read clearly. Please avoid the use of Auto-Balloon functions.
4. Use the provided Sample Material List as a basis for your tube kit material list to ensure that you include Quantity, Outside Dimension, Wall Thickness, tube type, approximate length, and any mid part features (holes, slots or bends).
5. Confirm that all nodes are trimmed correctly and in the order to be manufactured. Check that there are no hollow nodes or extended trims.
6. Our process takes the profile path directly from your model. So the cut part is entirely dependant on the accuracy of the 3D data provided to us. Ensure that each tube is included, and modeled correctly (tube size, wall thickness, etc in model matches design intent).
7. All bent tubes must follow our bend contraints: bend radii for a given wall thickness, distance from bend to end of part, distance between bends, bend profile tangency.
8. If using a 3D model software other than SolidWorks, your neutral format files are exported as solid bodies, NOT surface bodies.
9. Square tube require a minimum of 0.010" offset from the inside face for profiles.

Request for Quote: Checklist

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

1. **Send**
   - ☐ pdf of ISO weldment drawing with balloons.
   - ☐ Material List (with all required tube information).
   - ☐ Entire model (Weldment)
2. **We review model**: Trims, formed tubes, etc and provide quote.
3. **Once the model is approved**: Send individual 001.sldprt files (numbered per balloon and BOM)

The files are described in detail in the following sections.

Inquiry Documents

In order to provide an accurate and timely quote, we require a specific set of information about your project or design. If you do not currently have a 3D model, we are able to provide technical support. Please see the Services page on our website.

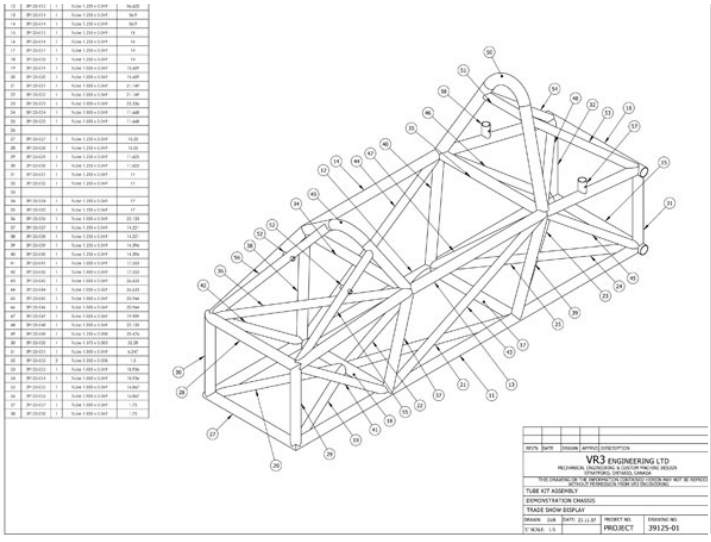
The information relies on two key aspects, an overall assembly drawing of the tube structure and the material list. This information clearly and quickly defines the scope of supply, primary features and part numbers, and also allows us to quickly and easily submit a proposal.

1. Material List noting each part in the weldment assembly

	A	B	C	D	E	F	G	H	I
1									
2	**VR3 Engineering Cutting Summary**								
3									
4	**Customer**			Univ of xxxx					
5	**Project**			xxx					
6	**Description:**			Tube Set: FSAE 2012 chassis					
7	**Customer dwg/file number**			xxx					
8	**Date**			Jan-11					
9									
10									
11	Part No	Item No:	Rev.	Qnty	Dim 1	Type	Dim 2	Dim 3	Additional
12	(customer)	(assy dwg)			outside dim	(rd,sq)	wall thickness	aprox length	Features
13					(in.)		(in.)	(in.)	
14									
15	xxxx	11		1	1.250	rd	0.049	57.00	
16		12		1	1.250	rd	0.049	57.00	
17		13		1	1.250	rd	0.049	56.00	2 bends
18		14		1	1.250	rd	0.049	56.00	2 bends
19		15		1	1.250	rd	0.049	16.00	
20		16		1	1.250	rd	0.049	16.00	
21		17		1	1.250	rd	0.049	14.00	
22		18		1	1.250	rd	0.049	14.00	
23		19		1	1.000	rd	0.049	16.00	
24		20		1	1.000	rd	0.049	16.00	
25		21		1	1.000	rd	0.049	21.00	
26		22		1	1.000	rd	0.049	21.00	
27		23		1	1.000	rd	0.049	24.00	
28		24		1	1.000	rd	0.049	12.00	
29		25		1	1.000	rd	0.049	12.00	
30		26		1	1.000	rd	0.049	17.00	
31		27		1	1.250	rd	0.049	16.00	
32		28		1	1.250	rd	0.049	16.00	
33		29		1	1.250	rd	0.049	12.00	
34		30		1	1.250	rd	0.049	12.00	
35		31		1	1.250	rd	0.049	11.00	
36		32		1	1.250	rd	0.049	11.00	
37		33		1	1.000	rd	0.049	17.00	
38		34		1	1.250	rd	0.049	17.00	4 slots

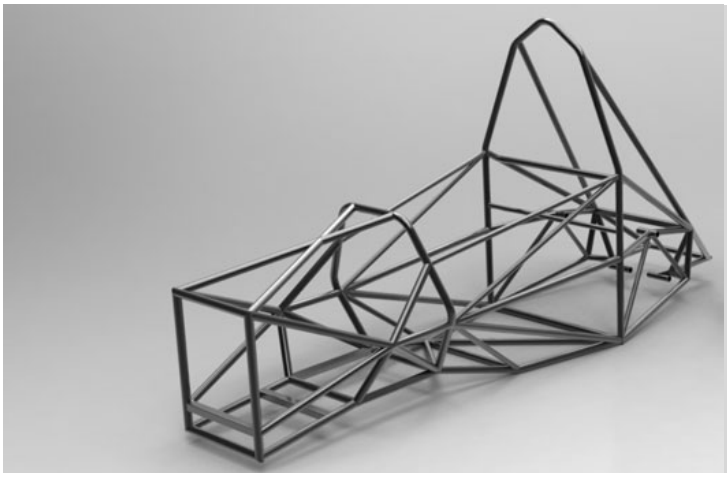
- This document should match the cutlist produced and supplied in the assembly drawing.
- The following information needs to be supplied for each part: Quantity, Outside Dimension, Wall thickness, tube type, approximate length, and any additional features.
- Please use the BOM template file supplied in the Reference Document section of the website.
- This document is mandatory for us to create an accurate and timely quote.

2. Assembly drawing with the cutlist/material list



- This drawing requires balloons that clearly indicate each member of the weldment and is referenced by the Bill of Materials.
- Each part requires a unique part number and will be used to ensure part numbers and model match the specifications supplied. Mirror parts are not identical.
- Create this drawing from the 3D model, save as a PDF file. Please send us the PDF file version only.
- See an example of an appropriate assembly drawing in the Reference Document section of the website.

3. 3D Solid body model of the tube structure with properly defined geometry



- In the quoting process, this model is primarily used to double check that bending standards and geometry have been properly defined.
- This is to be supplied in addition to the two documents above and does not replace them. In order to receive a accurate and quick quote, we require the two documents listed above and will use the solid body model to ensure that all specifications and constraints are met.

Preparing Solidworks for Modeling

SoldidWorks is a very useful program for design tube kits and weldments. Spending time to setup the program to be optimized for weldments will reduce the amount of time required to produce the information needed for quoting.

Weldment Setup: Properties File

The Weldment properties file is a text file which determines what options will be available in the dropdown menu for a cutlist item. This information is used when creating automatic cutlists

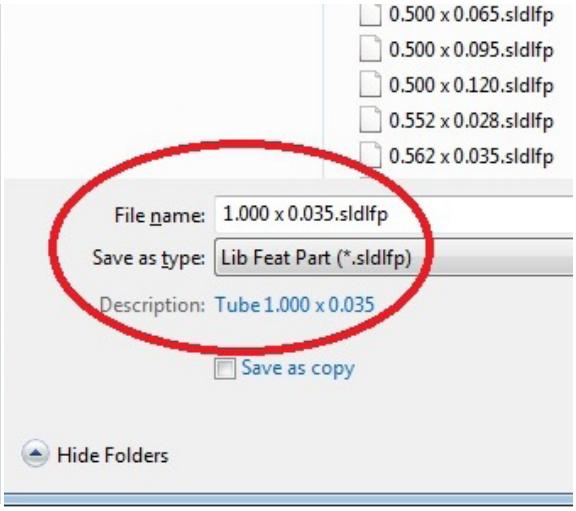
1. Find your weldment properties file > Tools\System Options Tab\File Locations\Show Folders for: Weldment Property (This will show the folder in which your weldmentproperties.txt file should be located. Typically will be in C:\SolidWorks Data\lang\English)
2. If there is no folder shown, then you will 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: Creating Library Feature Parts (.sldlfp)

1. Create a new part in SolidWorks.
2. Create a sketch on the front plane.
3. Sketch: Create the Outer Diameter (OD) of the tube. Offset the OD inward by the wall thickness. Ensure that you offset inward, as the inside radius is not equal to the outside radius.
4. Save File
5. File\Save As..
6. File name: 1.000 x 0.035
7. Save as type: Lib Feat Part (*.sldlfp)
8. Description: 1.000 x 0.035
9. Repeat steps above to complete your weldment profile library

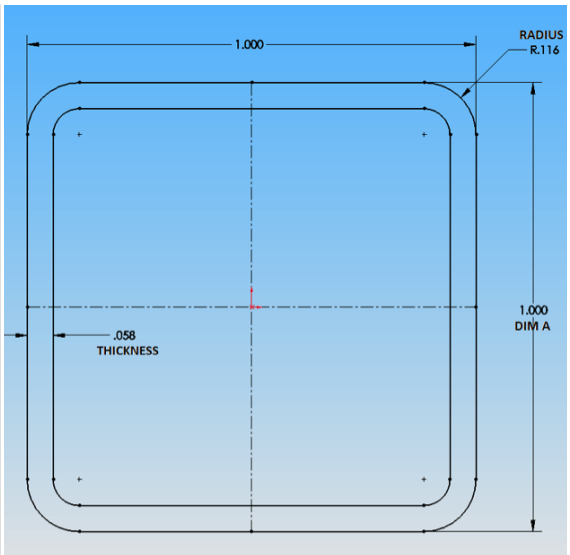


Weldment Profiles: Selection and Use

We have a large selection of Chromoly 4130N in stock. We recommend using round tubing because it is more readily available and significantly cheaper.

Sometimes square tubing is required. To insure that the tubes are modeled as accurately as possible, here are the profiles which must be followed when doing square tubes.

SQUARE TUBES		
DIM A	**THICKNESS**	**OUTSIDE RADIUS**
0.750	**0.035**	**0.066**
0.750	**0.049**	**0.093**
0.750	**0.058**	**0.110**
0.750	**0.065**	**0.125**
1.000	**0.035**	**0.066**
1.000	**0.049**	**0.093**
1.000	**0.058**	**0.116**
1.000	**0.065**	**0.123**



Weldment Modeling: Step-by-Step Instructions

The 3D model, drawings, material lists and design must be well organized and properly presented. Anyone can create a 3D model 'cartoon'. It takes additional time and effort to properly prepare, design, organize and present this information for us to efficiently manufacture the components.

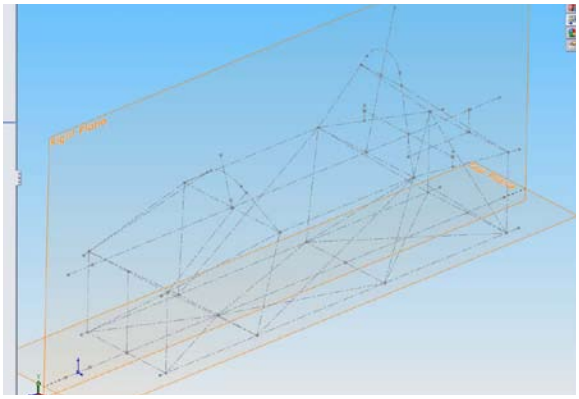
- Our main concern is generally regarding the tubes with bends. The bends must be modeled with centerline bend radii that match our dies and some other basic constraints.
- Ensure all trims are complete, especially at multi-tube clusters.
- Ensure left side and right side parts are assigned separate part numbers. Do not mirror parts! Mirrored profiled tubes do not have identical tool paths.
- Ensure the tubes are modeled as continuous solid bodies. Not surfaces or splined curves. All segments of a curved tube must be modeled correctly. Do not use fixed points.

Weldment Modeling Procedure

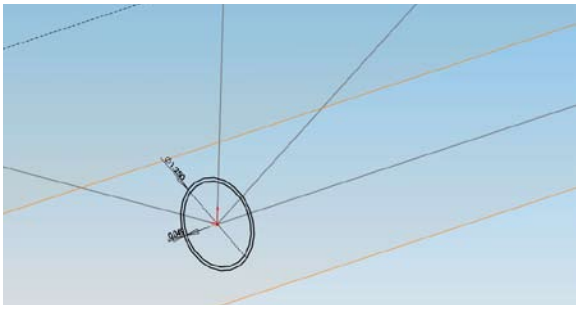
This guide is to assist designers in preparing information suitable for our tube profiling process. These methods and suggestions are specifically for Solidworks but should be applicable to other 3D cad packages.

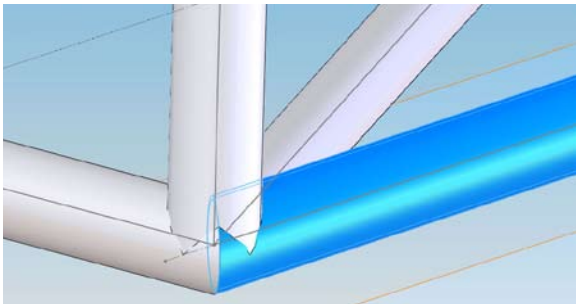
Tubing structures are best suited to be built as a 'weldment' part. Some advantages are:

- 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
- profile library of tube sizes is convenient for creating a more 'automatic' BOM
- one file to update with any changes to tube sizes or geometry



3D wireframe made using reference planes, fully defined sketches and sketch line segments for tube centerlines.





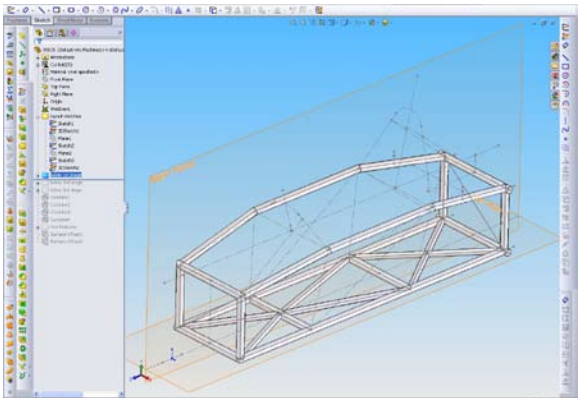
Structural Member feature using weldment library profiles.

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.
2. Create a 3D wire frame representing the tube centerlines of the major structural tubes ensuring the parametric features are fully defined as the structure grows.
3. Build the tube structure on the solid model generally in the same order as one would build the structure on the shop floor.
4. Define formed tubes in unique sketches/planes.
5. Add the tubes as 'Structural Members' with the correct od and wall thickness. Do not combine lines and paths into one solid body until the structure is complete. The trimming features are generally more reliable this way.
6. Use 'Trim/extend' to trim each tube as it is added to the existing structure. This is less confusing than determining the trim sequence with all tubes left untrimmed in the structure.
7. Ensure the parametric design features and geometry are fully defined as you proceed. (Fully define sketches)
8. Avoid 'mirroring' tubes. Mirrored tubes generally have unique profile paths and require separate profile programs to produce. Trim features are more robust. Changes are easier to apply downstream.
9. Once all the tubes in the solid model are added and trimmed, combine the tube segments as required to produce single tube components. This applies mostly to formed or bent tubes.
10. Assign each tube in the solid model a unique part number. Using 'SolidWorks', this is done in the cutlist.
11. At this point, the 3D model is just a 'cartoon'. The required information to produce this structure is not readily accessible or organized and the dimensional design cannot be checked easily.

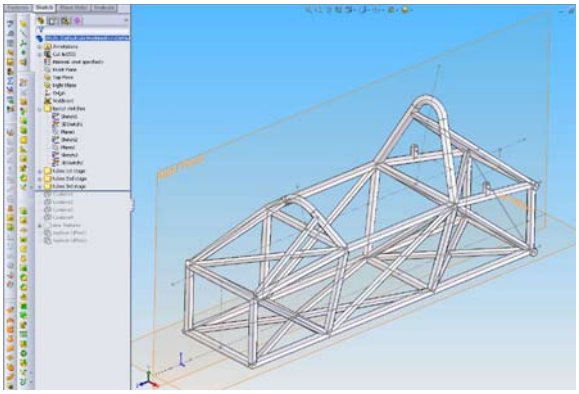
12. A weldment drawing with material list must be created to itemize the number of pieces. Create a drawing with an isometric view, bill of material and add balloons for each tube component.
13. Create a drawing with the 3 main views (side, front, top) plus any important section views. Add overall dimensions, station dimensions and layout dimensions to verify the design configuration. These dimensions will also be required to assemble and build the tube structure on the weld table.
14. Review and check the design:
    - confirm the design dimensions
    - confirm tube sizes
    - confirm all tubes are shown
    - review all trims at tube clusters
15. Confirm that bend radii and tube sizes are compatible with available bending dies and manufacturing procedures.
16. Produce a part file for each tube in the weldment maintaining the parametric link to the solid model weldment.
17. Open each part file for the round straight tubes. Visually check the end trims to confirm end trims. Save this file in 'iges' format which will no longer have a parametric link to previous files. This file must represent a single solid body. Name as unique balloon number. Desired format: 5 digit weldment number - three digit part number (99999-001.iges).
18. Square tubes require an additional step to round tubes. Verify radii and offset requirements for the square tube cross section.
19. Tubes with Bends:
    - Send the formed tubes as an iges file without adding a new coordinate system and accompany this with a drawing file.
    - VR3 will import this into Solidworks, 'straighten' the tube while maintaining the profile orientations, adjust the length to suit bend allowances, profile the tube and then bend the tube into the final shape.
    - Please review bend features with us prior to approving the final design to ensure compatability with bend dies and manufacturing procedures.
20. Send the iges files and a material list to us for manufacturing the profiled tubes.
21. From this file we can extract the profile data and generate the cnc tool path for the tube profiler.
22. We will profile, label, package and send a set of tubes ready for weld assembly

Notes:

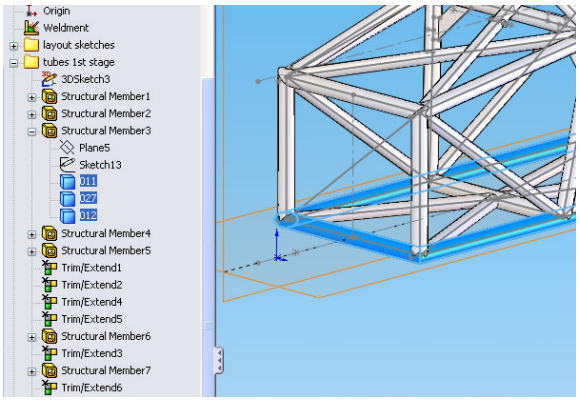
Feathered faces: Our tube profiling process always cuts the tube perpendicular to the tube surface. Our programming path will eliminate the 'feathered' edges which result from 3D modelling. This perpendicular cut to the surface is ideal for welding and results in a complete tube to tube contact around the perimeter.



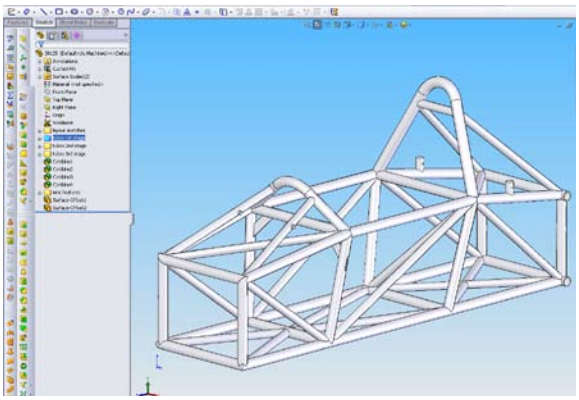
Weldment modelled and trimmed in the same order as the shop floor build.



Continue to insert and trim structural members in the same order as the desired build.



Each tube in the requires a unique part number (prefered three digit number) that links to the BOM and assembly dwg.



Review and check design for correct tube dimensions, sizes, confirm all tubes shown and trims are correct and without error.

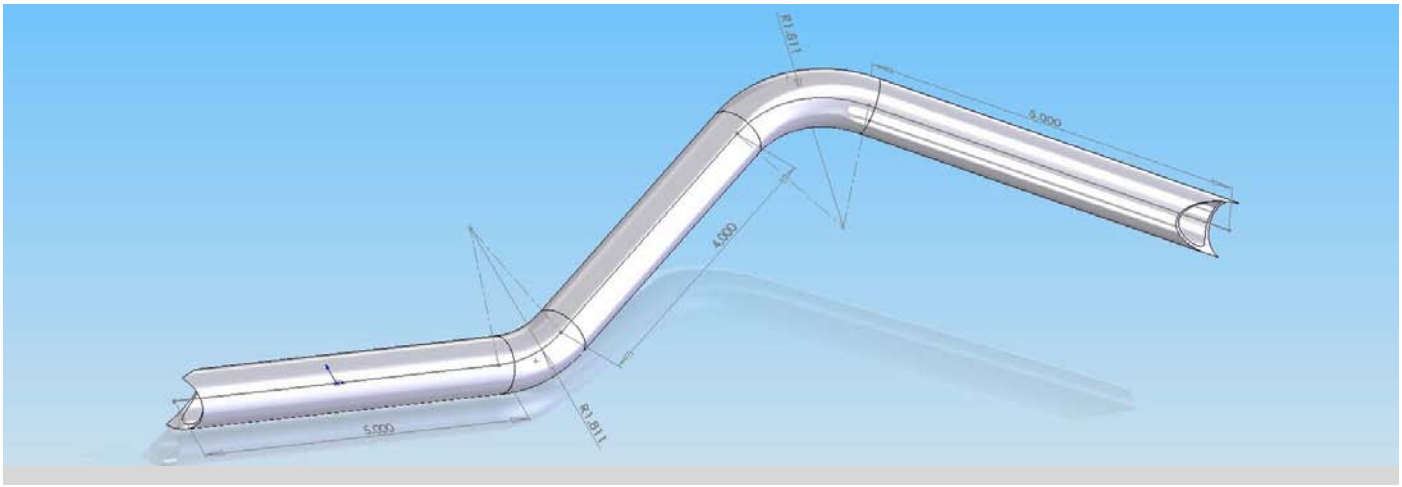
Bending: Information and Guidelines

Our tube kits include the formed or bent round tube components in most cases. The value and convenience of our combined profiled and formed tubes is exceptional. However, there are restrictions and limitations. Note that we profile the tube as a straight tube and then bend with the correct orientations and bend allowances. The critical step is setting up the procedure. There are many factors and variables in the bend process that require concise procedures and a knowledgeable operator. However, once it is set up, consistency and repeatability are very good. Bending is about 80% science and 20% art. We can produce formed tubes with multiple bends in different bend planes subject to restrictions. However, 'simpler is better'.

1. Bending Constraints

- All of the tubes are bent with an electric rotary bender. There is no mandrel on the bender so there are limitations to the size of the bend radius relative to the thickness of the tube.
- Bend radii must be selected and modeled corresponding to the dies we have available and appropriate for the tube od / wall thickness / bend angle required. See the attached document.
- There is also a minimum distance between bends of 4-5" typically for out of plane bends. This distance can be slightly reduced if the bends are in the same plane.

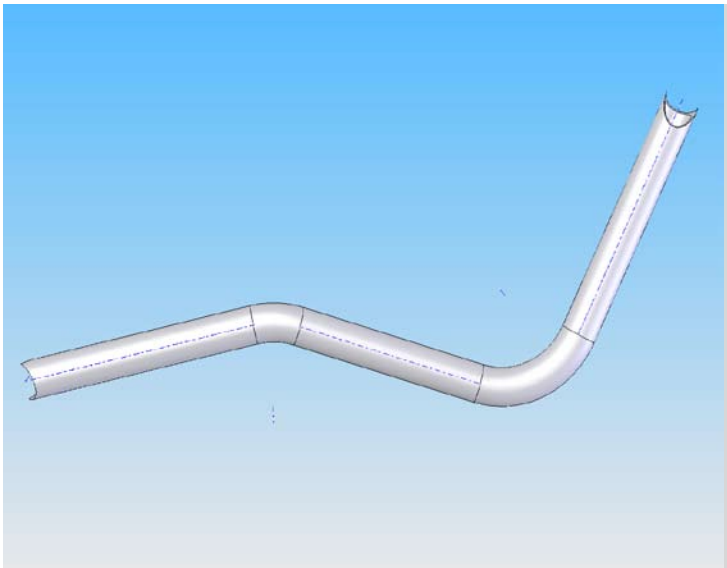
2. Example Bent Tube Component



Example: 0.750" od x 0.058" wall tube. It has 2 bends which are out of plane from one another; this is why there is a minimum length of 4" between these bends. There is also 5" minus the profiles on the end of the tube to allow for bending.

The centerline radii of these bends are 46mm/1.811"; this is allowed because the tube has a 0.058" wall. The thicker the tube, the easier it is for us to bend. If this tube was 0.049" wall, we would not be able to bend it with the 46mm/1.811" radius; the radius would have to be larger; you would need to use the 67mm/2.638" radius.

3. Modeling considerations for bent tubes



All the radii in a bent tube need to be tangent to the straight sections before and after the radii! It is geometrically impossible for us to bend a tube where the bends are not tangent to the straight sections. An easy way to check if your bent tubes are correct is to make a 3Dsketch with lines on the axis; all of the lines should be able to connect to one another in succession.

If using a different program we prefer to have all of the tubes as .stp or .igs formats. When saving parts under this generic format, make sure that you are saving your solid bodies as solid bodies (Not surface bodies).

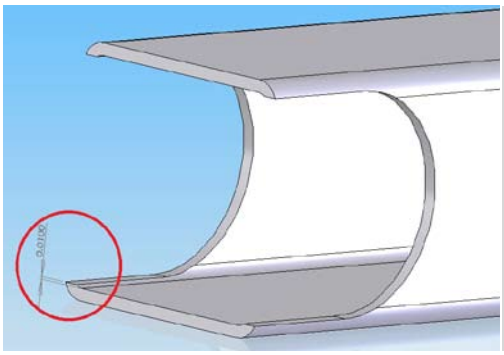
How to check: Should have a temporary axis (means that it is a solid body).

Bending: Die Chart

The following are constraints and preferences of our tube bending process when combined with end profiles:

1. Tubes are formed using bending dies specific to the tube outside diameter and centerline radius.
2. Thicker wall tubes can be bent to tighter radii.
3. Thinner wall tubes have minimum bend radii and limited bend angles.
4. Bend angles greater than 90 degrees require thicker walled tubes.
5. Shallow bend angles less than 20 degrees can sometimes be achieved with thinner wall tubes.
6. Minimum distances between bends are required for multiple bends, typically about 4.00" inches.
7. Minimum straight tube lengths of aprox 4.00" are also required from the end of a tube to the start of a bend. This is required to properly support the tube during bending.
8. Minimum bend radii are also subject to the material properties.
9. Refer to the chart for bending dies currently available.
10. If possible by design, consider using the larger bend radius.

Tube OD (in.)	Centerline Bend Radii (mm/in)	Minimum wall thickness (inches)
		(in.) (mm/in.) based on 90 degree bends. Thinner walls are not recommended but may be achievable on bends < 20 deg
0.375"	36mm / 1.417"	>or = .035"
0.500"	36mm / 1.417"	>or = .049"
0.625"	46mm / 1.811"	>or = .049"
0.750"	46mm / 1.811"	>or = .058"
	67mm / 2.638"	>or = .049"
	98.4mm / 3.875"	>or = .049"
	6.000"	>or = .035" *new/special
0.875"	46mm / 1.811"	>or = .083"
	67mm / 2.638"	>or = .049"
1.000"	56mm / 2.205"	>or = .120"
	67mm / 2.638"	>or = .095"
	82mm / 3.228"	>or = .058"
	149.2mm / 5.875"	>or = .035" *new/special
1.125"	67mm / 2.638"	>or = .083"
	82mm / 3.228"	>or = .058"
1.250"	82mm / 3.228"	>or = .083"
	112mm / 4.409"	>or = .049"
1.375"	82mm / 3.228"	>or = .083"
	112mm / 4.409"	>or = .049"
1.500"	100mm / 3.937"	>or = .083"
	150mm / 5.906"	>or = .065"
1.625"	130mm / 5.118"	>or = .083"
1.750"	150mm / 5.906"	>or = .083"
	170mm / 6.693"	>or = .083"
2.000"	190mm / 7.480"	>or = .083"
½" pipe (.840" od)	R56mm (2.205"	
¾" pipe (1.050" od)	R67mm (2.638")	
1" pipe (1.315" od)	R67mm (2.638")	
	R112mm (4.409")	
1 ¼"pipe (1.660"od)	use 1.625" OD tube die (R130mm / 5.118")	



- We can also produce profiled square tubes. Square tubes can be very beneficial as design features for mounting components onto the chassis structure.
- Cutouts on square tubes need to have a minimum 0.010" offset from the inside face.

We suggest using round tubes wherever possible. Consider the following:

- Our preferred sizes are 1.000" square x .035", .049" or .065" wall thickness.
- See the Weldment Profiles: Selection and Use for sketch profiles noting the required corner radii.
- Square tubing raw material is much more expensive than round tube raw 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 or bent.
- Square tubes 'nest' onto round tubes 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.

The sequence and options used when trimming is incredibly important to the weldment design. Below are several common areas where errors occur in weldments and need to be done in the correct method and sequence. **In some situations, having incorrect trims negates having CNC profiled tubes.** Proper trims will allow for easier and more accurate fabrication. Please review the situations below carefully to ensure that you follow the proper procedure. The majority of models which we receive have most of their errors in the trims. Missing trims, improper trims and hollow nodes are the most common mistakes.

General Trimming Procedure

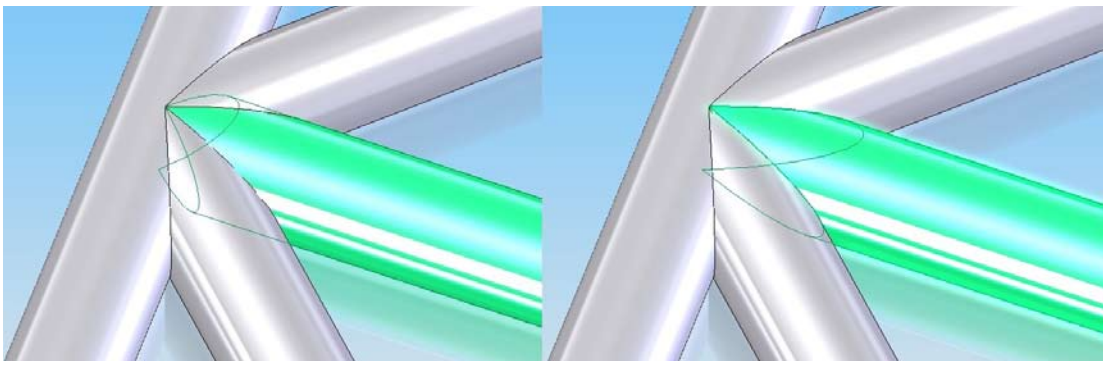
In SolidWorks, the general procedure for trimming is as follows. In certain situations, different options are used in order to create a proper trim.

1. As you build up the weldment, create and place structural members (tubes) in the order that you will be fabricating the structure.
   - Use a unique structural member for each tube. Do NOT put all of the same size tube in one structural member definition. Using a unqie structural member allows for easier change of tube size.
   - Trim the tubes as you create structural members, rather than doing all of the trims at the end of the design tree. This limits missed trims in tube clusters.
   - Trim in the order you will place tubes. Ensure that there are no hollow nodes in any cluster.
2. Trim a single tube at a time, both ends. If you are having difficulty with a trim, do one end at a time. DO NOT trim multiple tubes in one operation, this leads to confusion, errors and limits ability to parametrically change the model. Having organized trims allows for ease of modeling and verification.
3. For options, use the 'Trim to Body', do not 'Allow Extension' settings as a base point. If trimming to bodies does not create a correct resultant trim, set the Trimming Boundary to Face/Plane/Surface. Check the bodies to keep and the trimmed bodies to discard. Review that the trims have been completed using the surface are correct.

Specific Trim Issues : Hollow Nodes

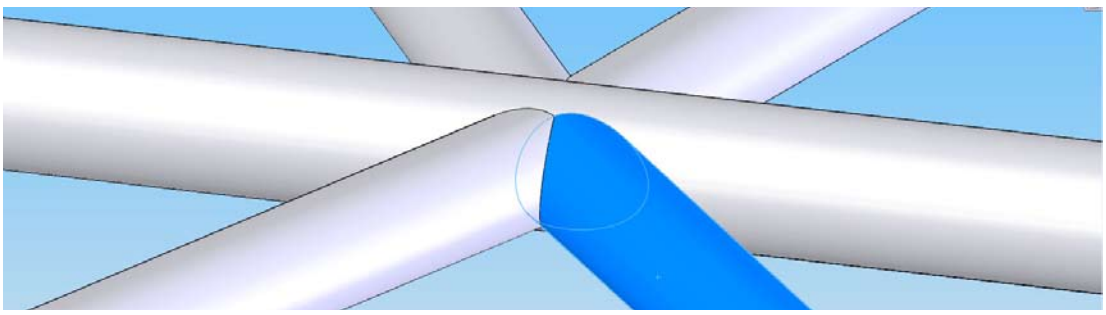
correct trimming order

tubes trimmed to each other, incorrect hollow node



correct trimming order

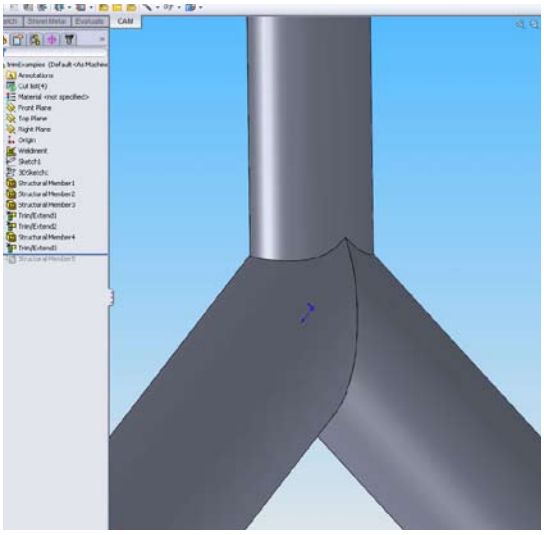
tubes trimmed to each other, incorrect hollow node



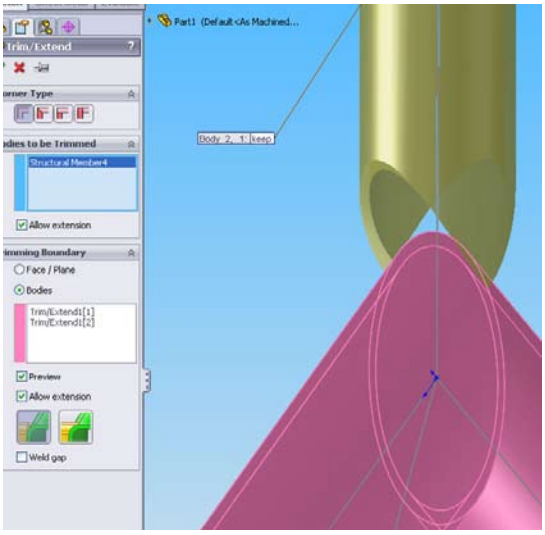
correct trimming order eliminates hollow node

- Hollow nodes occur when all of the tubes are trimmed to all the tubes which they come in contact with.
- This causes seams and nodes which only contact on the edges of the tubes; thus creating hollow nodes.
- This makes the kits difficult to weld and it makes the nodes structurally weak.
- Ensure that the trimming sequence is done in a way that eliminates hollow nodes

Specific Trim Issues : Trim Extension

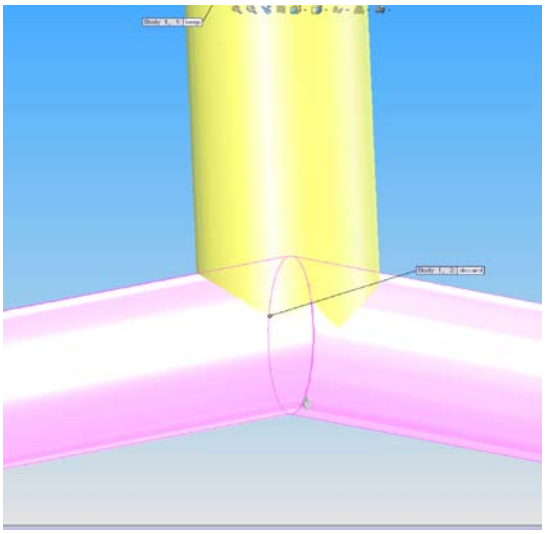


correct trimming options

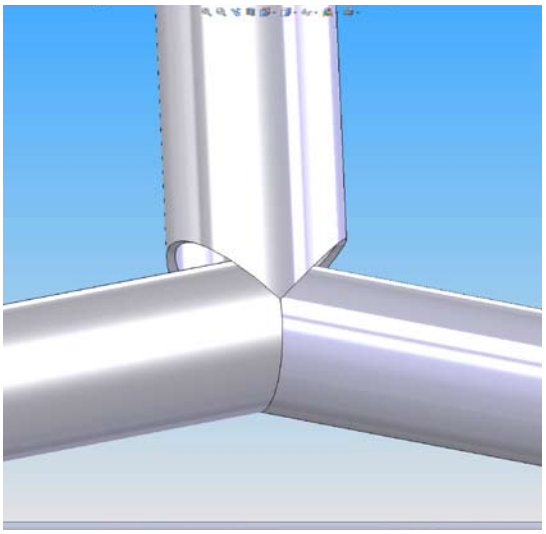


allow extension selected, incorrect trim

- Choosing the 'allow extension' option in the Trim Feature can lead to improper trims in a tube cluster.
- Ensure that all clusters are trimmed properly by visually inspecting trims before you send the file.



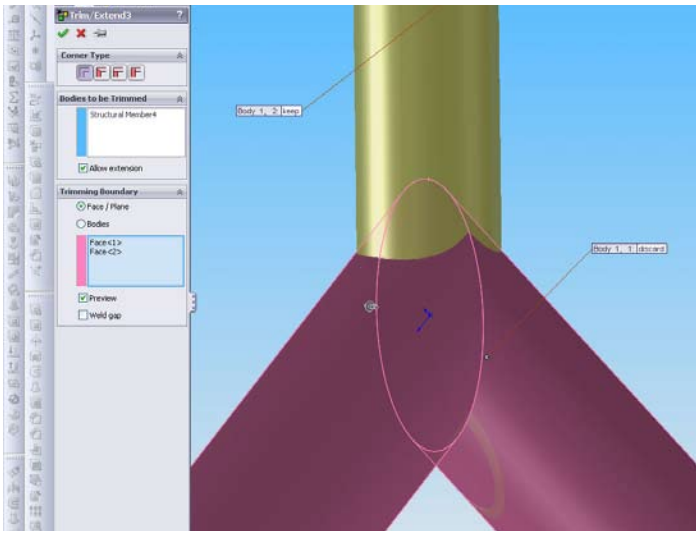
correct trimming options



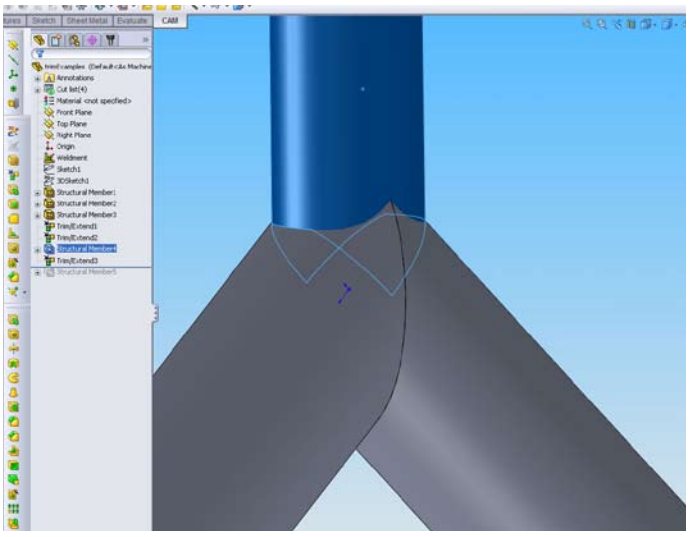
allow extension selected, incorrect trim

Specific Trim Issues : Trim to Mitre

Trimming to a mitre is a specific case where the Trimming Boundary needs to be set to Face/Plane/Surface. Ensure that the trim that is completed is correct.

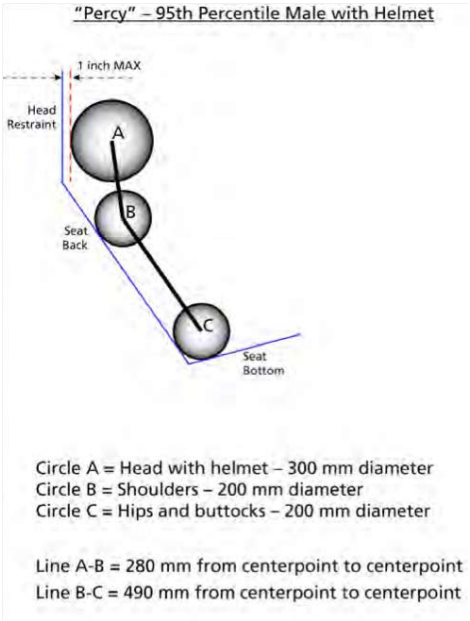


correct trim settings, using surfaces as trimming boundary



incorrect trim, mitre is not properly trimmed to

Creating a Model



Most teams start their suspension node sketches and build their cars around their suspension. We recommend doing a side view of the car and determining the fit of the driver within the car. FSAE has made a model of what a 95 percentile male will look like and call this model "Percy". The figure shows what Percy looks like according to a FSAE Workshop. We encourage putting "Percy" in your right side view of your FSAE chassis design to make sure spacing is correct.

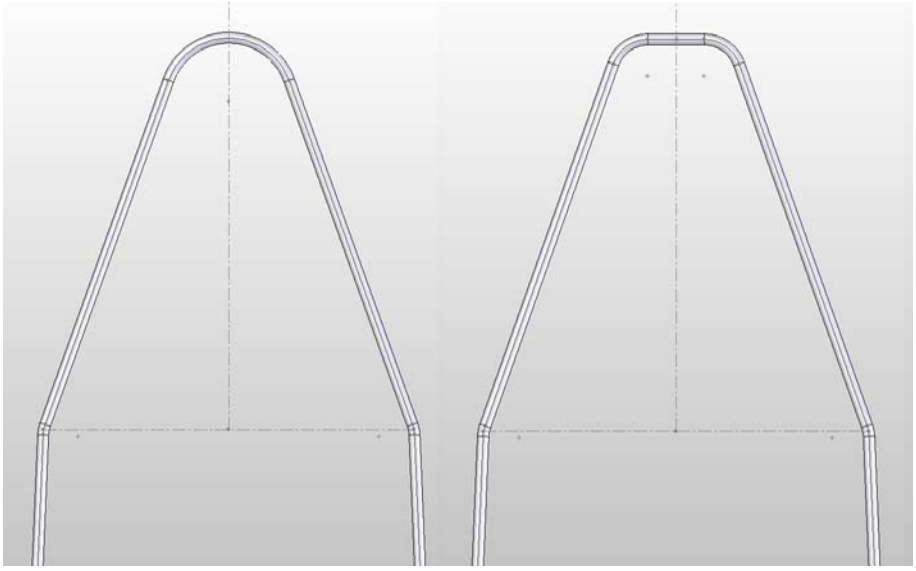
From pg20 of FSAE Workshop 10.25.2008

Fully define the sketches when building the wireframe. Fully defined sketches helps with the stability of the model. Breaking the model down into several sketches is also very important. Some teams attempt to put all of the tubes in a single 3Dsketch and this makes it very difficult to adjust; it also makes Solidworks less stable and makes design changes difficult to implement. We recommend doing a separate 3Dsketch for each of the bent tubes. Bent tubes must follow the "Bending Die Chart: Guidelines" Document. See the Bent tube section of this document for more details and examples.

Student Project Bent Tube specifications

All of the tubes are bent with an electric rotary bender. There is no mandrel on the bender so there are limitations to the size of the bend radius relative to the thickness of the tube. There is also a minimum distance between bends of 4-5" typically for out of plane bends. This distance can be slightly reduced if the bends are in the same plane.

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



Single Large Radius Double Smaller Bend

A 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 miter the bottoms of the roll bars with the cross tubes.

We prefer that Roll hoops have all of their bends in a single plane so that it is easier to ship and lower shipping costs to customers. However we are fully capable of bending multi-plane bent tubes as long as our guidelines are followed.

For additional information and a full list of constraints, bend die sizes and best practices, please see the Bending Guidelines document.

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