

## Suggestions for 3D Solid Modeling of Tube Structures

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This document is to assist designers in preparing information suitable for our tube profiling process. These methods and suggestions should be compatible with most 3D solid modeling packages.

1. VR3 Engineering is most familiar with 'Solidworks' software and produces tube structure as a 'weldment' part file. 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
  - one file to update with any changes to tube sizes or geometryA disadvantage is that the file size becomes large especially when the drawing files are created.
2. 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.
3. 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
4. Build the tube structure on the solid model generally in the same order as one would build the structure on the shop floor
5. Add the tubes with the correct od and wall thickness and extrude along the lines and curves. 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. Trim/extend or cut/extrude each tube as it is added to the existing tubes in the structure. This is less confusing than determining the trim sequence with all tubes in the structure.
7. Ensure the parametric design features and geometry are fully defined as you proceed.
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 curved, 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. An assembly 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 shop floor.
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. If the model is created as a weldment in Solidworks; update the cutlist items to reflect the numbers assigned to each tube individually. This model is ready for us to program.

For tubes with bends (in a weldment):

Use the "combine" feature and select all of the line segments so that the tube is a single item. Do this at the end of the modeling process; rename this cutlist item to its' assigned number. VR3 will '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 desired shape.

Please review bend features with us prior to approving the final design to ensure compatibility with bend dies and manufacturing procedures.

For models created as assemblies: Send the assembly and all tubes associated with it. Make sure that the tubes are "named" as their unique item number.

#### **Models created with other cad programs: Pro-Engineer, etc**

Extract all of these tubes individually and save them as a .iges file and named as their item number. Under the options of saving to a .iges; select to save the tubes as **solid bodies** and NOT surfaces.

**IMPORTANT:**

To test if you are doing this properly; please open up some of your .iges files in Solidworks and under \view turn on "Temporary Axes". You should see temporary axis along the length of the tube member.

17. Send the appropriate files and a material list to us for manufacturing.  
From these files we can extract the profile data and generate the cnc tool path for the tube profiler.
18. 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.