

Friendess, Inc.

Tube Nesting Software

TubeT User Manual

Document version: V2.1

Software version: 1.41.3



Welcome

Thanks for choosing Friendess 3D nesting software TubesT!

Friendess 3D Nesting Software TubesT (hereinafter referred to as TubesT) is a set of nesting software for Friendess numerical control system for tube laser cutting. Aiming at the development of Friendess TubePro/CypTube pipe cutting software, it is capable to realize Parts Creation, Drawings Processing, Weld Compensation, Male-female Joint, Co-edged Nesting etc. Primarily it includes Smart Draw, Quick Import, Graphic Technique, Cutting Path Processing, Auto Nest, Sort, Cutting Files and Report Export and so on.

This manual is based on TubesT-v7.1. 41.3, and due to the continuous version updating, the TubesT you are using may differ in some respects from what is stated in this manual, we sincerely apologize for any inconvenience this may cause.

The latest software function definition and operation skills can be found on the official website of Friendess (www.fscut.com) for enquiry and download.

If you have any questions or suggestions during the use, please feel free to contact us via support@fscut.com!



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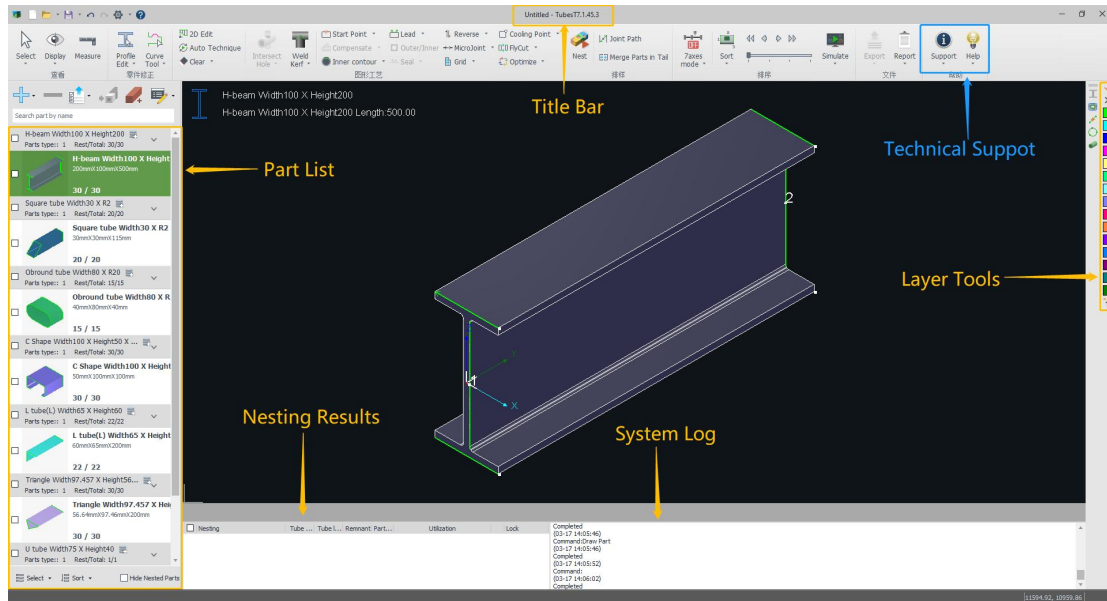
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I. Quick Access

This section mainly brief the TubesT operation flow in simple description. Details and application cases will be illustrated in later sections.

1.1 User Interface



Title Bar (on the top of TubesT) shows software version; Menu Bar is underneath the Title Bar where listed function buttons in groups; To the left is Part List, the right Layer Tools, while Nesting Results and System Log are shown on the bottom.

Double click the part in left list will display full image of corresponding part in editing window, in which the part marked by blue is the one displayed currently, the one marked with yellow icon  indicates it's created by TubesT, double click it to re-modify part in "Tube Draw" workspace.






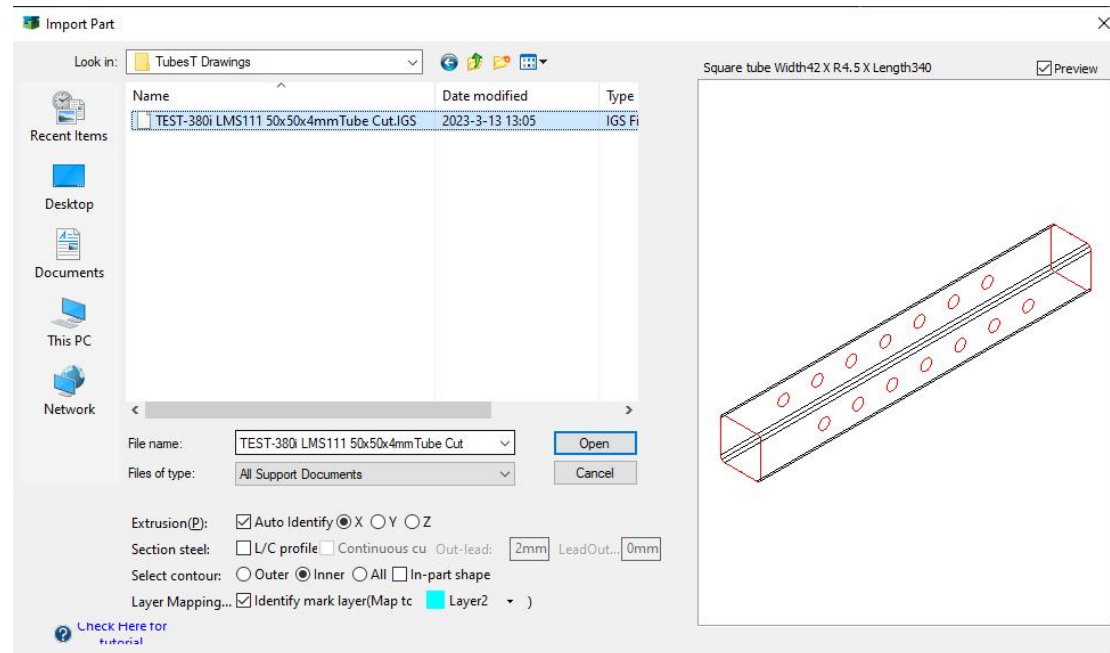
1.2 Operation Procedure

Above chart demonstrates TubesT operation flow, in which each process will be introduced in details below.

1.2.1 Add Parts

Click  in part list and select “Import from File”.

Browse from file folder and select the files needed (igs, sat jhb or jhbs format), finish parts import to TubesT by clicking OK after proper parameters and correct preview confirmed.



Please take notice that “L/C profile” option must be enabled when you’re importing Angle Steel, C-shape steel and I-beam Steel.

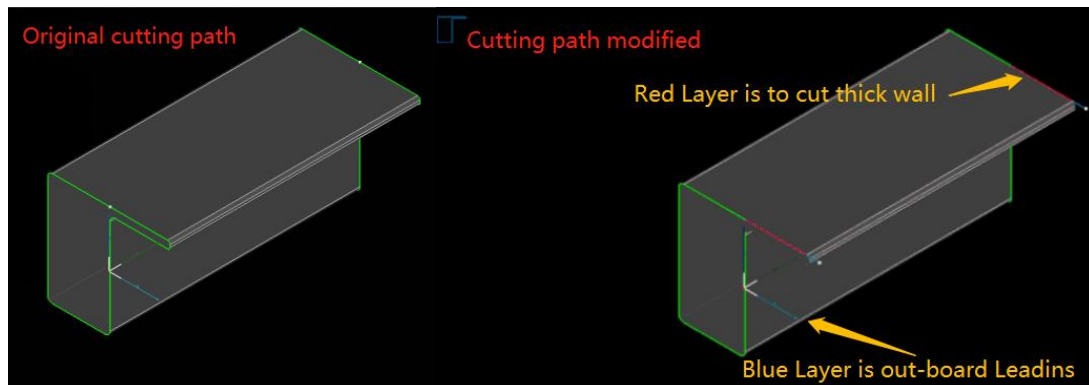
1.2.2 Parts Modification

TubesT allows user to edit part design by rotating part, modifying 3D Vector, Break line, Join Line and Draw Wrap etc.

For example, to make a part design fit for laser cutting, like the shape in below demonstration, it is necessary to delete the unnecessary cutting path and add Leadline,

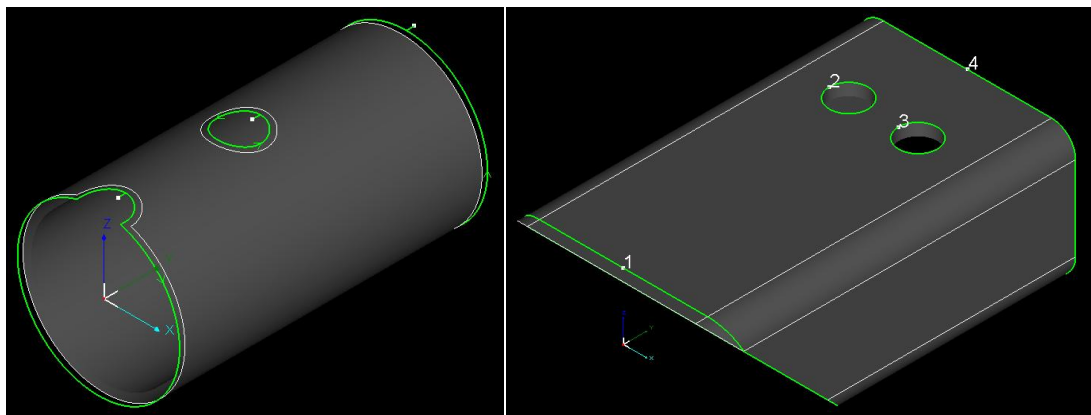


or modify the 3D vector at corner position by which it is capable to change the direction of laser head towards corner and avoid collision. More details and applications in subsequent chapter.




1.2.3 Techniques

By setting technique on parts including Leadline, Kerf Compensate and Intersect Hole etc., it is possible to improve cutting quality and facilitate post process in part joint and welding. The round part below applied Compensation and Leadline technique, while the square one Kerf Compensate only.




1.2.4 Nest

Input parts quantity and click  to open Auto Nest dialog box, when the nesting settings are finished, then click “OK” to start nesting immediately. The nesting results will be listed in bottom window, labeled with nested parts number, nesting quantity and utilization rate etc.




<input type="checkbox"/> Nesting	Tube ...	Tube l...	Remnant	Par...	Utilization	Lock
<input checked="" type="checkbox"/> H-beam Width100 X Height200	x0	6000	320	11	94.7%	<input checked="" type="checkbox"/>
<input type="checkbox"/> H-beam Width100 X Height200	x1	6000	1820	8	69.7%	<input type="checkbox"/>
<input type="checkbox"/> Square tube Width30 X R2_Ne	x1	6000	3330.02	20	44.5%	<input type="checkbox"/>
<input type="checkbox"/> Obround tube Width80 X R.20_	x1	6000	5220	15	13.0%	<input type="checkbox"/>
<input type="checkbox"/> C Shape Width100 X Height50	x1	6000	2820	30	53.0%	<input type="checkbox"/>

1.2.5 Sort



Click  in top menu to sort the nested parts.

1.2.6 Export

Click , select a file folder to save all nesting results in *.zx format respectively. Or you can click Export Production Task Package to save all nesting results in a single *.zx file, easy to manage and count.

If the TubePro on machine failed to recognize *.zx file, it might be that TubePro's version doesn't match with TubesT's, please update TubePro to corresponding version to solve the problem.

(For detailed operation instructions, please check relative technical documents on our official website.)

 TubesT	 TubePro
1.13	7.0.7
1.15	7.0.9
1.17	7.0.11
1.21	7.0.13.106~7.0.13.126
1.23	7.0.13.128~7.0.13.131
1.25、1.27	7.0.13.132~7.0.13.134
7.1.29	7.0.13.135~7.0.15.141
7.1.31	7.0.15.142~7.15.144.2
7.1.35	7.15.144.3~7.15.145.5
7.1.37	7.15.145.6~7.17.150.2
7.1.39	7.17.150.4~7.17.151.8
7.1.41	7.17.151.9~



The above is the corresponding versions table between TubesT and TubePro.

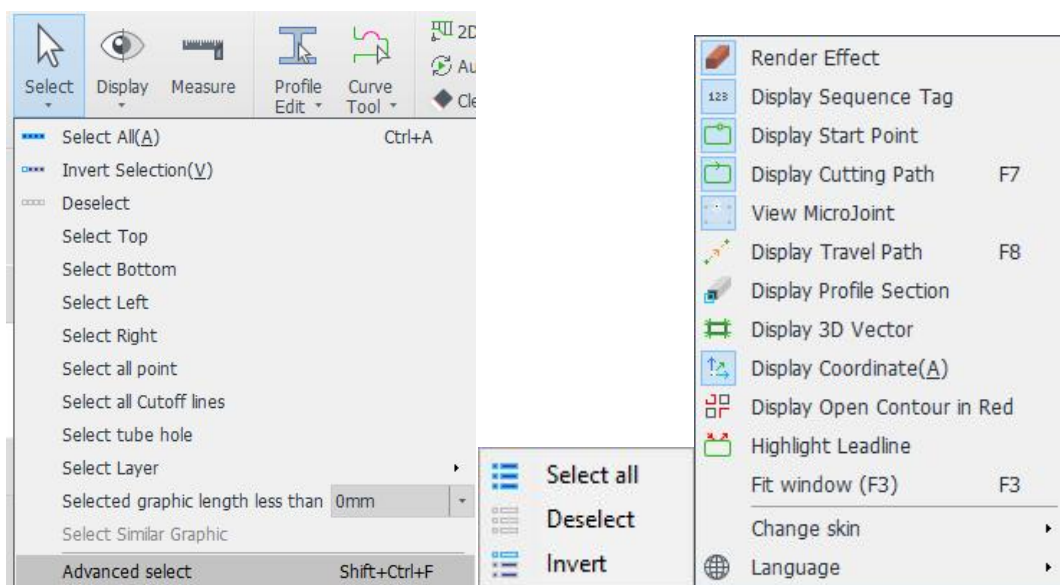


II. Functions


2.1 Application Interface

2.1.1 Select

On the top of Menu Bar, click  to open drop-down menu with many options listed like Select All, Invert Selection for users to select. In part list, click  to select among listed parts.



2.1.2 Display

Click  in menu bar to open the drop-down list to display selected feature tags applied on part design. The options frame-selected in blue indicates those effects have been turned on.

The render effect has relative high requirements of CPU and Memory, it's suggested to turn off the render effect. if software is jammed by large file or low-level firmware configuration, The geometry vector indicates the direction of laser head towards tube surface, this feature plays an important role in 'Intersect Hole' technique and "3D Vector Edit" function. The theme of TubesT user interface can be set to dark or light color by switching skin selection.

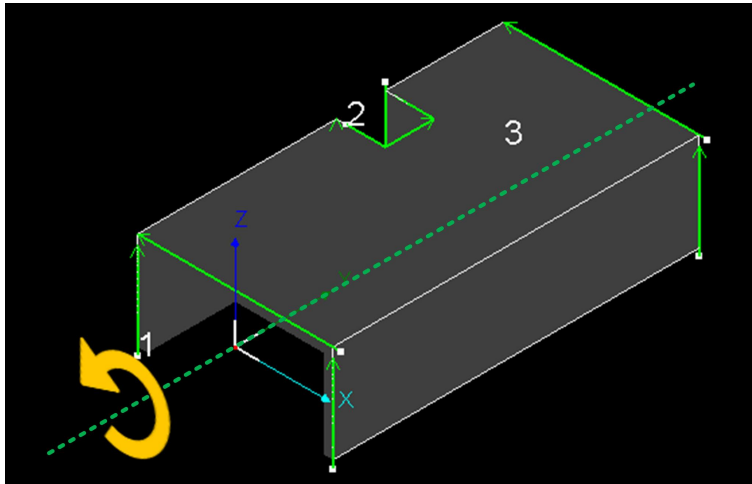


Other options are quiet intuitive and straight, there will be no more redundant descriptions here.

2.1.3 Adjust View

It is available to adjust view by below operations for any parts or nestings.

2.1.3.1. Rotate around Y axis



Shift + Scroll Mouse Wheel: slowly rotate part entity;

Shift + Press Mouse Wheel & Drag: quickly rotate part entity;

Ctrl + Scroll Mouse Wheel: quickly rotate part entity.

2.1.3.2. Translate View

Ctrl + Press & Drag Mouse Wheel.

2.1.3.3. Toggle View

Double click at position near the part/tube entity will prompt the floating box



, select any one of them to switch the view.

2.1.3.4. Free View

Press mouse wheel and drag it to view part entity freely.




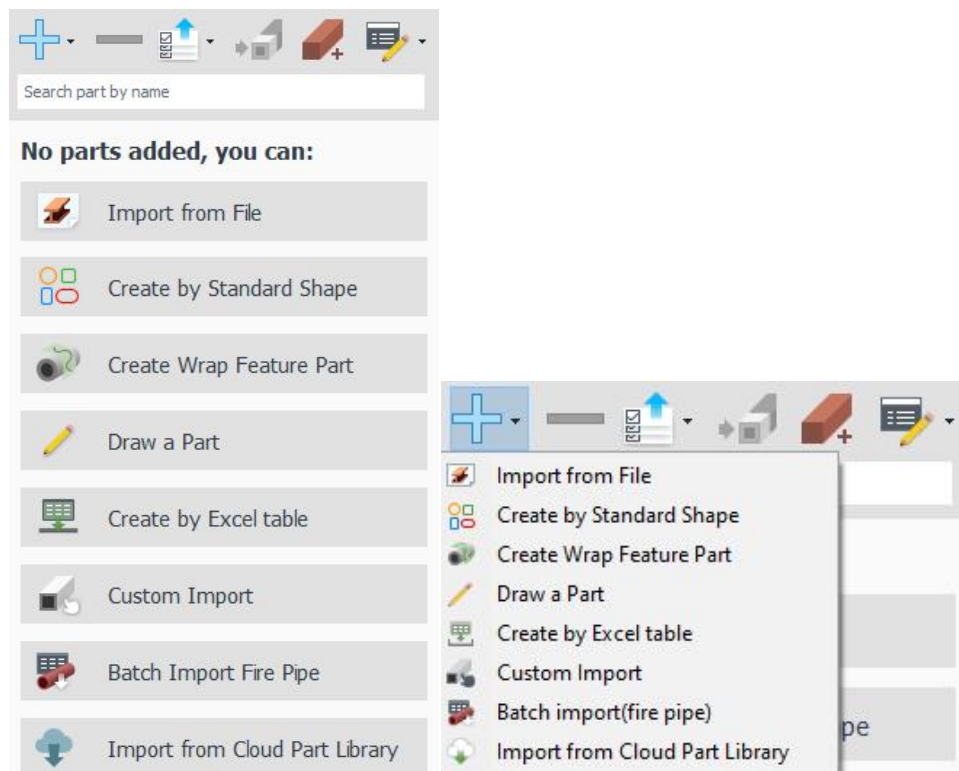
2.1.3.5 Restore Default View

You're allowed to adjust view fit the edit window automatically by pressing F3; Pressing F4 will return to default view and direction.

2.2 Add Parts

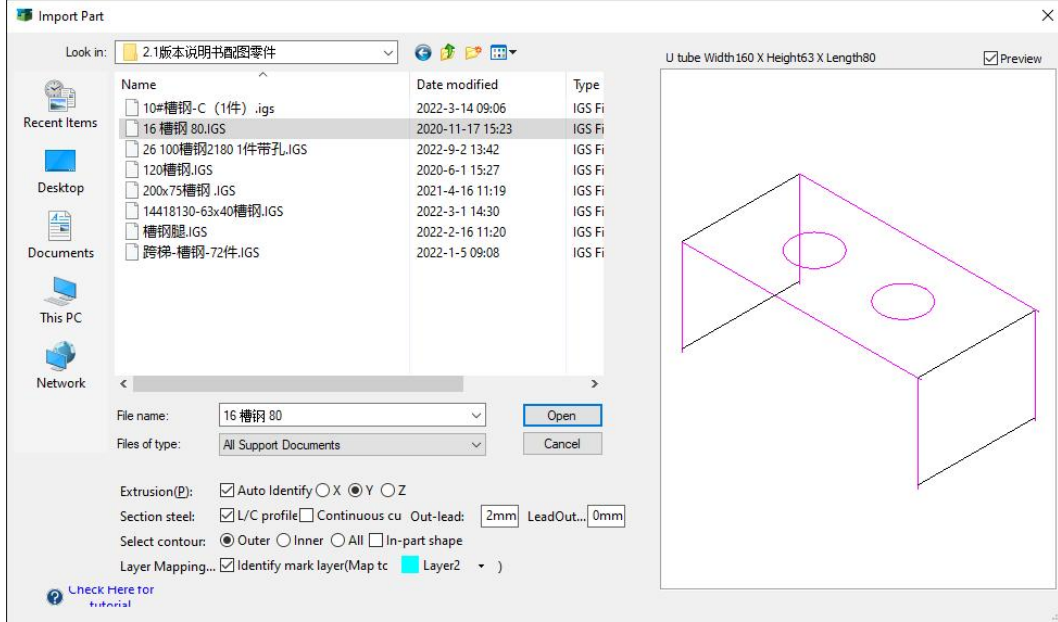
TubesT allows user to add parts in multiple methods for different scenarios.

When you open TubesT or create a new workspace, there are five options in parts list: "Import from File", "Create by Standard Shape", "Create Wrap Feature Part", 'Draw a Part' and 'Create by Excel Table'. If not, click  to select an option to add new parts.



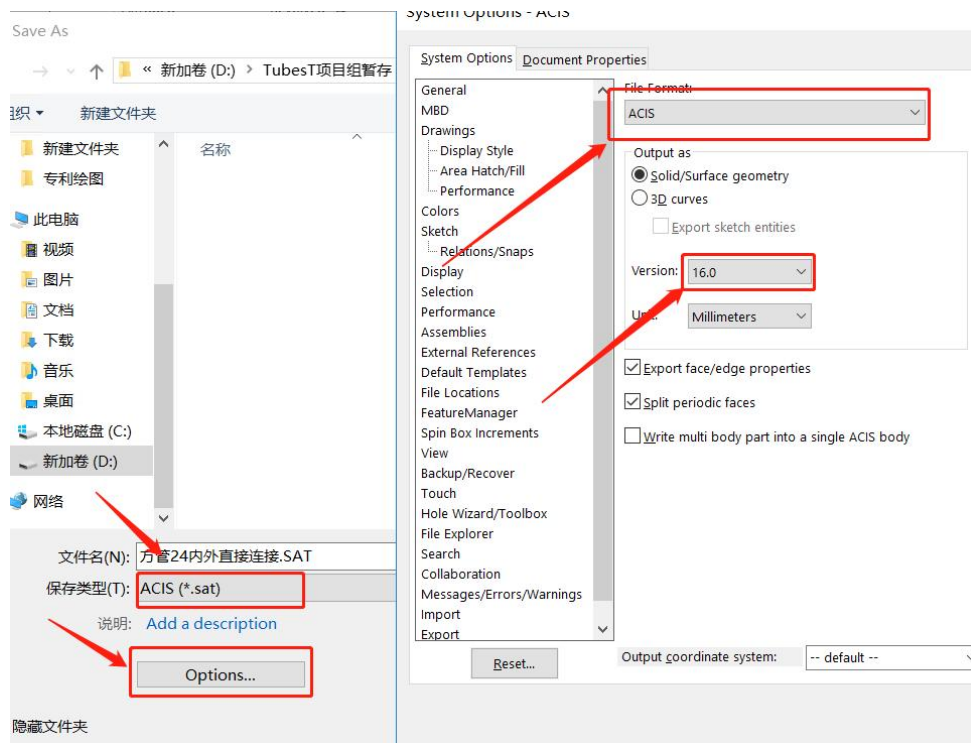
2.2.1 Import from File

Click Import from File to open file browser.



This is the most common way to add parts for TubesT, four formats in total supported:

1. IGS: IGS file created by UG, SolidWorks (extrusion direction must be along X/Y/Z);
2. SAT: SAT file created by SolidWorks 2016 and above (no limit of extrusion direction for SAT file, but ACIS 16.0 in export setting is necessary);



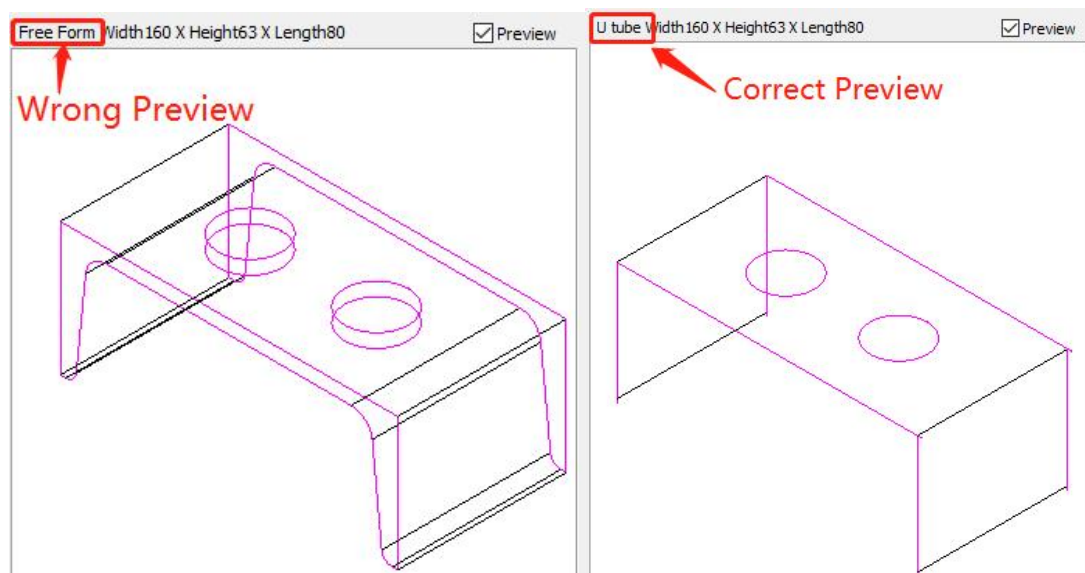


3. JHB: JHB file is sketch file of part drew and created by TubesT.

4. JHBS: JHBS file is sketch file of part design drew and created by TubesT.

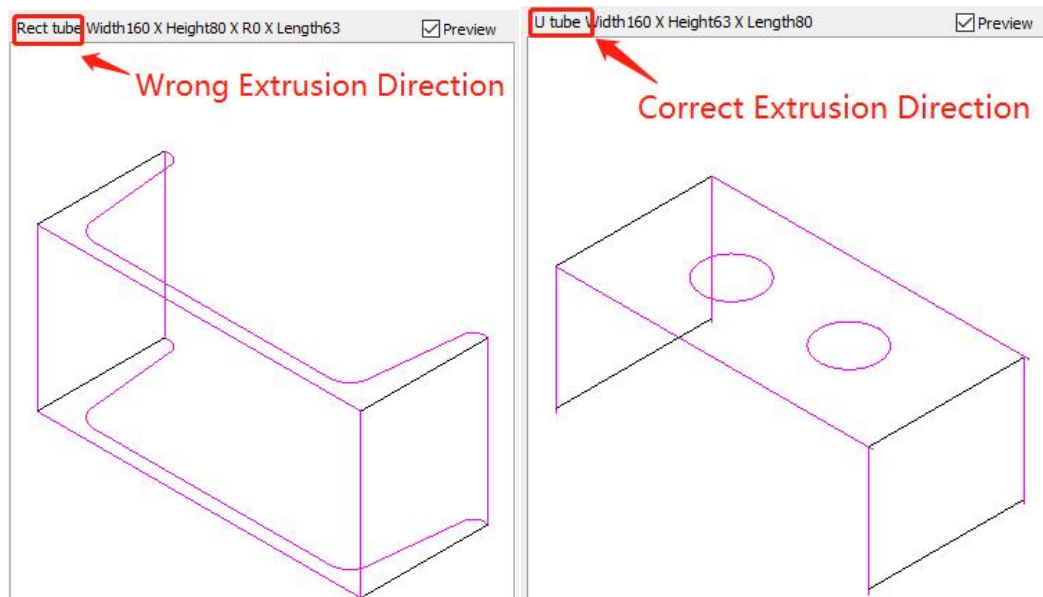
There are also some important parameters in this window.

1. Part Shape: The part shape and dimension recognized from file will be shown on top of “Preview Window”. If recognized part shape is not consistent with original one, for example the original design is H beam while the recognized result is “Free form”, then it means the file reading failed, which will effect nesting and actual cutting. “Loading” indicates that program is reading file, while 0.00x0.00x0.00 indicates no file is read in.



2. Preview: You can check current drawings found by Preview/ If the preview of part geometry is not same to original one, that means importing options may be set improperly in file browser.

3. Extrusion: Auto is to recognize extrusion direction of part entity automatically; if program read wrong extrusion direction, the part profile might be recognized as other shapes, and you're allowed to manually select extrusion direction to ensure parts identification is correct.



4. Select Contour: generally tubular entity has two contours in profile by nature of wall thickness, but only one needed for cutting. Usually we keep the Outer Contour as the cutting path. More details about this are shown in Compensate and Inner Contour sections.

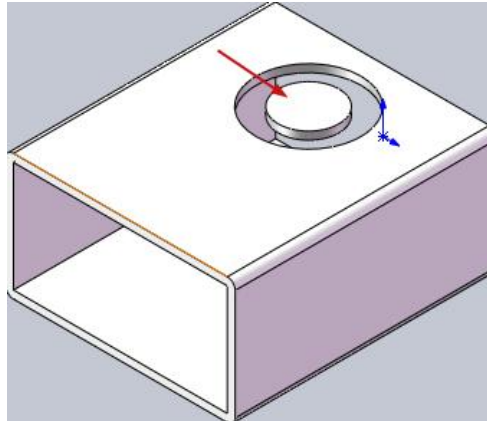
5. Section Steel: for section steel of U shape, L shape, H shape and C shape, it is necessary to select L/C Profile when importing parts to confirm correct section identification and automated special cutting path generation.

Continuous Cut: At present, there are two alternatives for channel steel/angle steel: Cut by Face and Continuous Cut. If Continuous Cut is checked, the channel steel or angle steel parts will be cut by continuous trajectory; If not, it will be cut face by face by default.

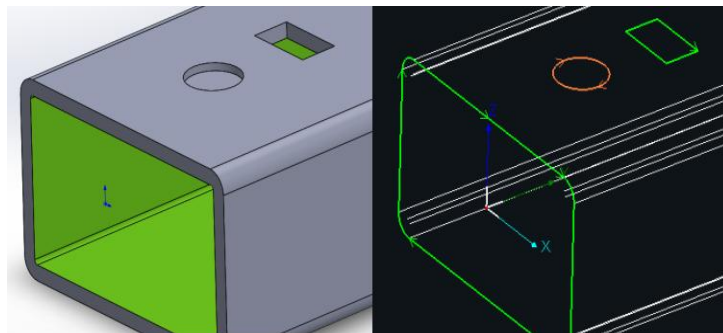
Out-lead is the leadin length outside the parts when profile path for angle steel or channel steel is created automatically, while Lead Out Length is Leadouts length.

6. Bevel: Only available for the tube cutting machines with bevel cutting head which is capable to realize bevel cutting effect by tilting the cutting head.

7. In-part shape: Check this option to ensure TubesT can identify the facet parts in waste area and generate correct cutting path.

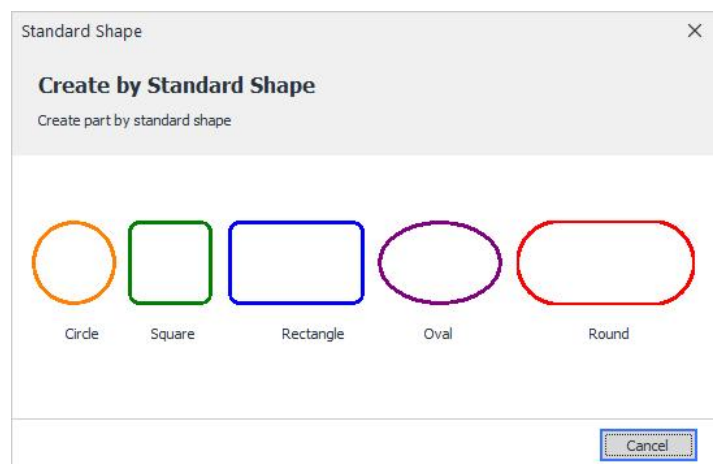


8. Layer Mapping: When importing drawing parts, check Identify Marking Layer and select a mapping layer. After importing, it can automatically distinguish the feature layer without stretching and cutting through from the one with stretching and cutting through. But only *.SAT is available for this function temporarily.



2.2.2 Create by Standard Shape

You can create a part quickly by standard shape in TubesT. The performance for parts design is limited but it is quick and easy for simple part creation.

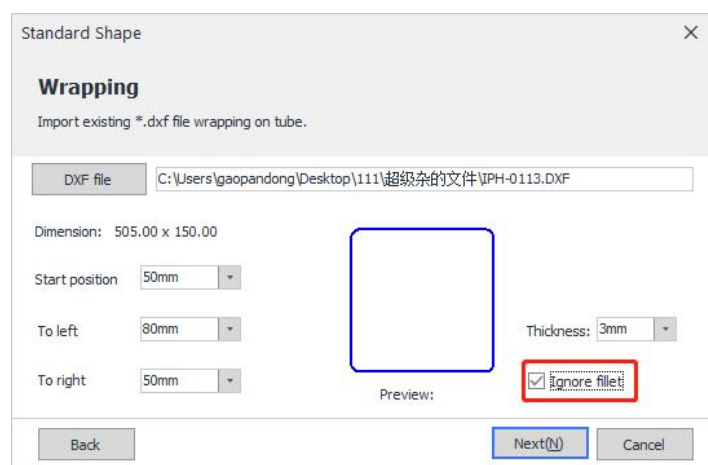




2.2.3 Create Wrap Feature Part

Drawing Wrap-featured parts is to wrap the plane graphics on the surface of the pipe as a cutting path, so as to realize the processing of various patterns or artwork parts. The function can be applied to round, square, rectangular, elliptical and obround pipe.

Besides, it is possible to enable Ignore Fillet so as to calculate the wrapping position by the edge of rectangular tube instead of that of fillets.



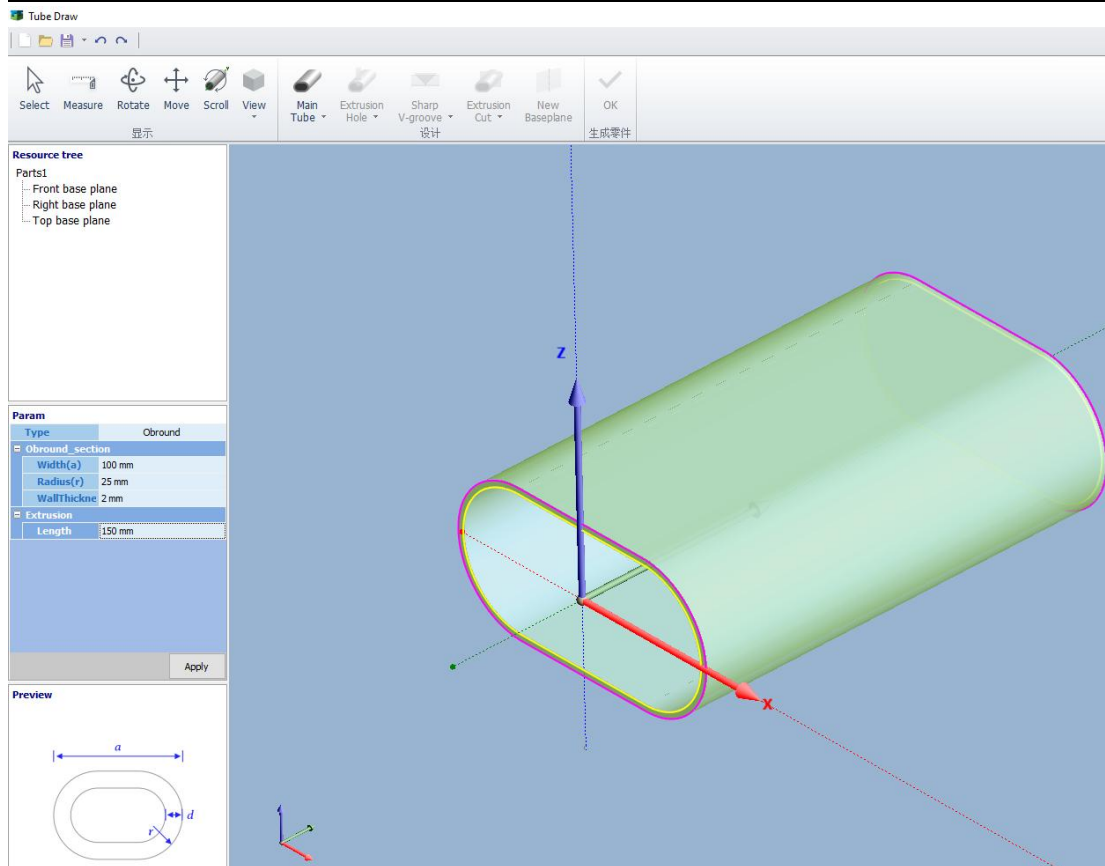
2.2.4 Draw a Part

2.2.4.1 TubeDraw

Available part shapes in TubesT are circle, rectangle, Obround, L shape, U shape, C shape, triangle and ellipse; Available to add hole shapes on tube surface are circle, rectangle and Obround; Available to array holes along X/Y/Z direction; Available to generate cut-off section from any direction.

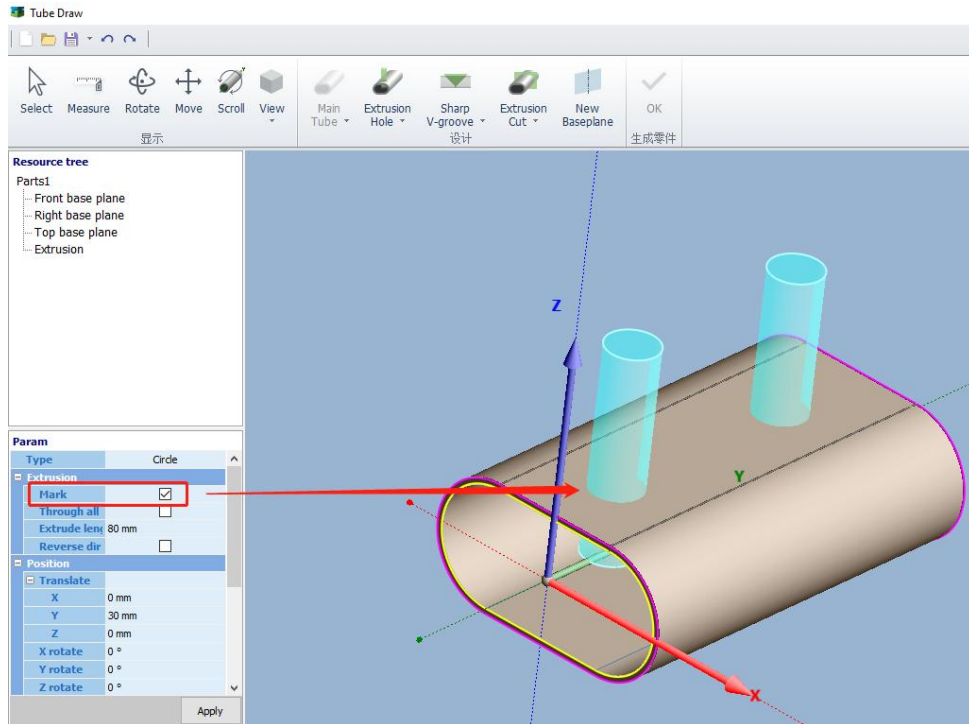
Click New to create a new part in TubeDraw interface.

Click "Main Tube", profile shape, parameters and profile preview will be shown on the left side. And the part entity will change accordingly as the parameters are modified, click Apply to apply the current.



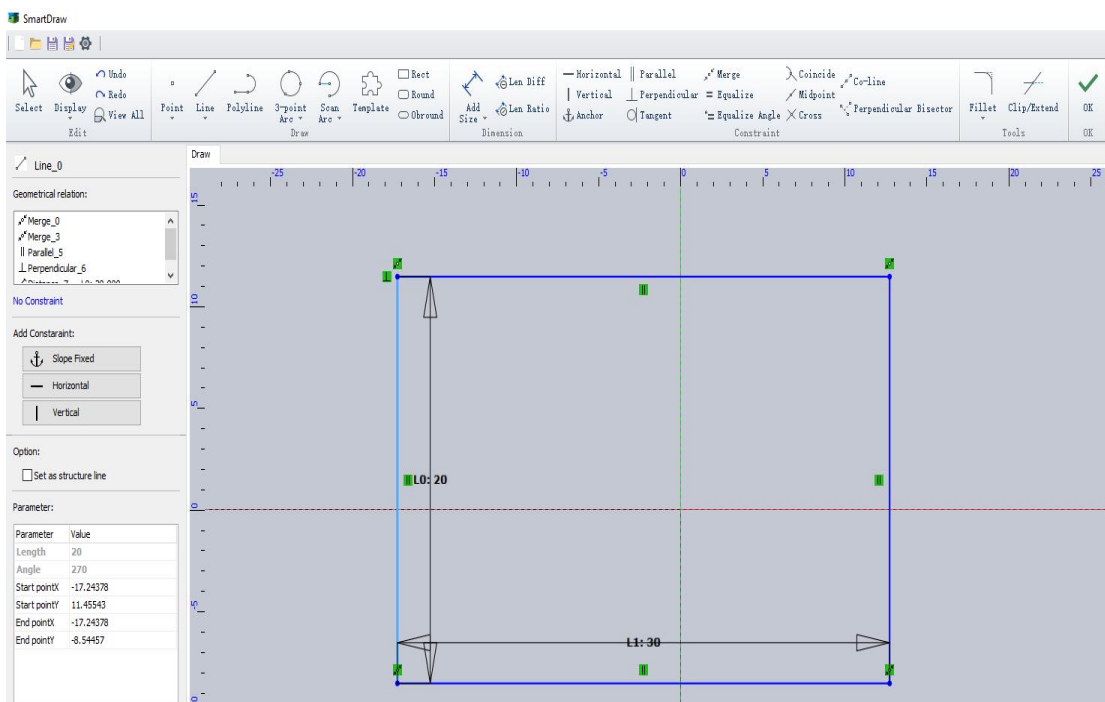
Click “Extrusion Hole” to add designated holes on tube body, in which Available hole shapes are circle, rectangle, Obround, and it is possible to choose Smart Draw to create specific hole as needed, or import a DXF file as a brand new hole by Free Form(Only one closed contour supported in a single DXF), and “X/Y/Z Rotate” can change the extrusion direction of holes. Also, you can enable Array to array the holes in X and Y direction alternatively, also in Z direction if Y Rotate is set as proper angle.

When applying Extrusion Hole, it will turn blue and its trajectory will be featured by Mark if Mark option is enabled so as to draw In-shaped parts and generate corresponding cutting path before processing.



2.2.4.2 Smart Draw

Commonly used pipes can be made in Tube Draw, while special ones can also be customized in Smart Draw under Main Tube, which is available for special Extrusion Holes as well, capable to generate the section type as needed.

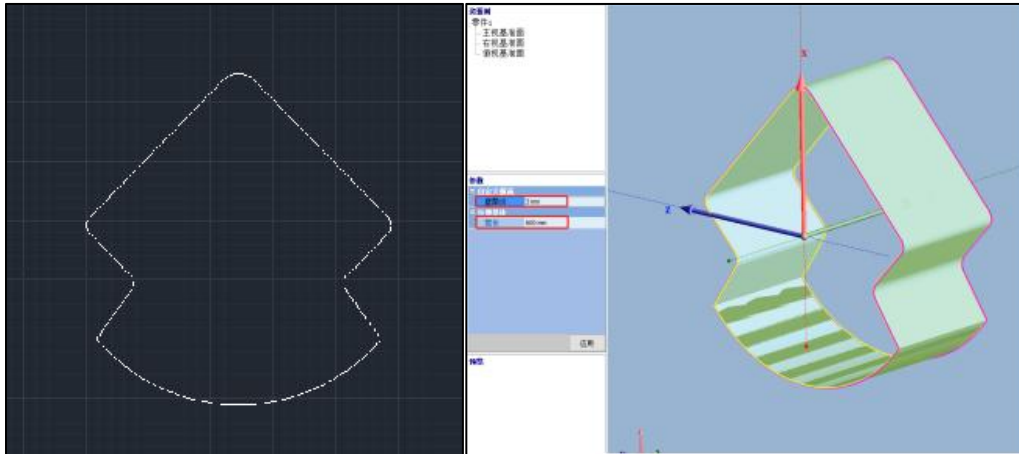


For more details please refer to related articles on our official website.



2.2.4.3 Free Form (Extrusion Hole)

Click Free Form in the drop-down menu of Main Tube, then select a *.dxf file with single closed figure whose preview can be seen in Preview area during the importing. Then click Open and set the wall thickness and parts length, finally click Apply to generate the very part.



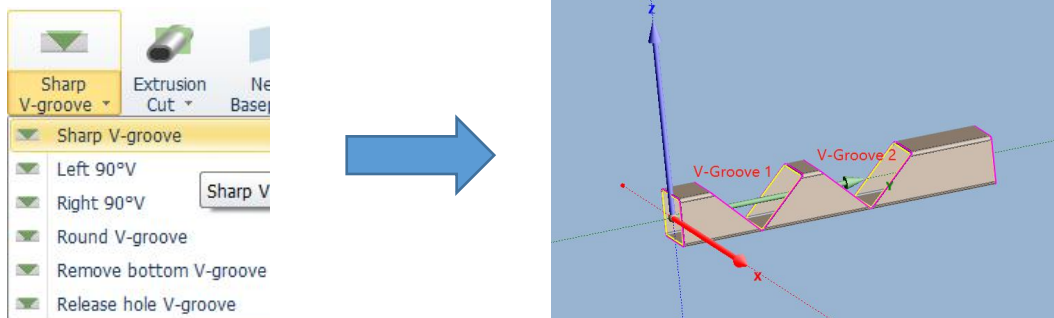
2.2.4.4 Draw a Part-V Groove

At present, TubesT supports Sharp V-groove, Left 90° V Groove, Right 90° V Groove, Round V Groove, Remove Bottom V-groove, Release Hole V-groove etc. The application scenarios and functions of each type will be introduced respectively below.

For more details please refer to related article “TubesT-Draw a Part-V Groove”.

1) Sharp V-groove

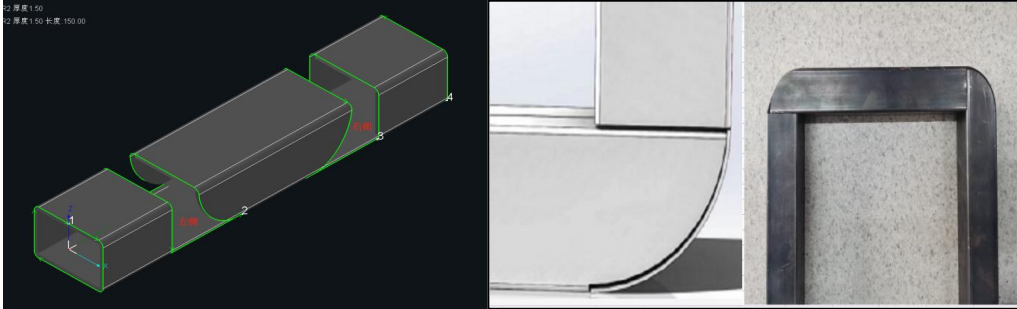
The option Sharp V-Groove in drop-down menu of V- Groove will be validated for all part sections to generate corresponding V groove for bending and formation.





2) Left (right) 90 ° V Groove

After bending, part's outer diameter is rounded and its inner diameter is right-angled, which can only be set on rectangular pipes with a bending angle of 90°.



3) Round V Groove

In Draw a Part Interface, select Round V Groove to quickly and efficiently generate the corresponding cutting path for efficient bending.



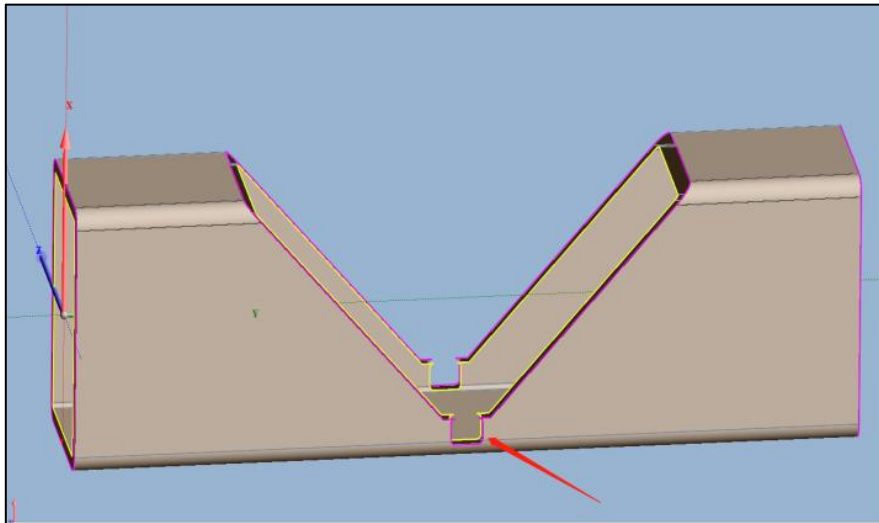
4) Remove Bottom V groove

Remove Bottom V Groove refers to a process in which a slender strip will be cut at the bottom of a V-groove to relieve bending stress and thus reduce stress concentration (This kind of V-Groove should be combined with actual application scenario to set relevant resection parameters. If there is no practical application experience, it is recommended to use sharp V-groove).



5) Release Hole V Groove

The Release Hole at the bottom of V-groove is a process to avoid stress concentration caused by interference between the bottom edge and inner wall of Right V-groove, and the related dimensions can be set according to the angle and wall thickness of V-Groove.

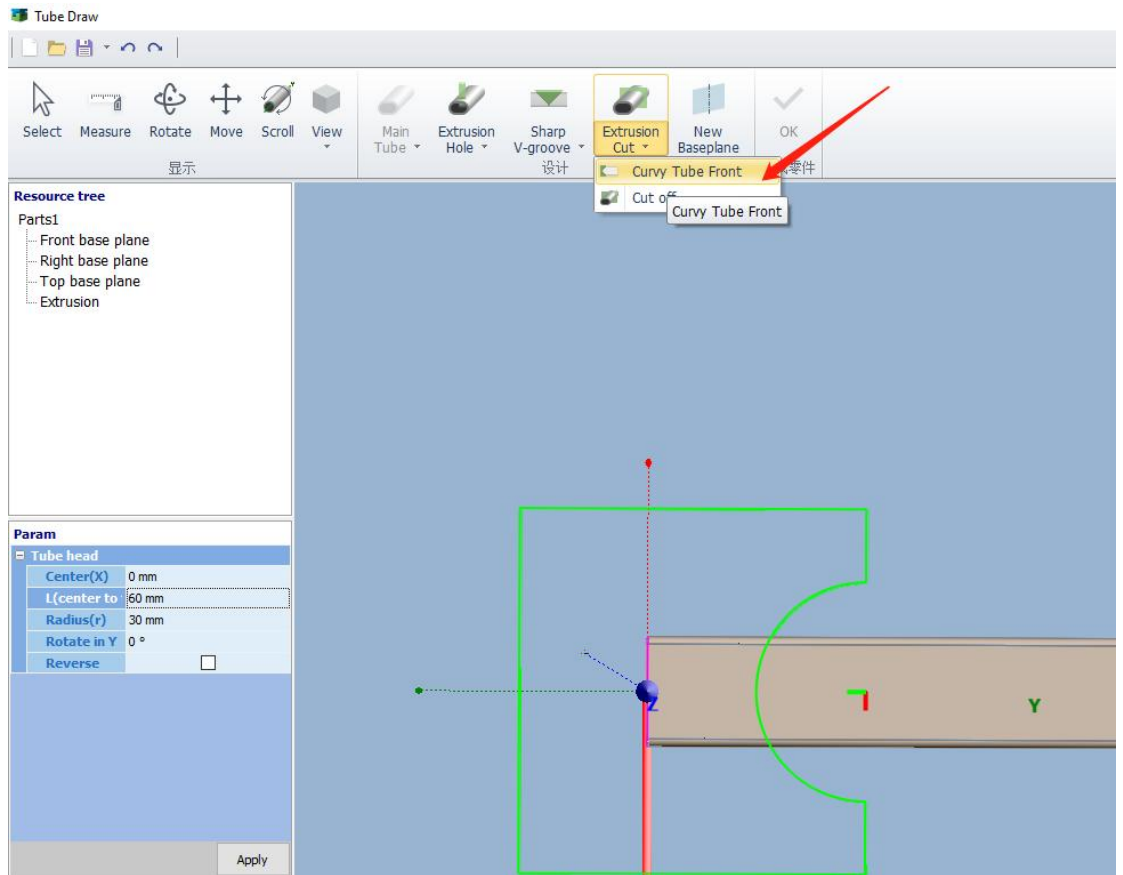
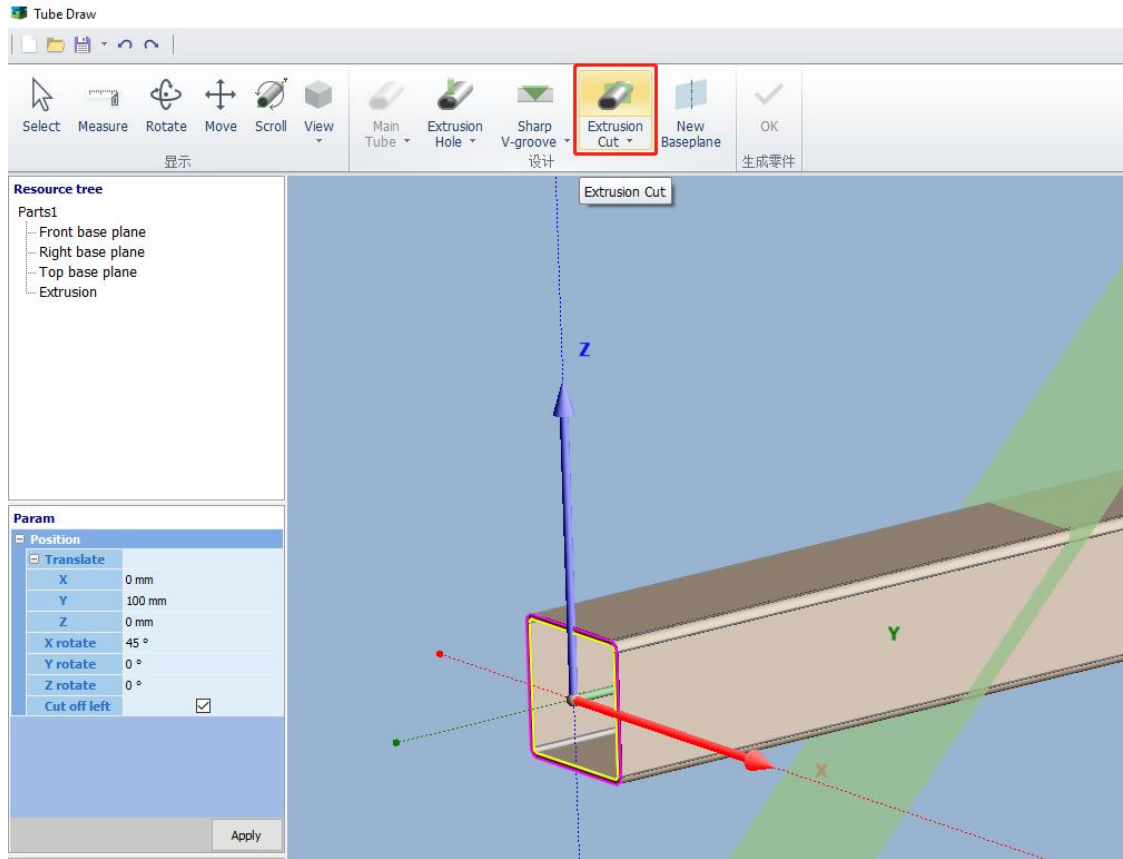


2.2.4.5 Draw a Part-Cut Off/Curvy Tube Front

If you need to draw a Cut Off Face, select Cut Off, if a tube front with a circular arc surface, select Curvy Tube Front in the drop-down menu of Extrusion Cut.

Set the position, rotation direction and other parameters for Extrusion Cut. If “Cut Off Left” is enabled, the body on right side of Cut-Off section will be preserved, if not, the body on left side.

When the Main Tube is applied, the following effects can be achieved with the Cut Off and Curvy Tube Front respectively.

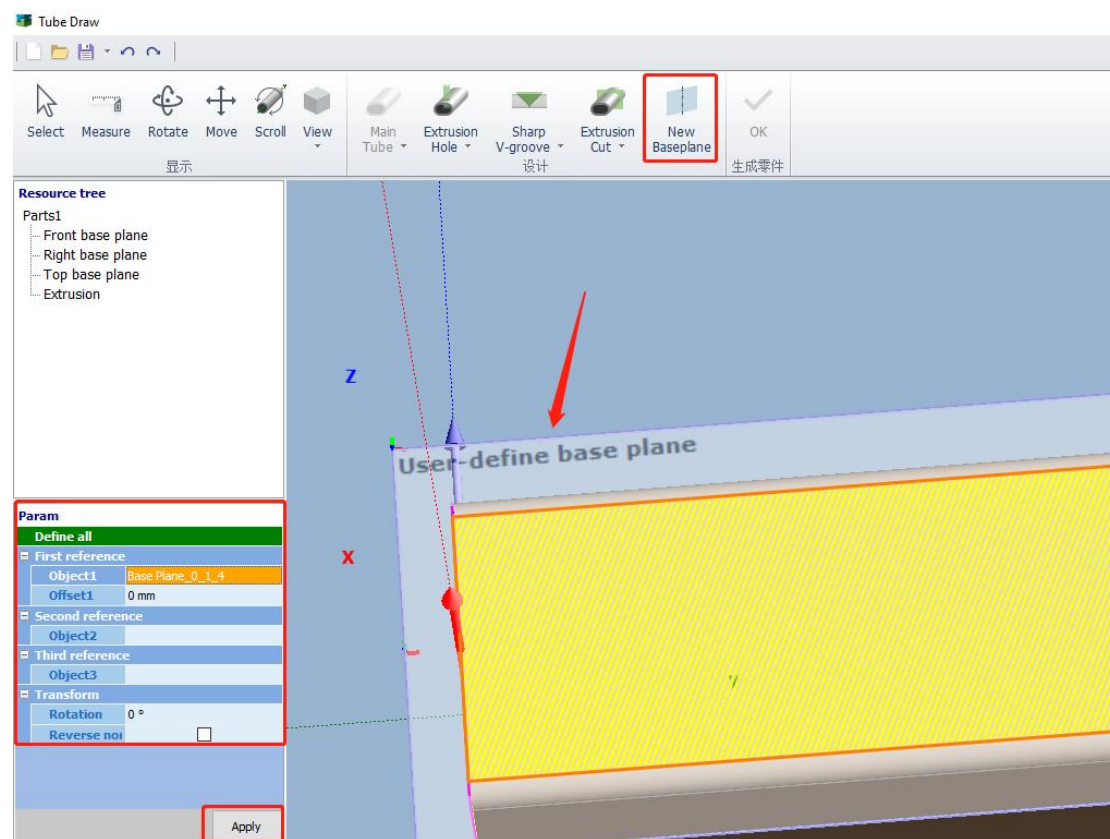




2.2.4.6 Draw a Part-New Baseplane



The sketch of Smart Draw must be on a specific datum plane as Front, Right, Top base planes or a newly created plane.

Click New Base Plane, and select the objects necessary, if it is possible to express a plane with the objects selected, TubesT will generate a preview of this plane. After the desired effect is realized by appropriate objects selection and parameters modification, click Apply to generate current base plane.



2.2.4.7 Display

“Resource Tree” on the upper left shows the design features in each creation step, right click to copy, hide or delete designed features.

Click  to finish part design and it will be added to Parts List automatically with an icon , double click it to re-modify specific part.

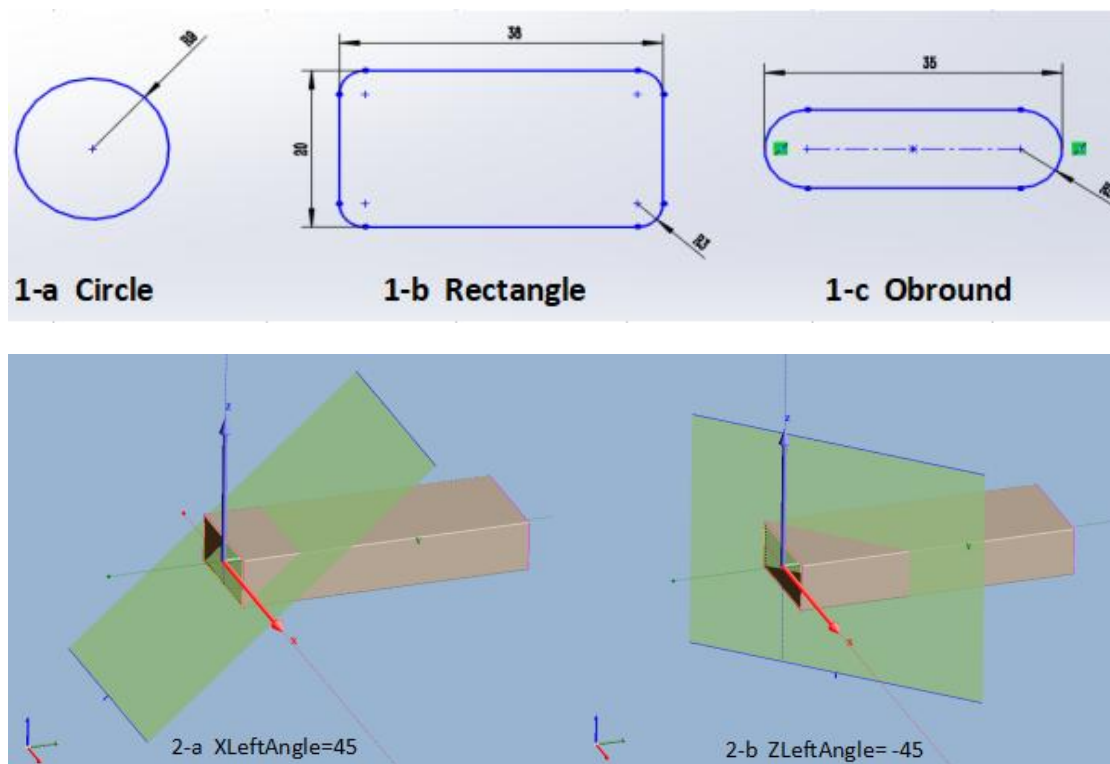


Notes: You're allowed to save part sketch as *.jhb file by Save or Save As, which can be opened through Import from Files in any nesting tasks; Open is to open a file in format in Tube Draw interface, while New is to refresh Main Tube information to start drawing.

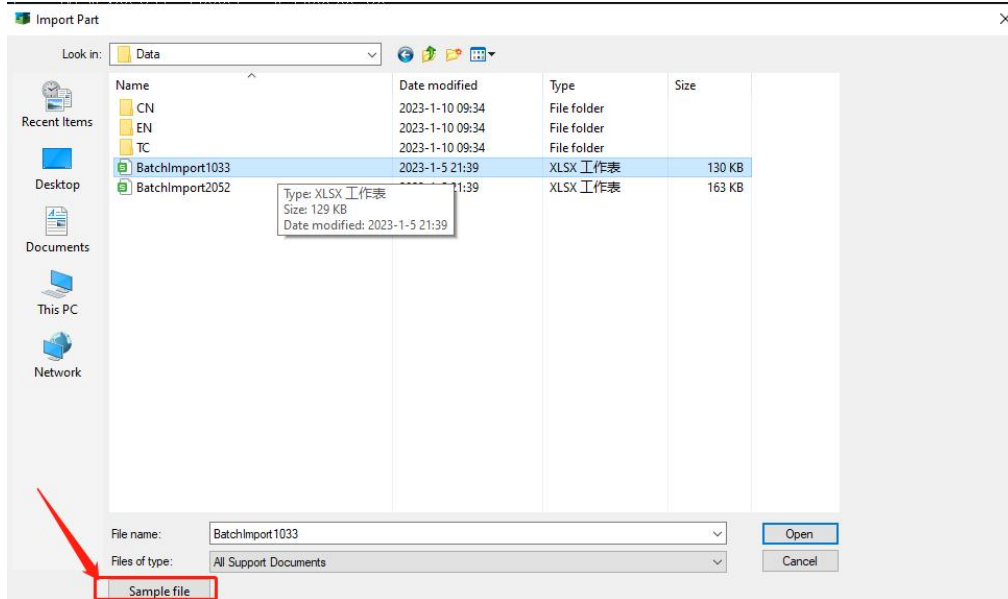
2.2.5 Create by Excel Table

“Create by Excel Table” is to import the parts by an Excel table in which the tube profile and dimension are defined by some parameters, according to which TubesT is capable to create parts automatically.

Supported parts profile includes circle, rectangle and Obround without wrap/hole feature, bevel cut-off section is also possible.



For detailed operation please check Excel template in Import Window as shown below,



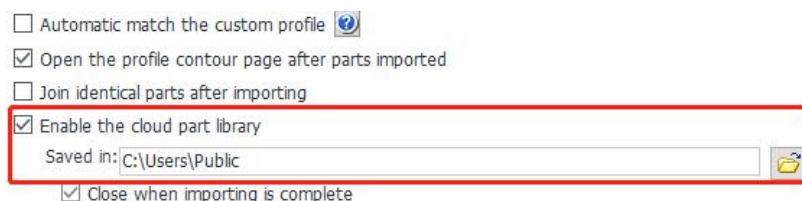
2.2.6 Custom Import

If any parts not created by recommended applications like SolidWorks and UG or created in specific method and you failed to import them by Import from File, then you can try the option Custom Import instead (For drawing requirements, please refer to “TubesT-Draw a Part” on our official website).

After adding a part by the option Custom Import, you can get correct cutting path by setting extrusion direction, cut-off section line, editing the tool path and processing the cutting path under the same layer.

2.2.7 Import from Cloud Part Library

It is only available when Enable Cloud Part Library is checked in User Habit Settings, and it will appear as a stand-alone option on the left side of the main interface. Besides you must ensure TubesT is running in stable network environment which means you need to log in with Wechat or Email Account, even though a dongle has been plugged in.





Synchronize the parts drawn or imported to Cloud Parts Library. When synchronizing to the Cloud Parts Library, if no part is checked, all the parts in the parts list will be uploaded simultaneously by default; if any checked, only the parts checked will be uploaded.

Parts uploaded to the cloud parts library can be imported by selecting specific parts directly in the parts library, or by Excel table. When importing by ticking method, click Import from Cloud Parts Library, check the corresponding parts to be imported, then click Add to import.



<input type="checkbox"/>	Thumbnail	Parts ID	Profile type	Profile size(mm*mm)	Length(mm)	Parts qty
<input checked="" type="checkbox"/>		Obround tube Width100 X R25	Obround tube	50.000 * 100.000	150	1
<input checked="" type="checkbox"/>		Square tube Width30 X R2	Square tube	30.000 * 30.000	200	1
<input type="checkbox"/>		L tube(L) Width65 X Height60	L tube	60.000 * 65.000	200	1
<input type="checkbox"/>		Round tube R49.5	Round tube	99.000 * 99.000	200	1
<input type="checkbox"/>		Rect tube Width80 X Height50 X R5	Rect tube	50.000 * 80.000	80	1
<input type="checkbox"/>		Square tube Width200 X R2	Square tube	200.000 * 200.000	80	1
<input type="checkbox"/>		1	C Shape	50.000 * 100.000	100	1
<input type="checkbox"/>		Rect tube Width60 X Height40 X R3	Rect tube	40.000 * 60.000	170	1

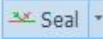
When importing through Excel table, click Excel Import (or download the table template first) and the number in the table should correspond to the number in the cloud parts library, and the quantity can be modified as needed. After opening the Excel table, the parts in the parts library will be automatically matched, click “Yes” to automatically import into the parts list on the left side of the software.


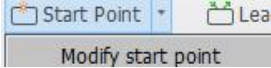


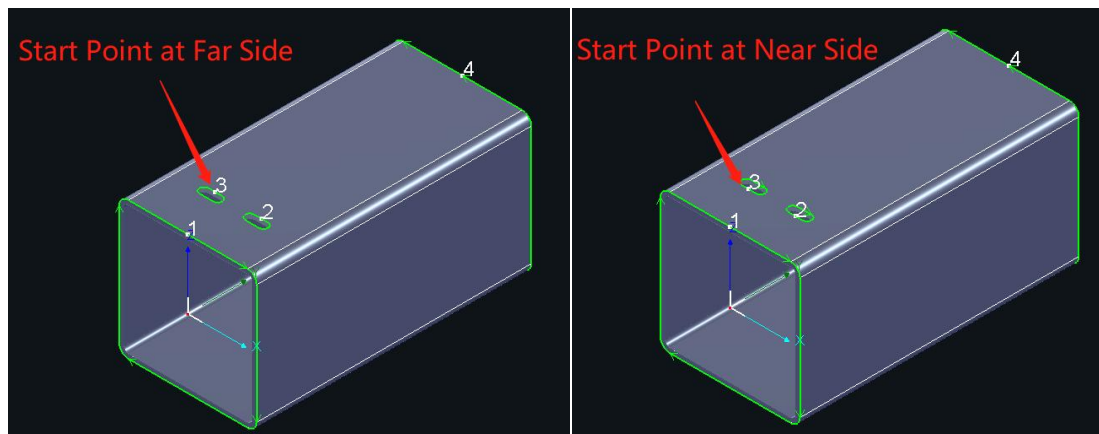
2.3 Technique Settings

2.3.1 Start Point

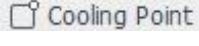
Start point is where laser starts cutting on a tool path segment. Click View, Display, Display Start Point respectively, then TubesT will visualize Start Point by the dot of white color.

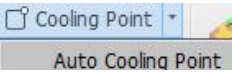
You can change a closed shape as “Seal, Gap or Over” by . A closed shape by default is Seal-featured, which means start and end point converge at a same point; Gap means start and end point doesn't join together with a gap in between, while Over is to add extra path at end point.

You can define any point on the graphic as Start Point by clicking  (press ESC or right click to exit this function), and click  to unify the start point of all shapes located to near-end or far-end to Y=0 position.



2.3.2 Cooling Point

Laser head will pause in cutting where there is cooling point, meanwhile gas will be opened to cool down the heated material. In TubesT, click  to add Cooling Point on selected graphics (press ESC or right click the mouse to exit), click

the drop down arrow of Cooling Point and select  Auto Cooling Point to apply Cooling Points on the parts automatically, and the Cooling Point Delay



can be set in Global Parameters, TubePro.

When Cooling Point is activated, it is possible to delete it by clicking specified one with holding Shift, and you can realize Batch Delete for selected parts by the option Clear Cooling Point in Clear.

2.3.3 Leadline

Leadline is capable to place the Start Point at the waste zone to keep fine enough cutting quality at Start Point.

Select a whole part or a shape and click  Lead to apply Leadline for it.

Leadline Setting

Lead In

Type: Line

Angle: 60°

Length: 3mm

Add cooling poin Joint Path

Lead Position

Lead on CutOff section

Angle with X axis: 30°

Lead on wrapping hole

Far-end Near-end

At long edge At vertex

Apply to

Apply to selected

Apply to closed contour

Cutoff section

Wrapping hole

Lead Out

Type: None

Angle: 90°

Length: 3mm

OK(O) Cancel

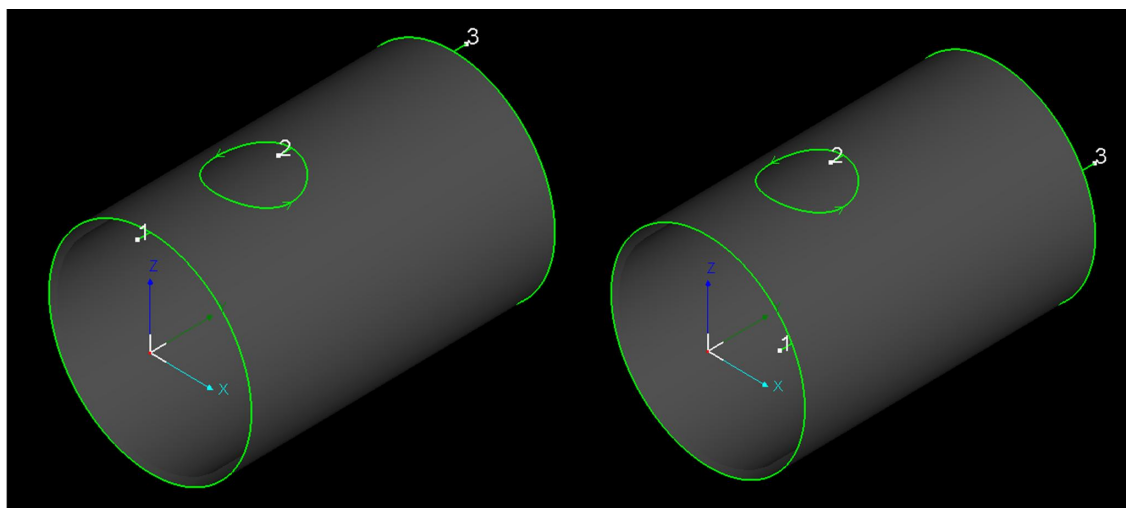
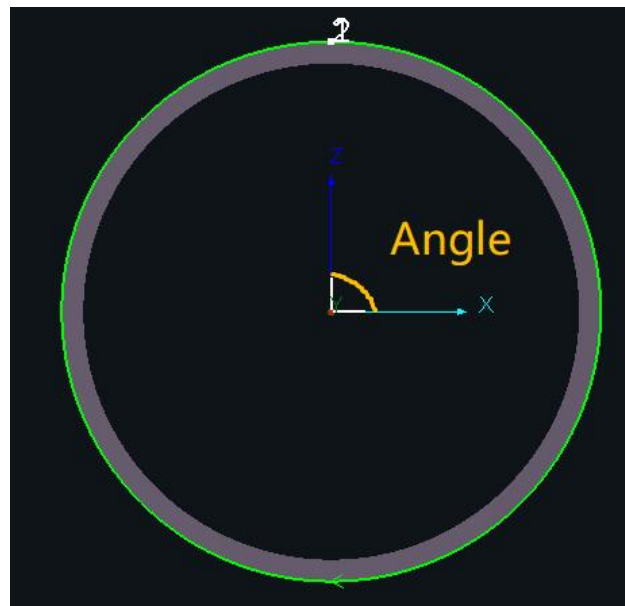
There are two types of Leadline: “line” and “line + arc”, and Add Cooling Point is to insert a Cooling Point automatically at the cross between Leadline and the graphic.

The leadline on wrapping hole and cut-off section should be set separately.

The Leadline position of the section is determined by the angle with X axis, which refers to the counterclockwise included angle from X axis to the target position



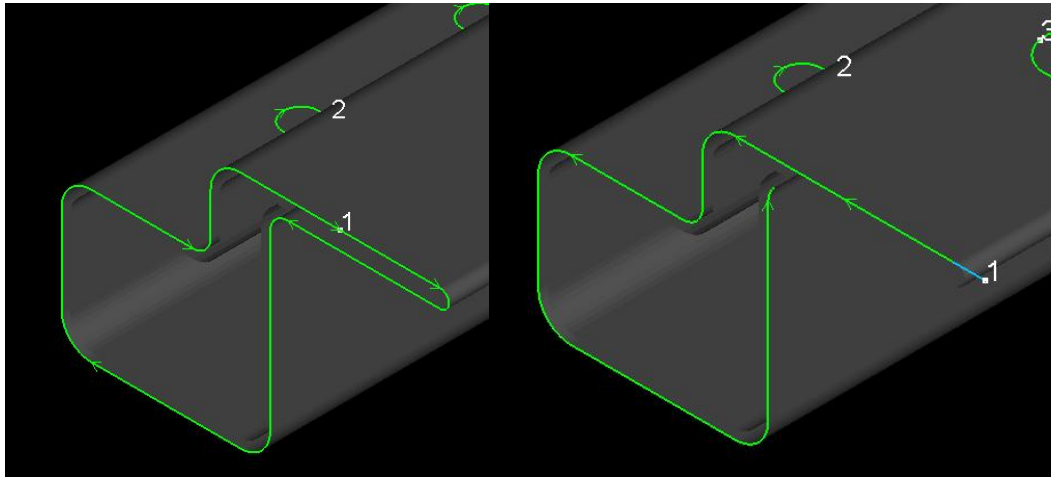
in the main view, and the Leadline position is shown at 90 and 30 respectively.



The Leadline position on wrapping hole can be set to near-end, far-end, long edge or vertex as needed.

Lead In, in the drop down menu of Lead, is to add Leadlines outside the tube, which is capable to start cutting from outside tube without piercing. Generally, Lead In and Lead Out are applied to section steel like angle steel and U-shaped steel. Take the free-formed tube as an example, the cutting path of its section is unclosed after the adjustment with Break Line, Delete and so on, it is proper to apply Leadin for it.


Click Display, Highlight Lead Line will highlight Leadlines in blue color.



Whether it is automatically or manually set Leadline technique, in order to prevent unreasonable lead setting from overlapping with cutting tool path, it is possible to click Check Lead in Lead drop-down menu to check the Leadlines applied. If there are illegal Leadlines existing, they will be marked in red and can be modified and adjusted automatically or manually.

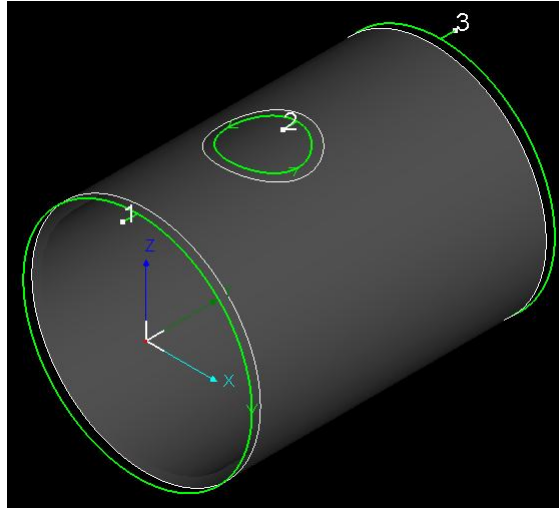
2.3.4 Kerf Compensate

Normal cutting according to original dimension will lose partial materials as the laser beam has width, which will make the part finished smaller than designed and inner holes bigger than designed. The technique to ensure the part precision by offsetting the laser beam is called Compensate in TubesT.

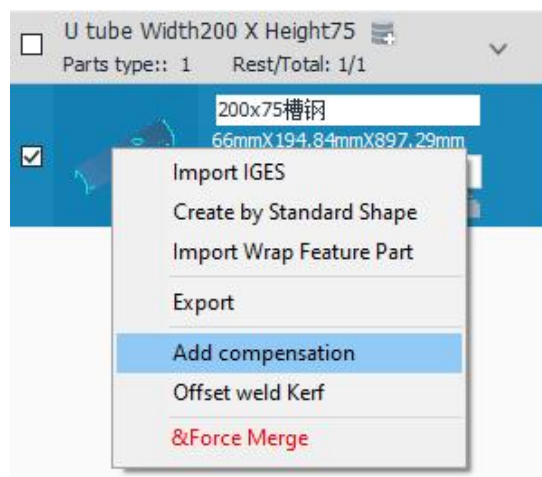
Click  **Compensate** to offset the selected graphics or parts.

You can set the compensation value for the parts of specified range as needed, generally half of the beam width.

Expand and Shrink in “Style” option is the approach to offset cutting path. Usually, “Shrink” is used on wrapping hole, while “Expand” cut-off section, and “Auto” means to shrink wrapping hole and expand cut section. As shown in the figure, the cutting path in white color is the original design, cutting path in green color is compensated one.



Select the graphics and press short keys “Ctrl + D” or click Select, Quick Compensate in turn to add compensation without Compensate dialog box opened.

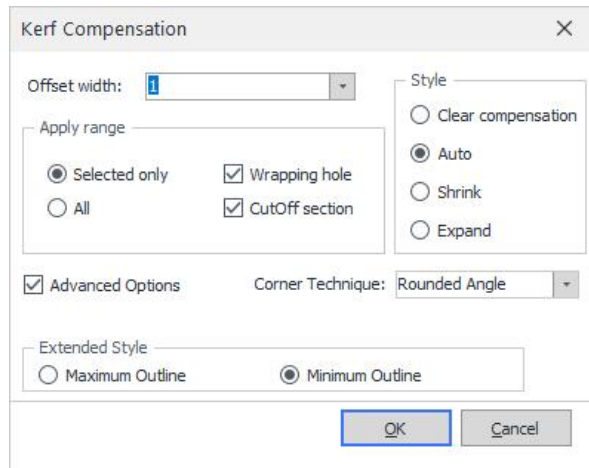


Select parts in Part List, then right click and select Add Compensation to add compensation for designated parts by batches.

It is suggested to compensate parts before nesting as compensation might narrow part-to-part space or even cause cutting path interference.

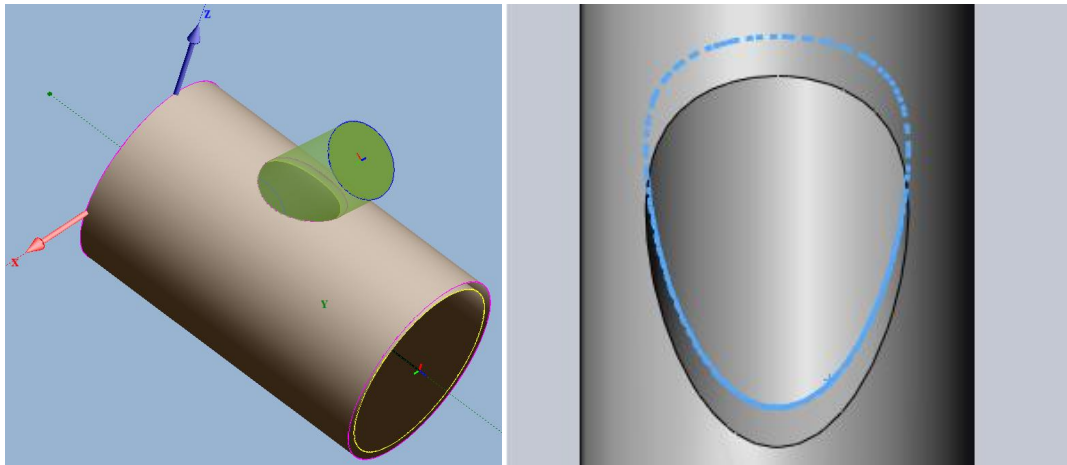
2.3.5 Extended Style

If Extended Style is enabled in Kerf Compensation, you can choose to set the cutting path as Maximum Outline or Minimum Outline.




When there are tilted intersect holes on the tube and you need to insert a branch tube with the same dimension, it will cause interference as the hole is smaller than the tube, because the outer contour is not consistent to inner contour and laser cutting is vertical to the tube surface.

Illustrated in below figure, black solid line is Outer Contour, while the blue dotted line Inner Contour.

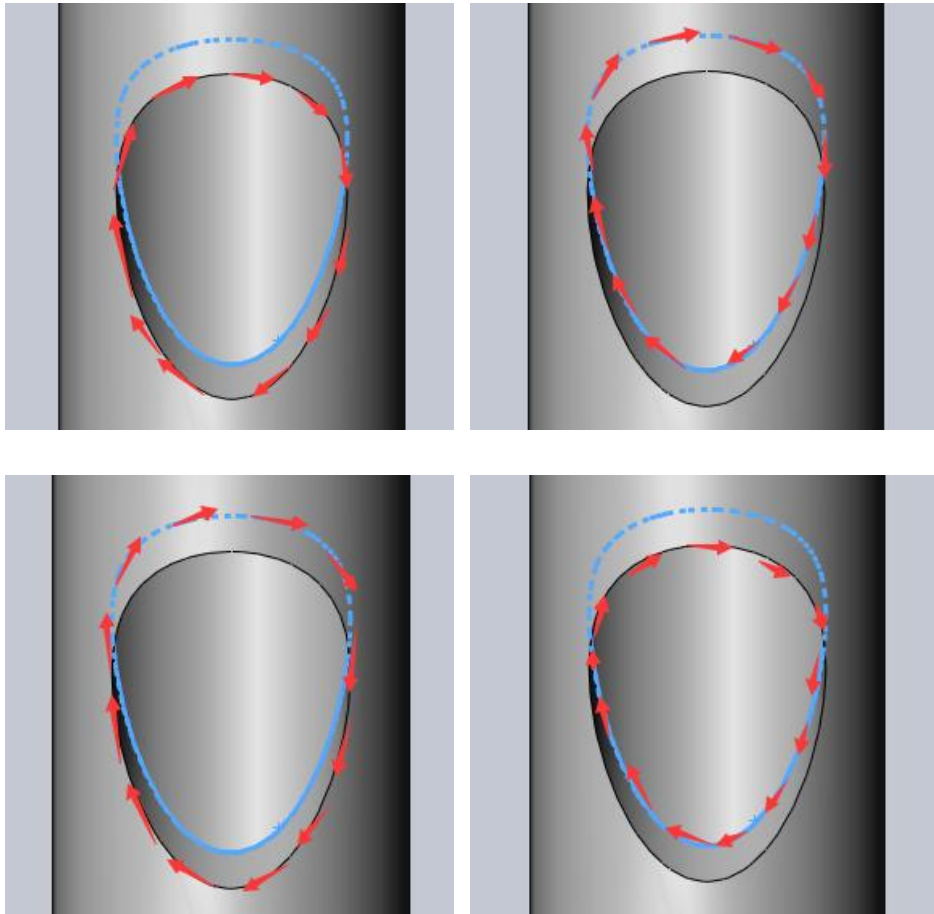


There are four methods to cut tilted intersect hole as followings:

1. Outer contour as the cutting path(default);
2. Inner contour as the cutting path (click  Inner contour to set it as Inner Contour);
3. Maximum Outline, the result of Boolean Summation of the inner contour and outer contour(it is to make sure branch tube can be inserted in the hole, but there will be a little gap on two contours

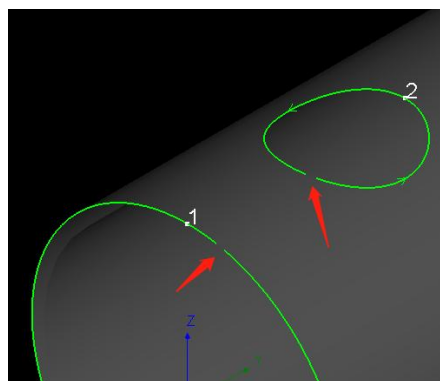


respectively); 4. Minimum Outline, the result of Boolean Difference of the inner contour and outer contour (partial materials will be cut off by second process to make ideal tilted holes which can realize the assembly without gap on any one contours).

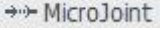


2.3.6 MicroJoint

Micro Joint is to leave a gap on cutting path, capable to keep the part cut off connected with main tube.





Click  MicroJoint to apply MicroJoint on parts (press ESC or right click the mouse to exit).

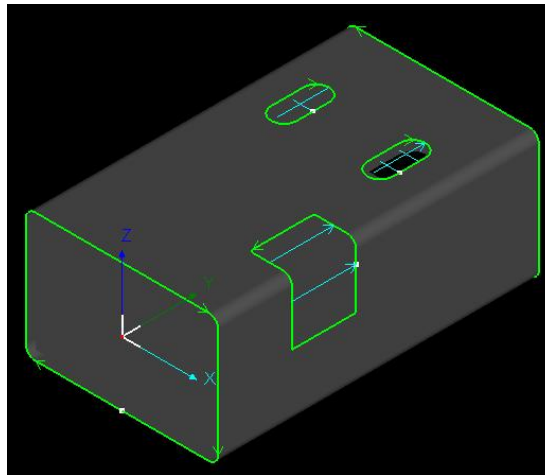
When MicroJoint activated, press Shift and click on Micro Joint tag to delete it, or Click Clear, Clear MicroJoint to delete MicroJoint on all parts. In the Display drop-down menu, you can turn on View MicroJoint Tag to make it easier to identify the location where the MicroJoint technique has been added.

2.3.7 Grid


Grid is to create cutting line in grid pattern on in-part waste that split large waste to small pieces falling down.

Face Grid is to create cutting line to holes in tube face; Corner Grid is to create cutting line to holes on corner; Manual Grid is to manually drag cutting lines to holes. The cutting lines created by Grid are in blue color, which will be cut first before hole's contour.

Click Clear, Clear Grid to delete Grid lines on selected parts or shapes.



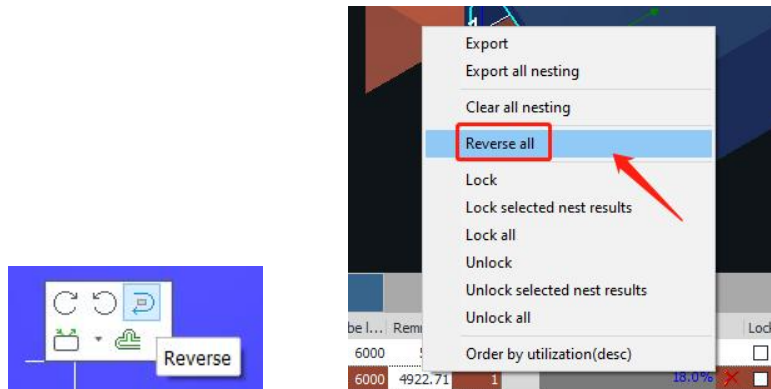
2.3.8 Reverse

Select specified cutting path, click “ Reverse” to reverse the cutting path from clockwise to counterclockwise.

If you want to reverse a single part, click the part and then there will be an icon



aside it, choose Reverse to reverse part direction 180°.

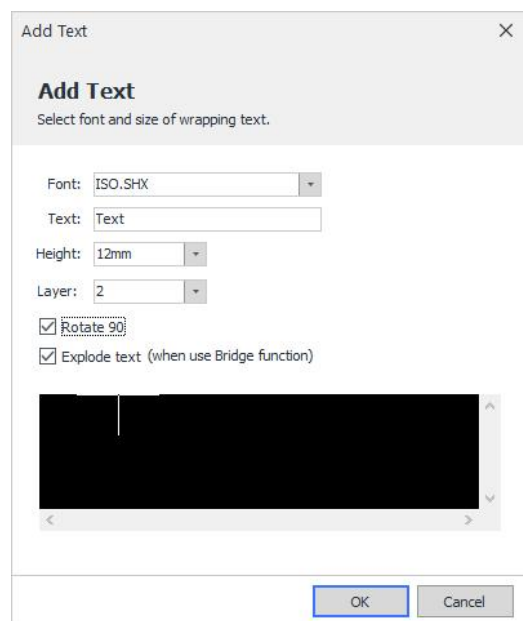


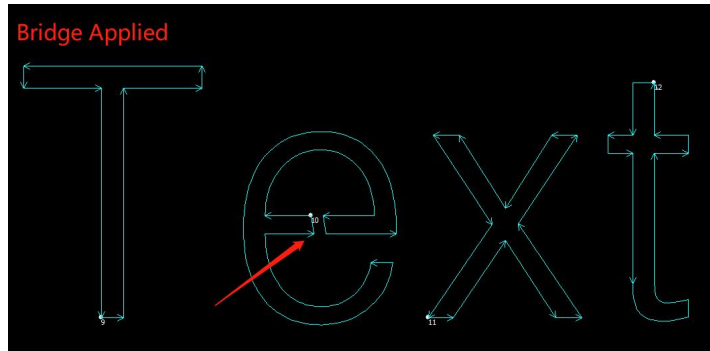
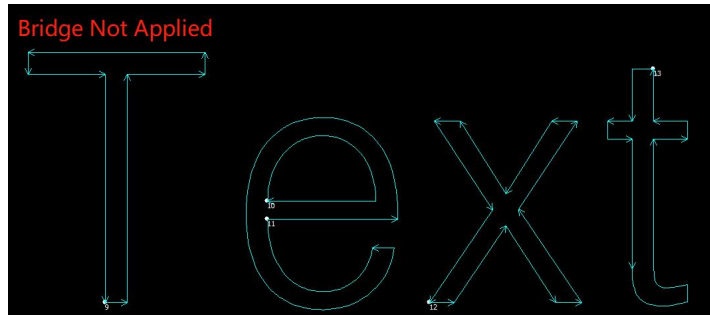
For the nesting results, right-click the one and click Reverse All to reverse the currently selected nesting result.

2.3.9 Bridge

When machining hollow fonts, you can add Bridge technique to avoid dropping the inner holes of the text.

Enter 2D Edit, click Bridge, set the Bridge Width, and then you can add a bridge to the text. Click Save to map the bridge results to 3D (It should be noted that if you generate the text by Add Text in Curve Tool of TubesT, it is necessary to check Explode Text before bridging it).



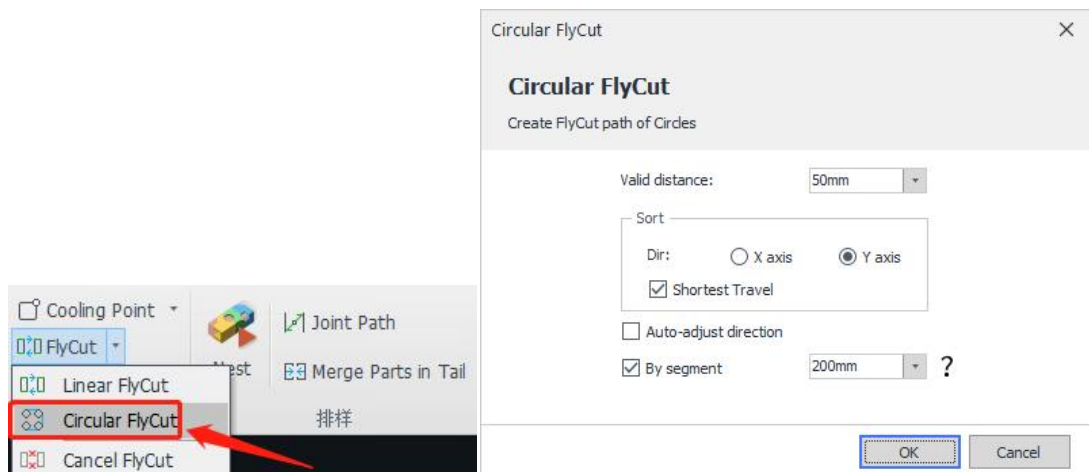


2.3.10 FlyCut

FlyCut is also called Flying Cutting and Scanning Cutting. When the holes in the pipe are regular figures (such as circle, rectangle, polygon) and arranged regularly, FlyCut will greatly improve the cutting speed and save the processing time by connecting the line segments in the same direction for flight cutting.

2.3.10.1 Circular FlyCut

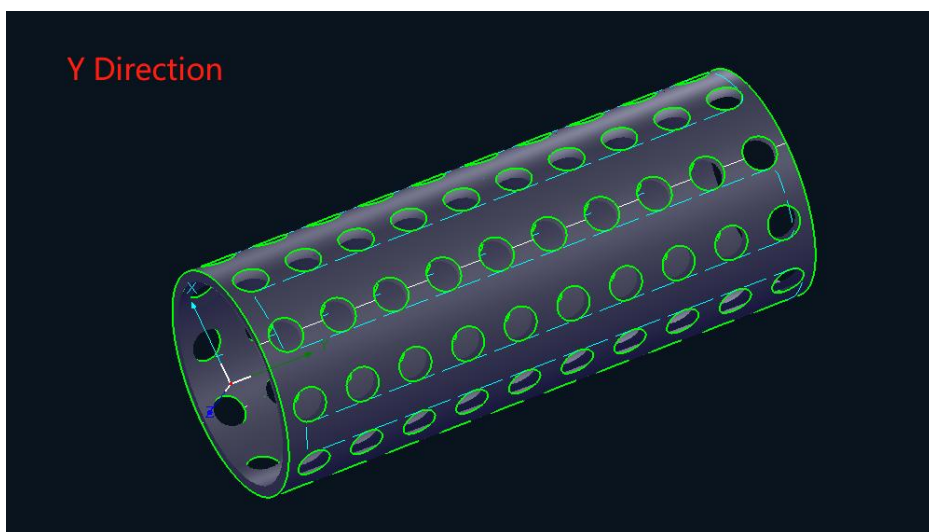
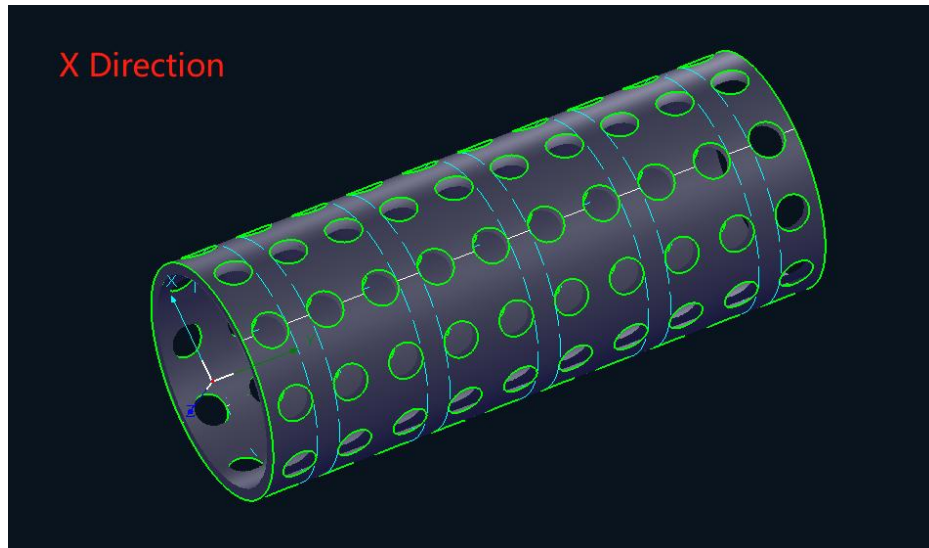
For the regular arrangement of circular holes on the pipe (whether round or rectangular), Circular FlyCut should be used.





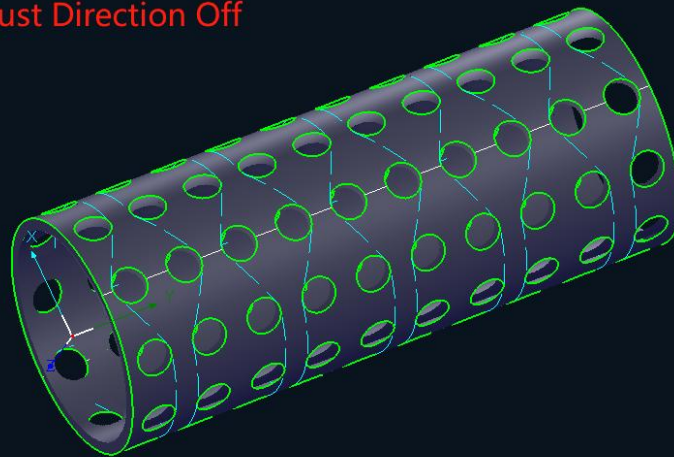
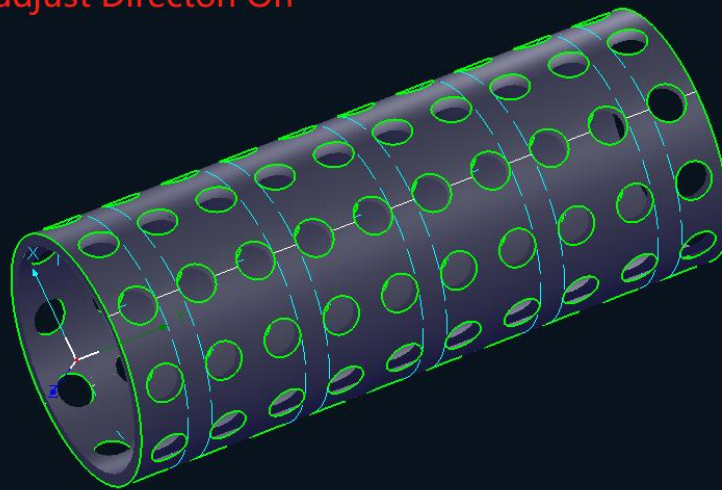
①**Valid Distance:** When the starting point distance between round holes is less than the set value, then FlyCut can be applied. It is recommended that this parameter be set slightly larger than the distance between round holes.

②**Sort Direction:**



③**Auto-adjust Direction:** When the starting point position and cutting direction of the round holes are different, the generated FlyCut lines will be chaotic and not smooth enough.

If there is no special need, it is recommended to enable this option.

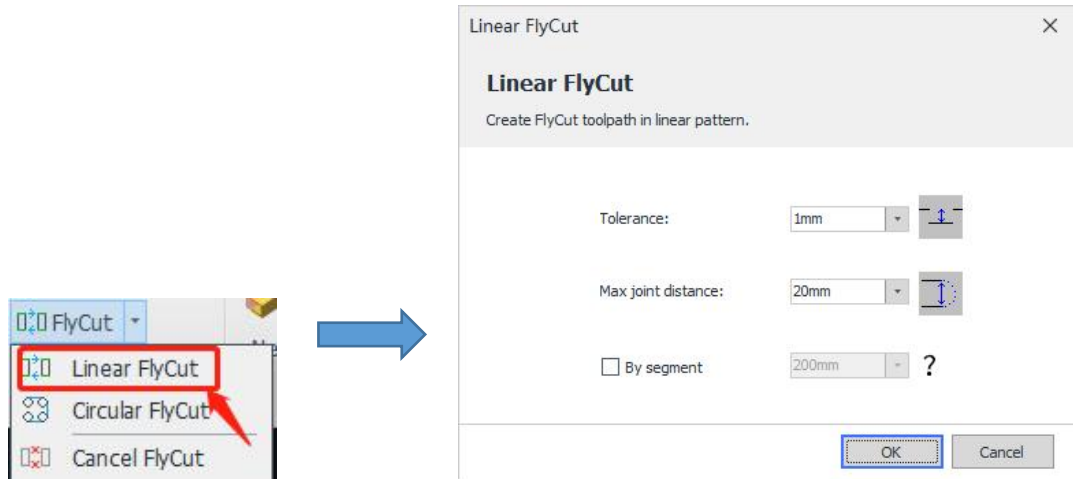
**Auto-adjust Direction Off****Auto-adjust Direction On**

④ **By Segment:** For long parts, it is recommended to use this function to avoid the tube movement forward and backward for a long distance during the cutting.

⑤ **By Face:** For rectangular pipes, it is recommended to use FlyCut By Face. The FlyCut efficiency of cutting the whole face one by one is higher than that of cutting with frequent rotation of B axis.

2.3.10.2 Linear FlyCut

For the regular arrangement of rectangular holes on the tube (whether round or rectangular), Linear FlyCut should be used (Note that rectangular holes with rounded corners cannot be set for FlyCut).



①**Tolerance:** When the FlyCut line connects all segments on a same line, the ones with a linear deviation less than this value will be jointed to this FlyCut line as well. line. The default value is 20mm.



②**Max Joint Distance:** When flycutting from one line to another, the two with a distance between them less than the set value can be smoothly connected (It is not recommended to modify this parameter).

2.3.10.3 Cancel FlyCut

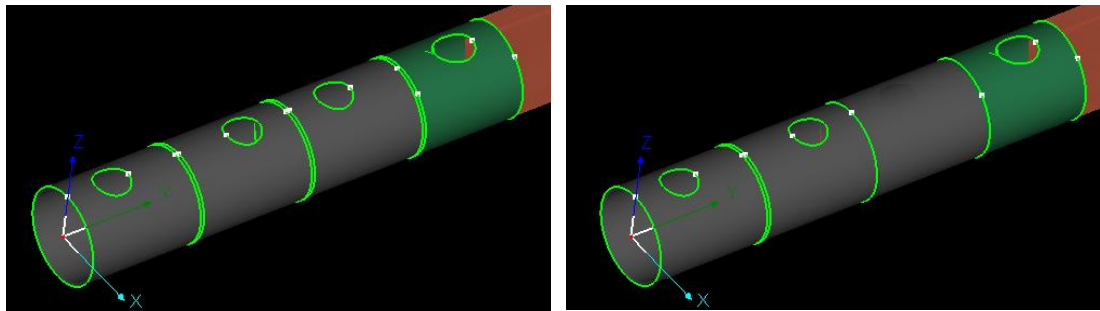
After FlyCut line is applied, if you want to adjust FlyCut parameters, it is necessary to select the part and click Cancel FlyCut before new FlyCut lines are generated.



2.3.11 Delete

To delete the original cutting path on the part, you can press “Delete” on the keyboard after selecting specific cutting path.

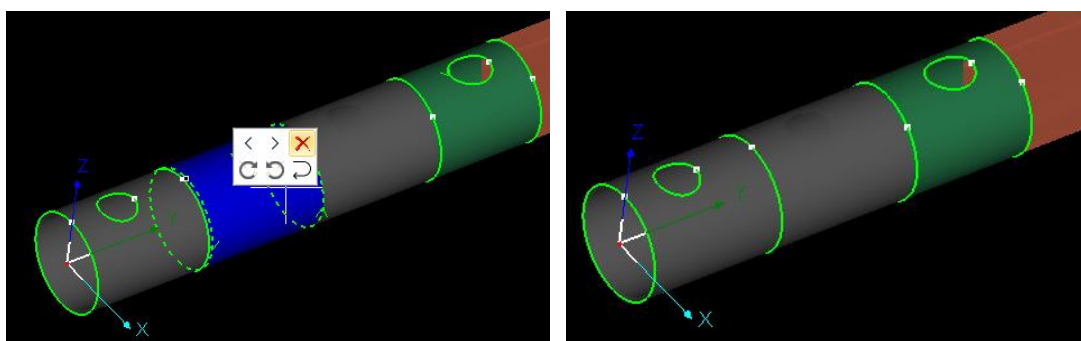
Please note that, only the cutting path selected will be deleted, but the parts amount will not be changed by this operation. In following figure, there are 3 gray and 1 green parts in current nesting, if you select gray one and press Delete, then all cutting path of which will be deleted, but it still occupies a part position on original tube. So this nesting result has 3 gray and 1 green parts still as TubesT considers it as a part without actual cutting path.



If you want to delete one of the nested parts on tube, left click the part and click

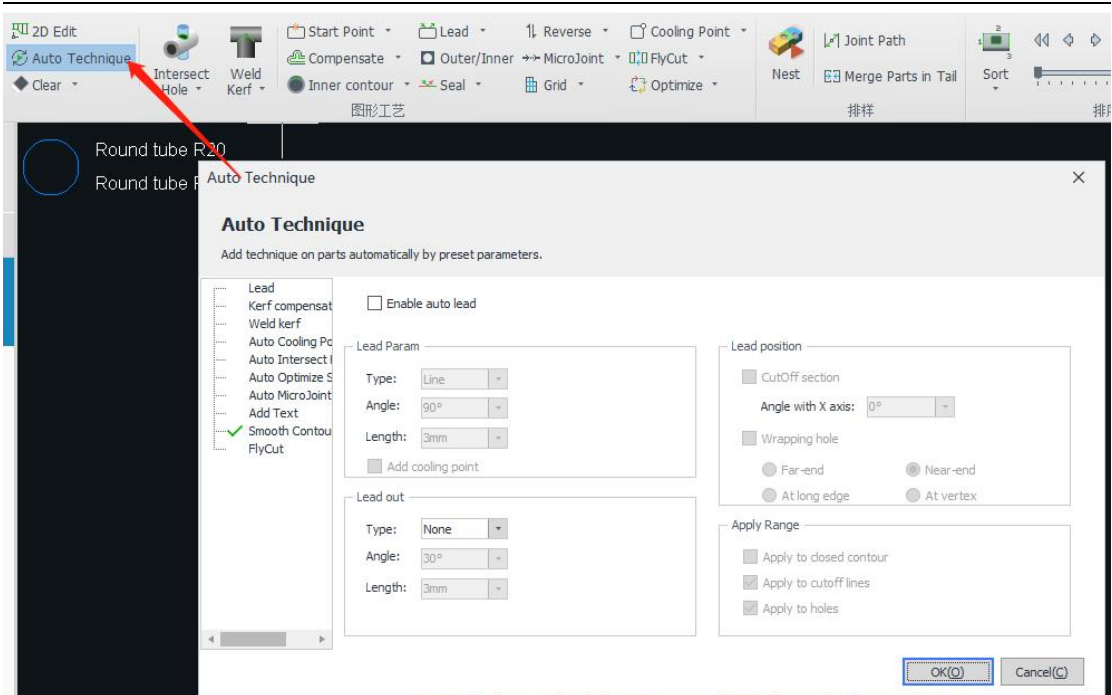


in the pop-up window.




2.3.12 Auto Technique

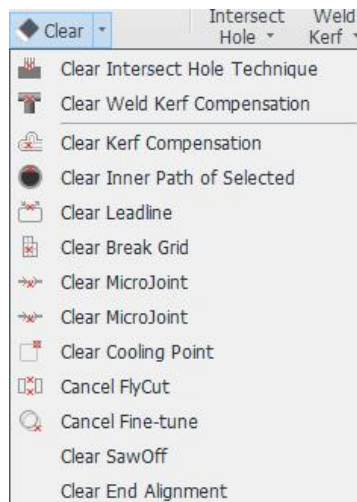
The preset Auto Technique will be added to the parts imported later to achieve Batch Apply of techniques.



(If the part has been imported into TubesT before Auto Technique is enabled, it will only be validated to the parts imported later, but not the parts imported already.)

2.3.13 Clear

Clear is to remove the techniques applied on the parts or graphics. Click  Clear to activate the Clear tool, the cursor will be changed into a small white square. Point it on feature tags of Leadline, Cooling Point, MicroJoint, Compensated Path, Grid etc. to delete applied techniques. Otherwise, you can clear designated technique applied on selected parts or graphics in the drop-down menu.

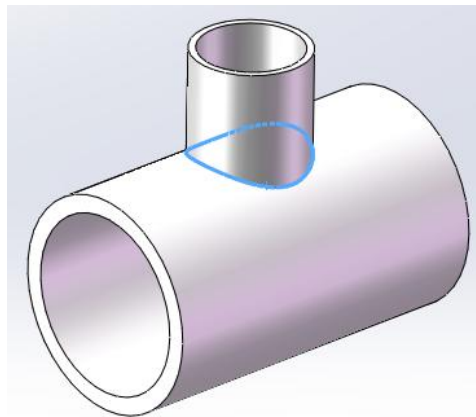




2.4 Advanced Technique

2.4.1 Intersect Hole

The hole dimension is not same in outer wall and inner wall respectively as the tube will rotate in cutting, which cause intersect tube cannot fit in.

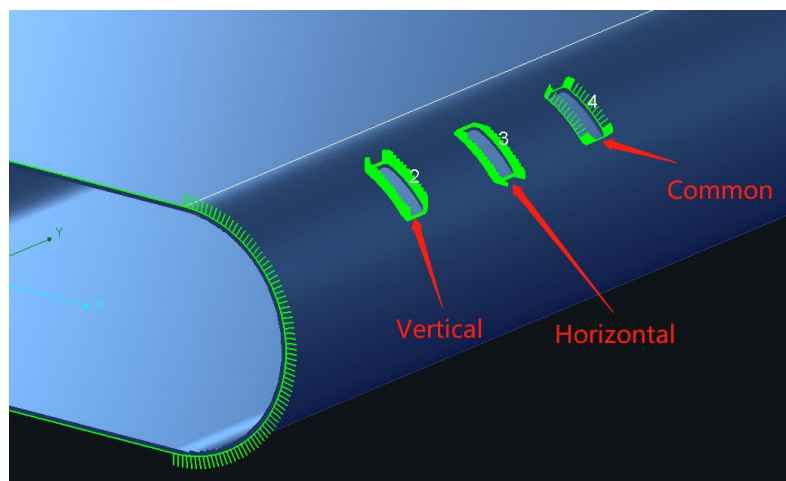


To make hole cutting fit for intersection, you have to apply “Intersect Hole” technique, by which tube doesn't rotate in hole cutting on curve surface.

Three styles for Intersect Hole are included in TubesT: Common Intersect Hole,



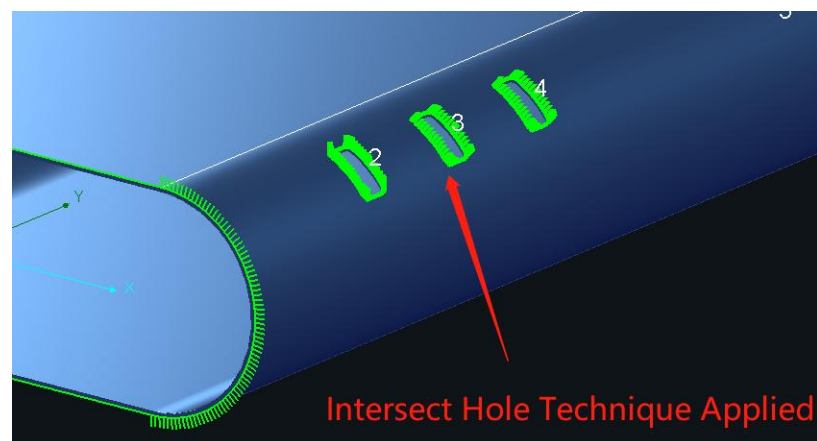
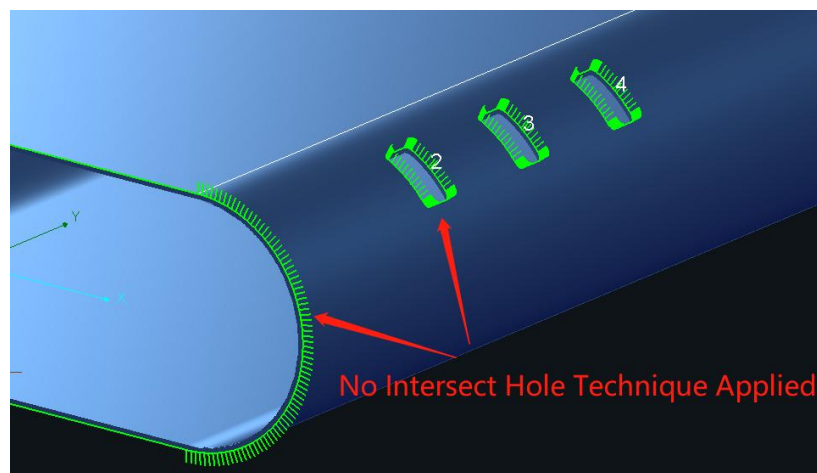
Vertical and Horizontal. Common Intersect Hole, the cutting direction of which is vertical to the plane where the intersect hole is on, is usually applied for the holes on round tube; Vertical Intersect Hole, the cutting direction of which is vertical to YOZ plane, while Horizontal, to XOY plane.





As the part shown in the picture, for Graphic 2, the tube will be cut with fixed angle and no rotation; for Graphic 3, the tube will rotate 90° in CCW and then be cut with no rotation; for Graphic 4, the cutting path in X direction will be machined by tube rotation.

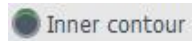
The shape applied Intersect Hole technique will be highlighted by bold lines. Otherwise, you can tell the difference between Intersect Hole and common holes if Display 3D Vector is enabled in Display.



2.4.2 Inner Contour

There are two contours, Inner Contour and Outer Contour, for the part on the tube by the nature of wall thickness. For the graphic with different Inner/Outer contours as tilted holes, they will get different dimensions when machined.

By default TubesT will apply Outer Contour as the cutting path, you can click

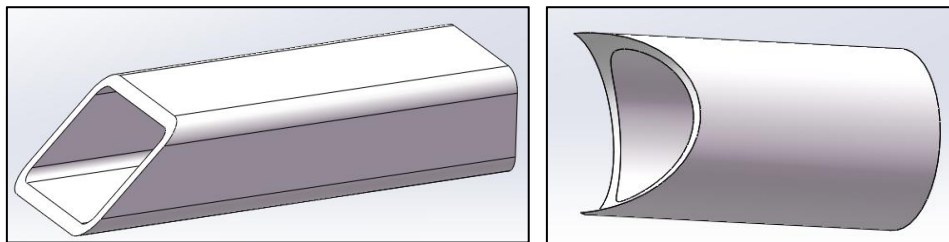



to set Inner Contour as the cutting path. If you have selected Inner Contour when import drawings, you cannot convert it to Outer Contour in TubesT; If you have selected to keep both contours when import drawings, you have to manually delete extra cutting path and only keep the ones needed.

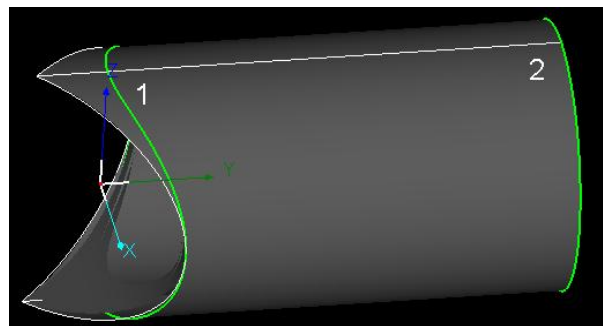
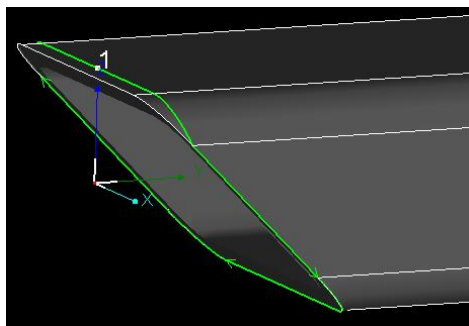
You can clear existing Inner Contour settings in the drop-down menu of Inner Contour.

2.4.3 Weld Kerf Compensate

The cutting direction for parts is always vertical to XOY plane as the laser beam is along with Z direction and tube along with Y in general laser cutting. As shown in the below, the common laser cutting machine can't machine this kind of parts which need bevel angle on the section, and it will lead that the parts finished fail to be assembled perfectly.



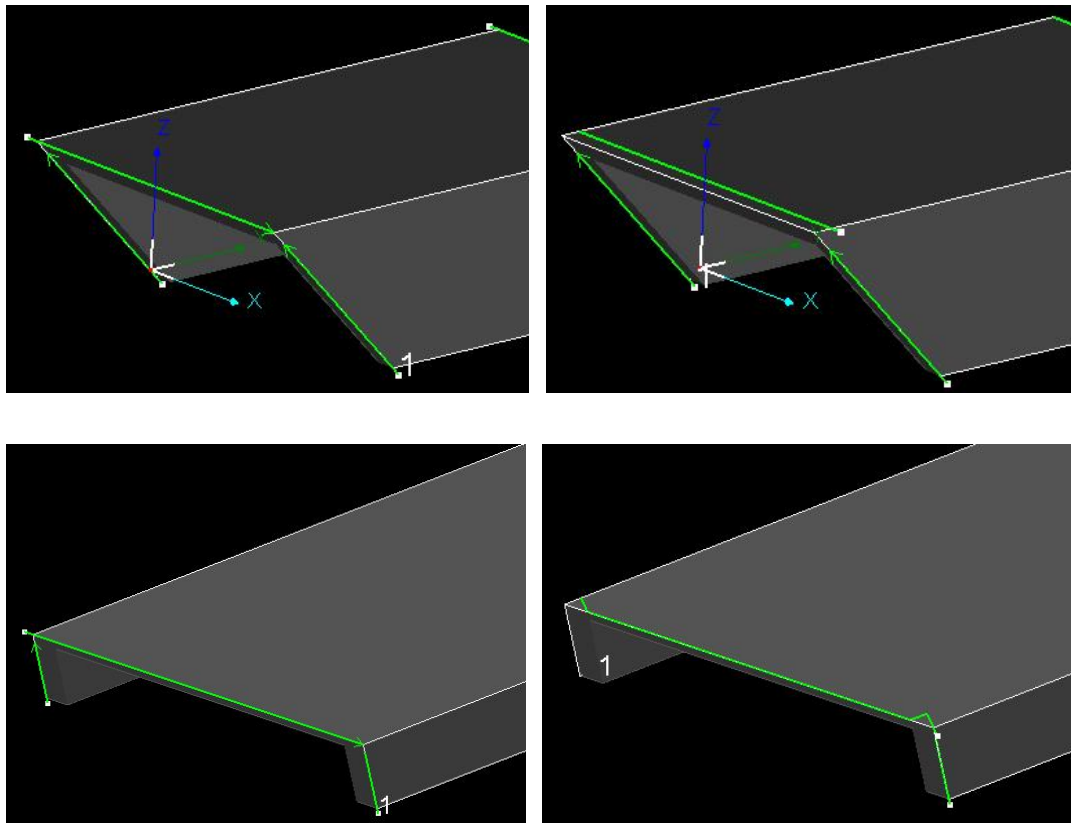
Select this kind of cutting path or parts and click  , then TubesT will generate proper cutting path according to the section information, and the parts by this can be assembled normally.





“Weld Kerf” can also be used to offset tilted cut-off section of U Steel.

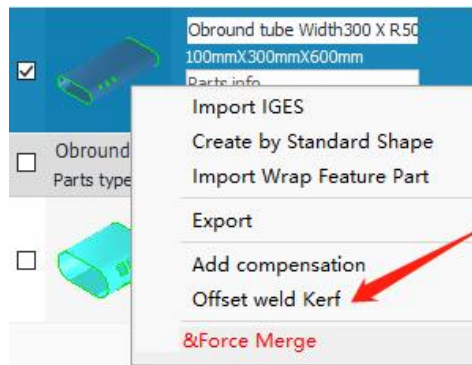
The below shows two different bevel cutting methods for U Steel and the cutting path created by Weld Kerf.



In the drop-down arrow of Weld Kerf, Offset All Sections is to add weld kerf for all sections in current interface and cancel it by the option Clear Weld Kerf in the

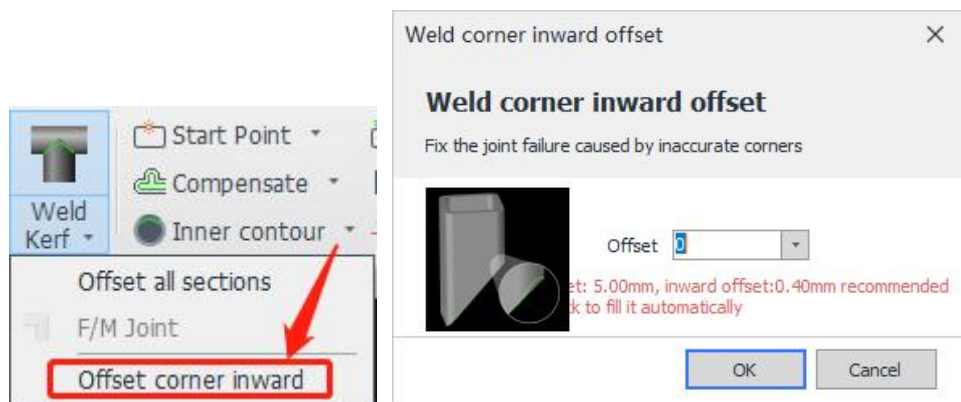


drop-down arrow of Clear. Otherwise, it is possible to select unnested parts in Part List and right click the mouse, click Offset Weld Kerf to add weld kerf for them.

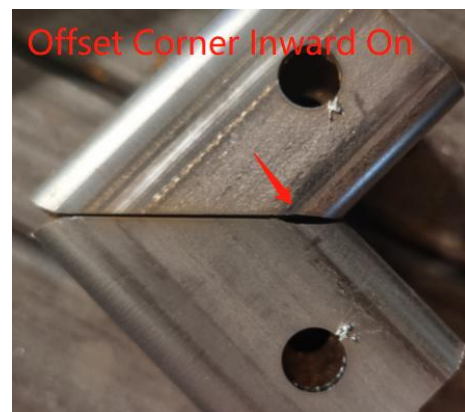


2.4.5 Offset Corner Inward

For rectangular pipes with large chamfer and thick wall, the interference problem during assembly can be solved by Offset Corner Inward when bevel cutting and weld kerf is needed. To enable it, you can check Offset Corner Inward in the drop-down menu of Weld Kerf and add Welf Kerf for that parts.



The effect contrast is shown in the following figure:

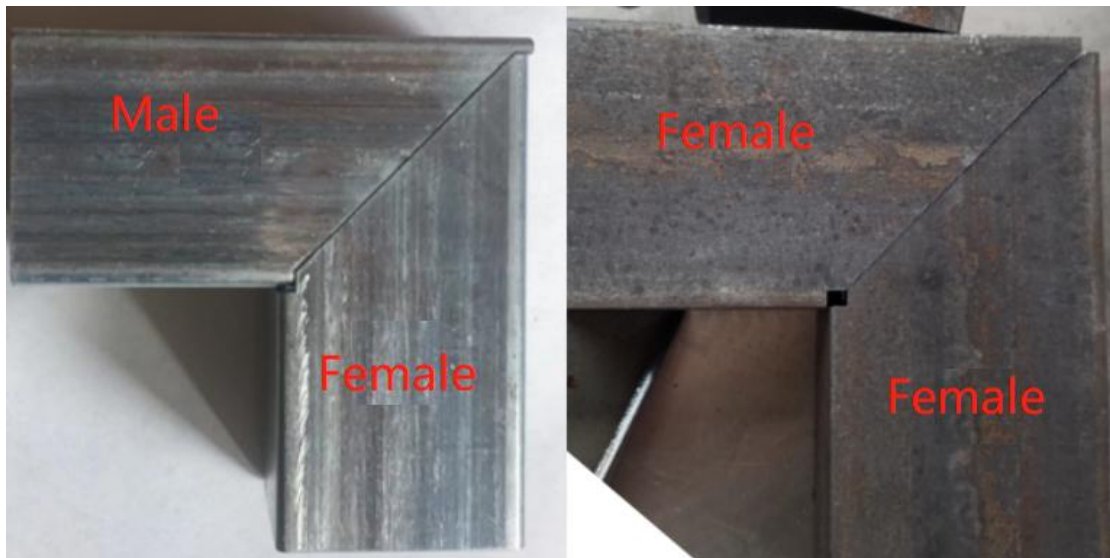




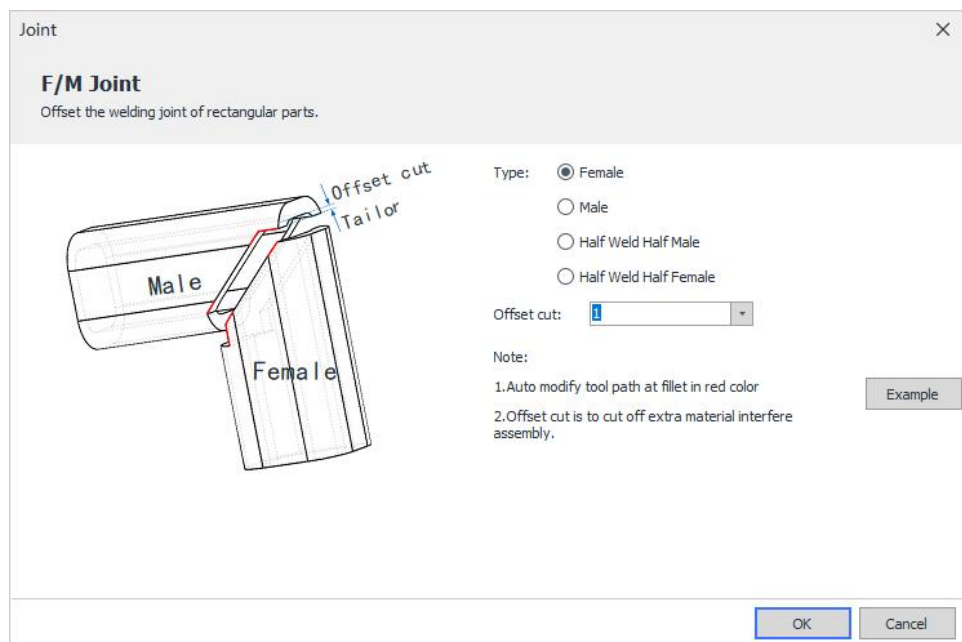
2.4.6 F/M Joint

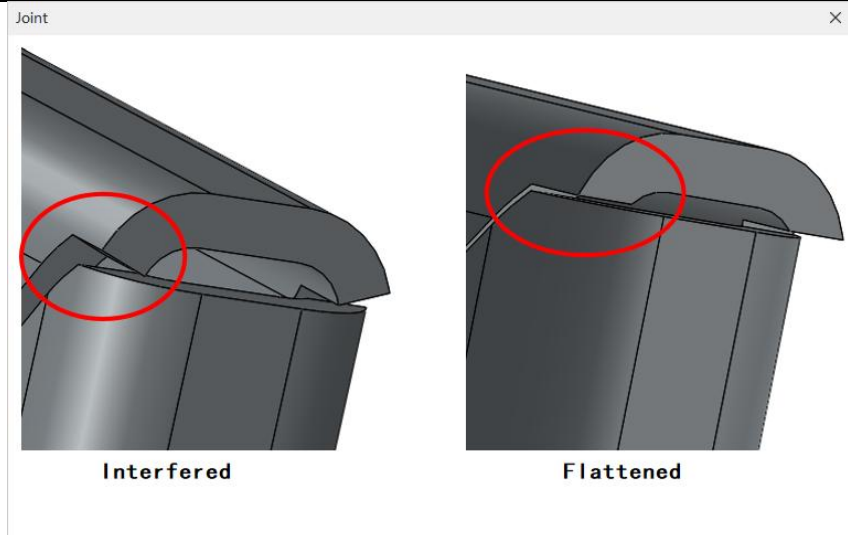
By changing the cutting path, TubesT can differentiate Female and Male parts, capable to realize the L-shaped assembly of rectangular parts.

It can be seen from the figure that “Male +Female” or “Female + Female” can get better L-shaped assembly effect. But it doesn’t work if the combination is “Male+Male”.

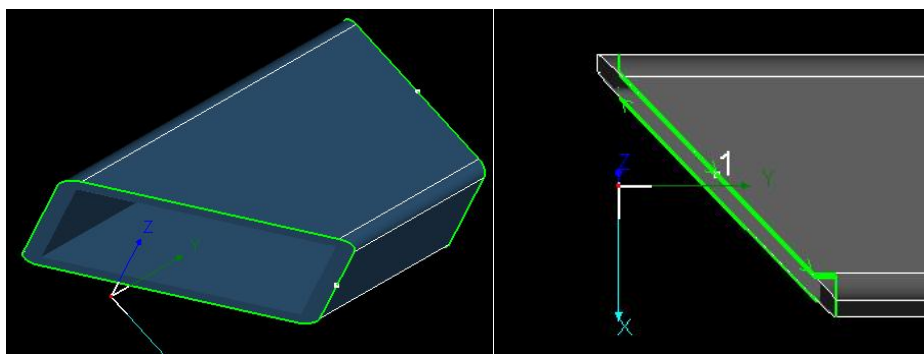


For the parts featured by bevel, click F/M Joint in the drop-down menu of Weld Kerf.



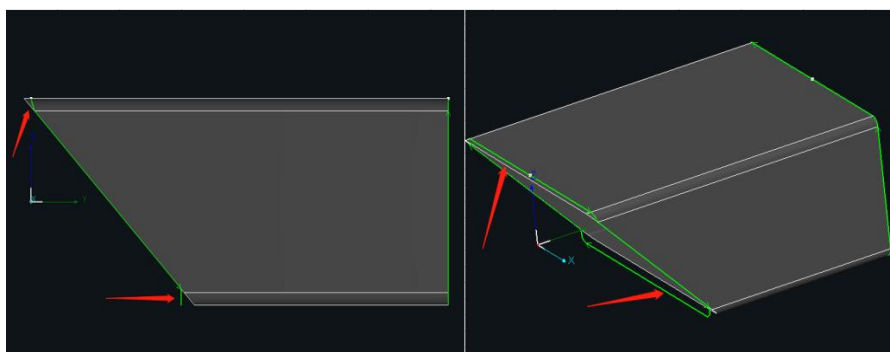


Set the current part bevel cutting type to “Male” or “Female” and the Offset Cut parameter, then the cutting path will be changed.



2.4.7 Half Weld Half Male/Female

For thick bevel-cut rectangular tubes with large fillets, due to the interference caused by inner bumping, it is still difficult to get the parts assembled perfectly after weld kerf added on the bevel-cut section. To solve this, you can apply Half Weld Half F/M function.






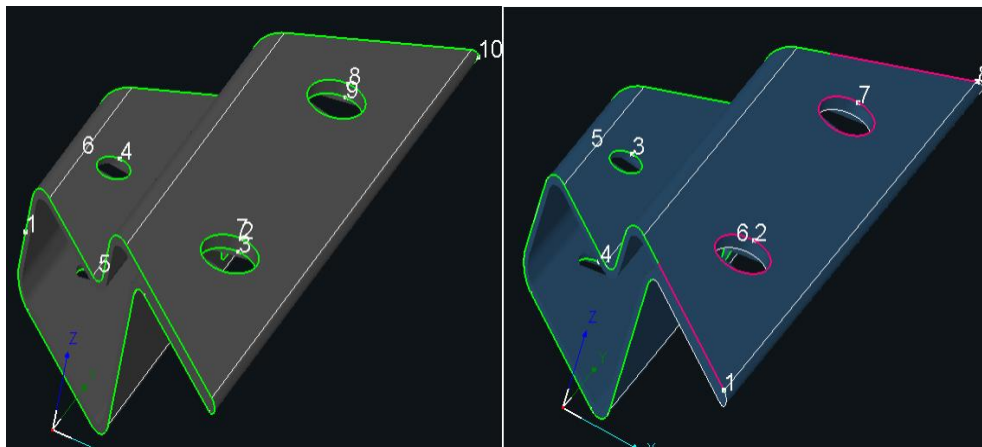
2.5 Parts Modification

2.5.1 Profile Edit

2.5.1.1 Custom Profile Contour

For free-formed tubes that need to modify the cutting path, there may be a need to modify the cutting path separately for each cut-off line and each part. Now, it is

only necessary to modify it in the drop-down menu of  and apply it for the section of all same parts.



With many curve tools in it, it is possible to break, joint or delete the cutting path; also you can add Leadlines or Gap for the cutting path according to the actual needs; and you can confirm whether the cutting sequence is reasonable. After the modification is completed, click Apply to All Parts with the Same Section. If there are multiple free-formed parts with the very section, then all parts will be modified to such a special cutting path.


2.5.1.2 Adjust Fillet

You're allowed to modify the fillets dimension within 3mm by Adjust Fillet, which is always used to modify the drawings quickly to improve the dissatisfied cutting results as the fillet dimension of the drawings is not same to that of actual tubes.

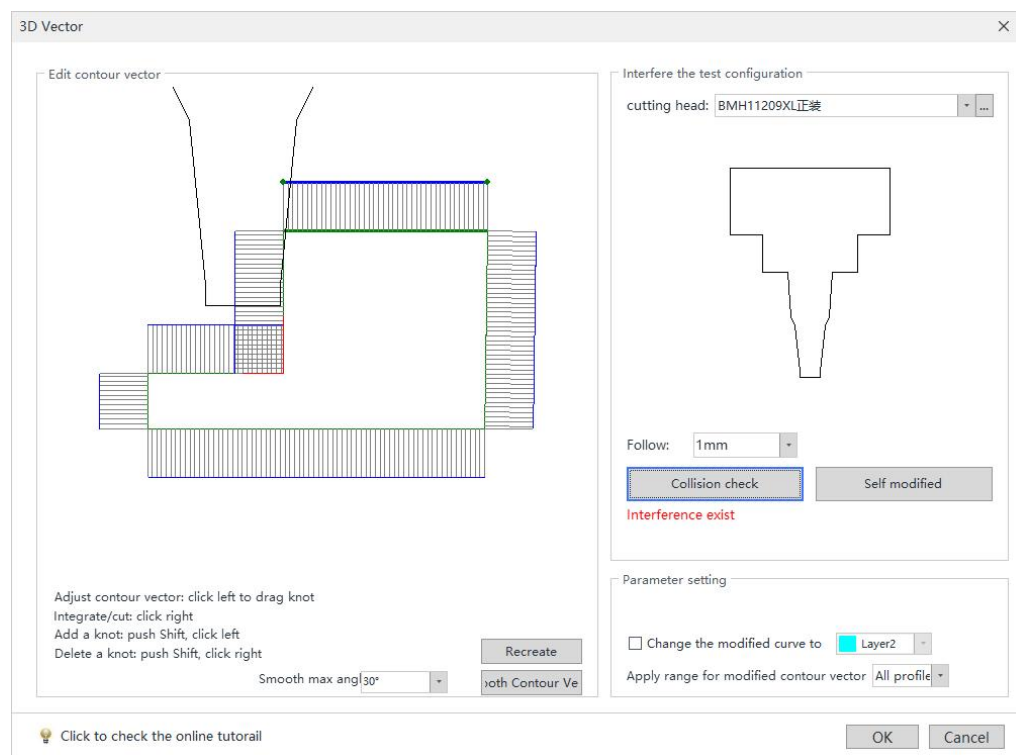


Take notice that this function only changes cutting path, but not the dimension of original drawing. Take a rectangular tube of 40x40, R3, if its fillet is changed into 4mm by Adjust Fillet, it will be machined according to this dimension, but the fillet dimension in drawings is still 3mm, which cannot be nested together with the parts of 40x40, R4(if needed, please see more details in Dimension Tolerance).

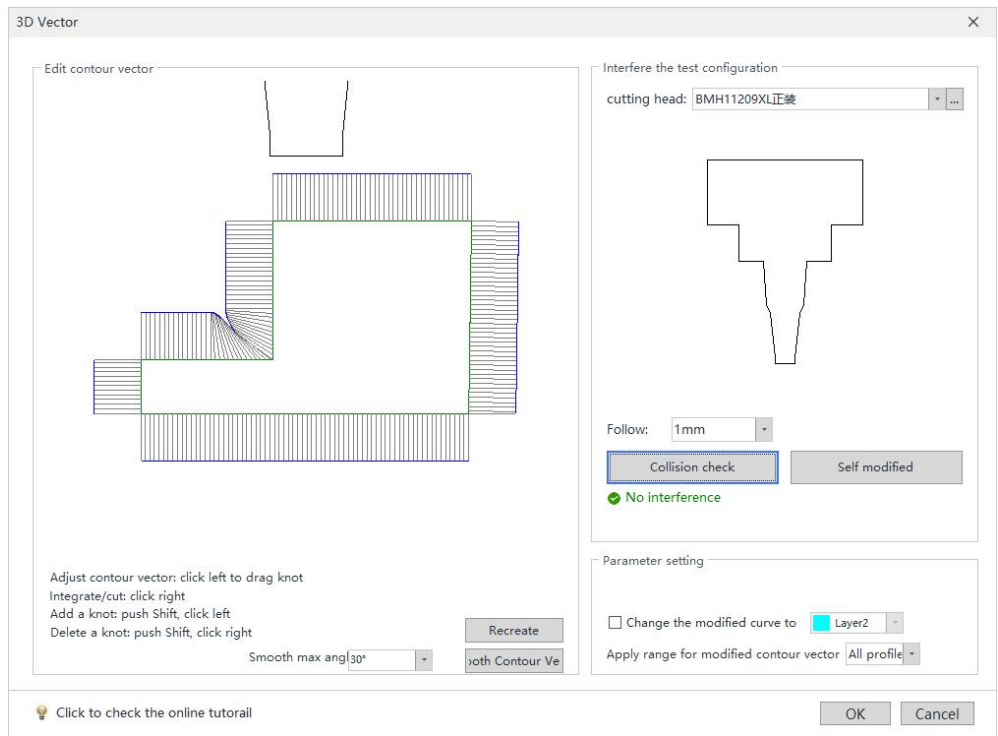
2.5.1.3 3D Vector

Click  **3D Vector** to edit the contour vector of current part automatically or manually.

Laser head will collide with the tube if the contour vector of some parts is not modified before cutting. The original contour vector of the part is as shown in the figure.



Select the cutting head model same to actual one in the machine according to the notes, then click Self Modified to automatically correct the vector; hold Shift and left click the mouse on the contour to create new nodes on it, then you can modify it by dragging the node.



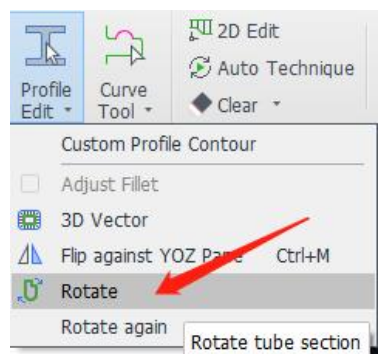
For more details please refer to related article Edit Normal Vector of Contours on our official website.

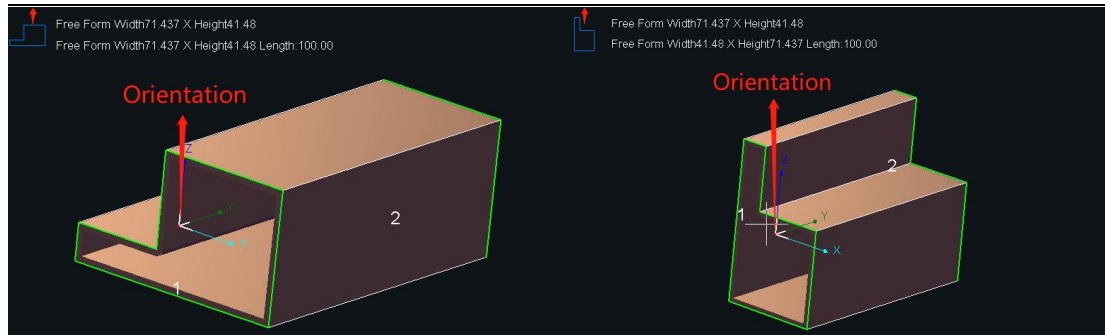
2.5.1.4 Flip against YOZ Plane

This function is to mirror the parts selected against YOZ plane.

2.5.1.5 Rotate

For free-form tubes which failed to be centered, you have to make sure the tube face clamped in machine is in same direction with drawing in the software. And Rotate is to change the orientation of the tubes in software.



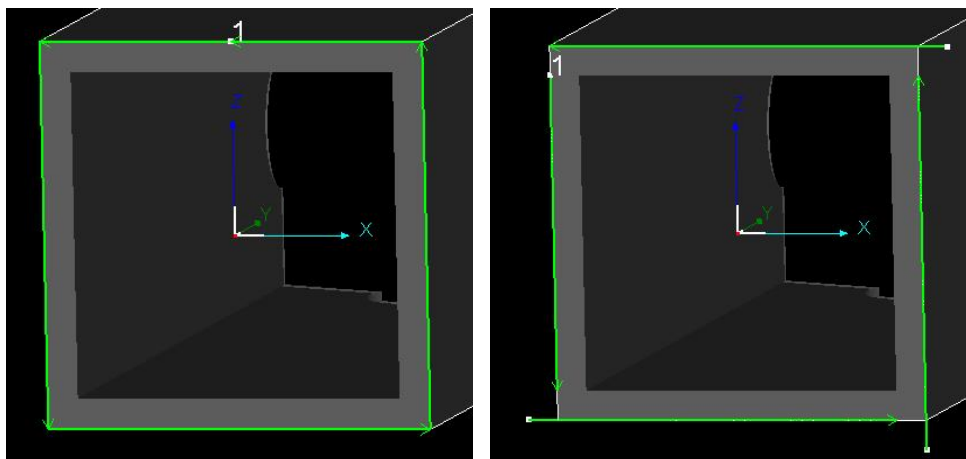


For more details please refer to related article on our official website.

2.6 Curve Tool

2.6.1 Quick Rectangle Path

For the square rectangle profile with sharp corner, it's hard to cut through the corners. You can select the part and click "Quick Rectangle Path" to create special cutting path in four segments (as the cutting path for U-Steel),



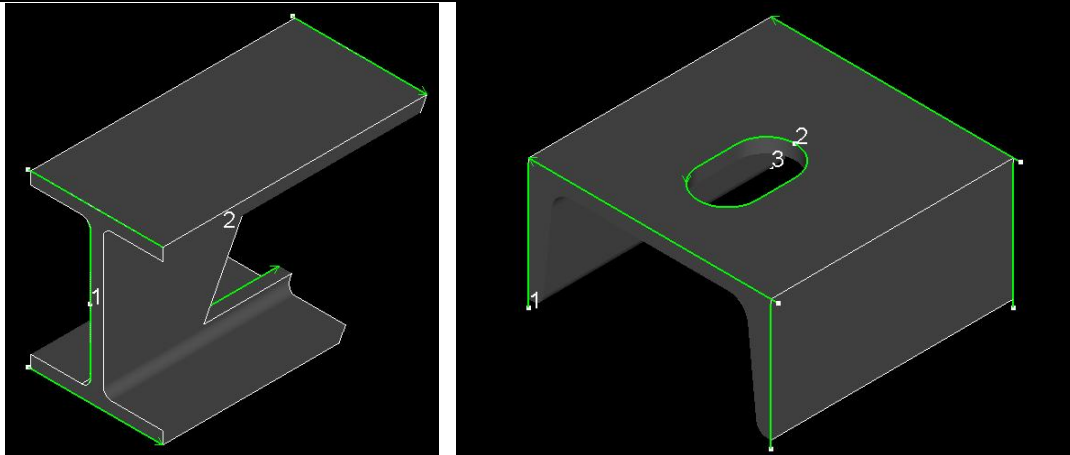
2.6.2 Profile Path

Profile Path is to modify the cutting path for profile steel of L shape, H shape, C shape etc., for the cut-off section is different from closed shape.

When importing the parts as mentioned above, TubesT will generate this kind of special cutting path if you enabled Section steel: L/C profil

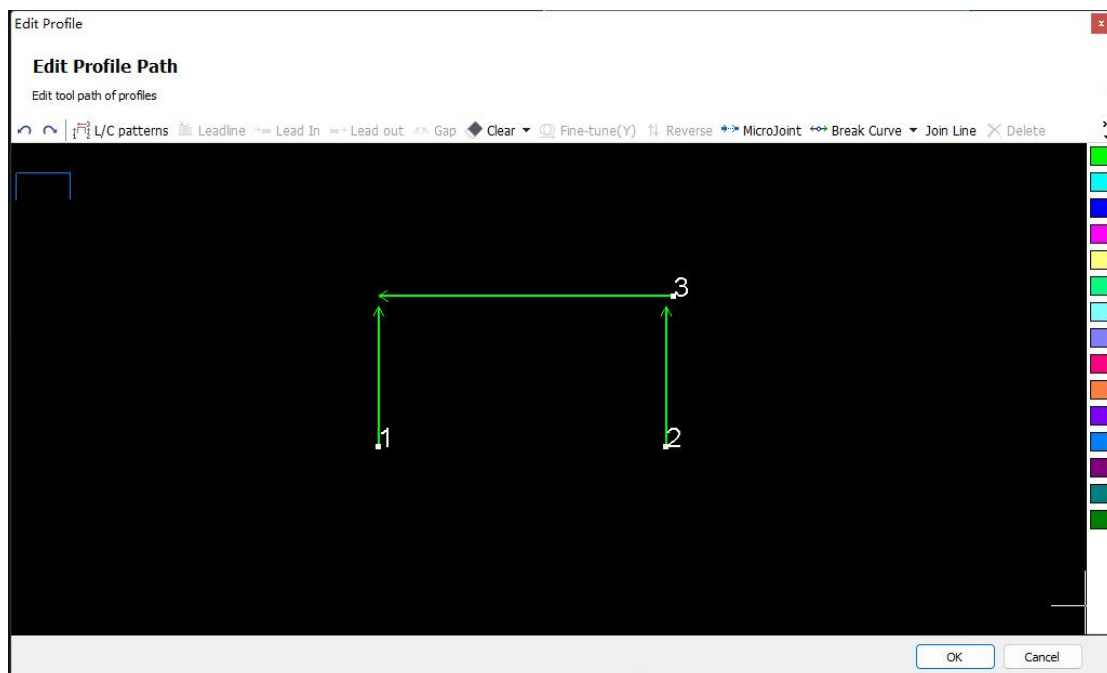
Out parameters preset in .

Below figures demonstrate the special cutting path of H and U steel.

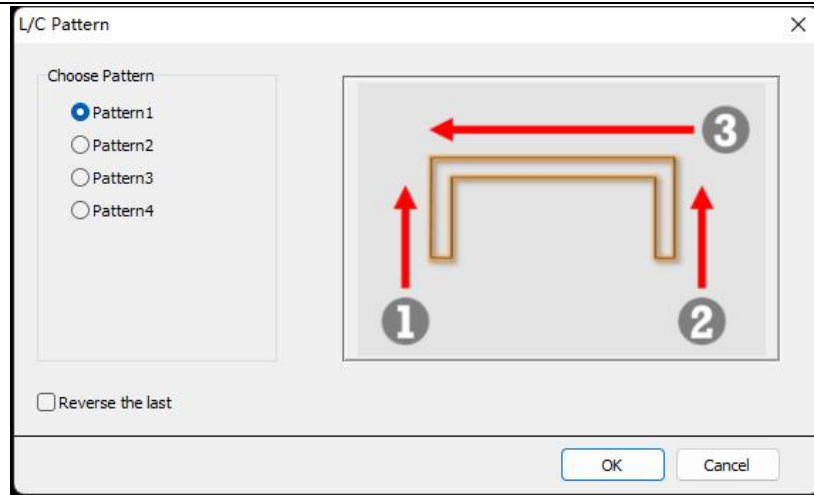


As shown above, even though there are three Start Points for this cut-off section (3 separate segments for one single cut-off section), it is still considered as a whole with integrated number 1, any single one in which can not be edited separately.

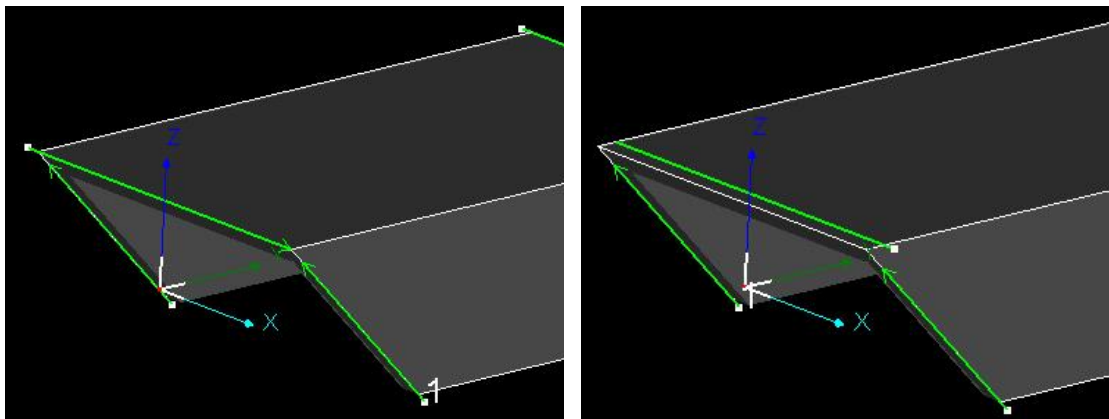
Select a profile path and click **Profile Path** in the drop down menu of Curve Tool to modify current cutting path further.



L/C Pattern can change cutting path sequence of L/C-shaped section. The below shows the profile path of C-shaped steel, whose cutting sequence is to cut side path 1 and 2 first, then 3 to cut it off. It is possible to change the current ones into the below by L/C Pattern.



Fine-Tune(Y) is to move selected cutting path towards Y direction in precise distance for quick adjustment and cutting. For example, you can adjust the cutting path from the left image to the right during bevel cutting of C-shaped steel .

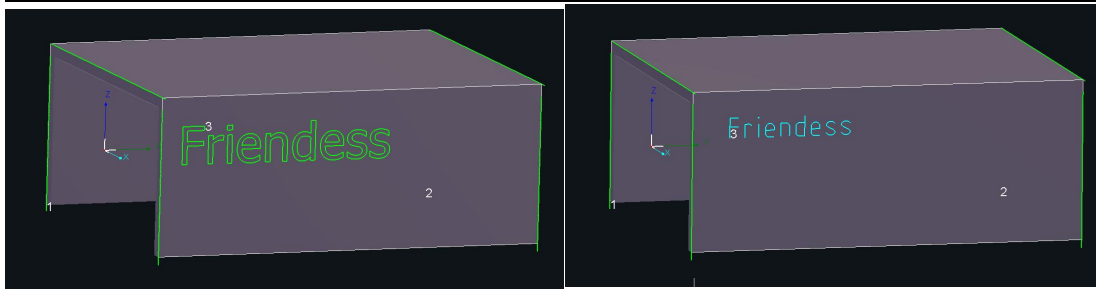


For the imported profiles, if you don't want to cut them by facets, you can also select the section trajectory and click Cancel Profile Path Modification in the drop-down menu of Curve Tool.

2.6.3 Add Text

2.6.3.1 Add Marking Text Manually

In the drop down menu of Curve Tool, click Add Text to add hollow text or text in *.SHX format.



If you need to mark the parts with numbers for sorting and picking after production, set the Marking to another Layer in which the cutting parameters can be modified on TubePro.

2.6.3.2 Batch Add Text

When there are many kinds of parts with large amounts, and you want to add Marking Text to the parts, click Batch Modify Parts button to enter the table page, and then fill in the Marking content in Column B. If you want to mark the part name directly, it is very convenient to copy and paste the whole list of part names with Ctrl + C and Ctrl + V.

	A	B	C	D	E
1		Draw Text			
2	Part Name	Text	Margin to Tube Front	Number	Price
3	Round tube R20	A-1	10	1	0
4	Obround tube Width300 X R50	B-2	10	1	0
5	Obround tube Width300 X R50	C-3	10	1	0
6	Obround tube Width300 X R50		10	1	0
7	Obround tube Width300 X R70		10	1	0
8	Square tube Width40 X R3		10	1	0
9	Square tube Width60 X R5		10	1	0
10	Free Form Width71.437 X Height41.48		10	1	0
11	Rect tube Width30 X Height20 X R0		10	1	0
12	L tube(L) Width100 X Height100		10	1	0
13	U tube Width80 X Height40		10	1	0

2.6.4 Draw Wrap

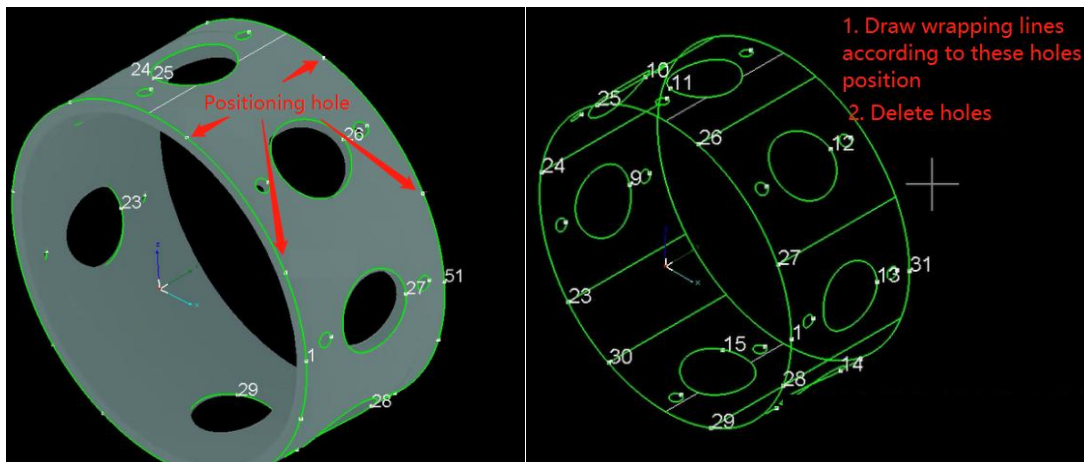
Draw Wrap is to draw graphic wrapping on tube face (press ESC or right click the mouse to exit).

Cursor will automatically grip the nodes on graphics. To make sure wrapping graphic in precise position, you might need to draw small positioning hole on tube

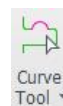



face then delete them after finish wrapping graphic.

To cut the facet part from tubular body in below image, you need to array the wrap holes and draw the positioning holes on tube face first according to which you can draw wrapping lines and delete them for actual cutting.



2.6.5 Break Line

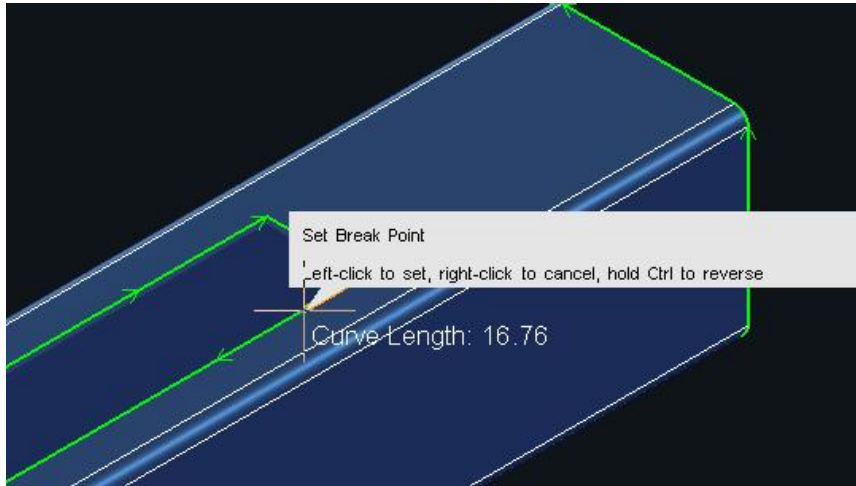


Click Break Curve in the drop down menu of  to break a whole line into multiple ones, benefit to edit them separately (press ESC or right click the mouse to exit).

You can also realize precise breaking by Break Curve with Basepoint, and after selecting the starting point of measurement, two distances will be displayed: curve

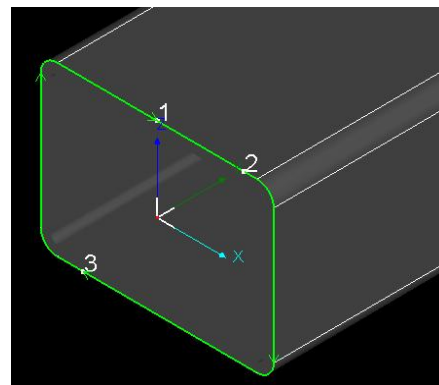
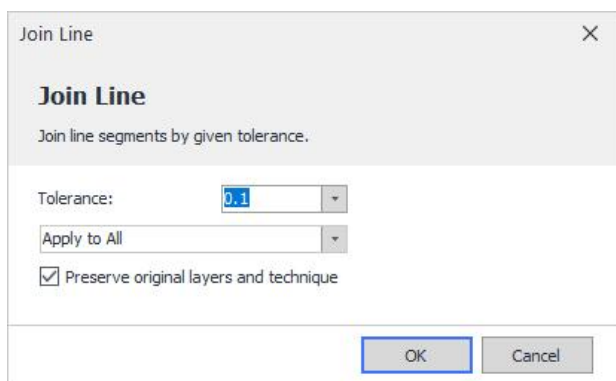


length (the length of yellow part) and two-point distance (the distance from the current cursor position to the starting point), which can accurately locate the position where the curve needs to be segmented.



2.6.6 Join Line

Join Line is to join multiple line segments as one by given tolerance. For example, there are 3 line segments in below image, and the machine will locate and pierce on each line segment respectively, it is possible to join these three into a whole one.

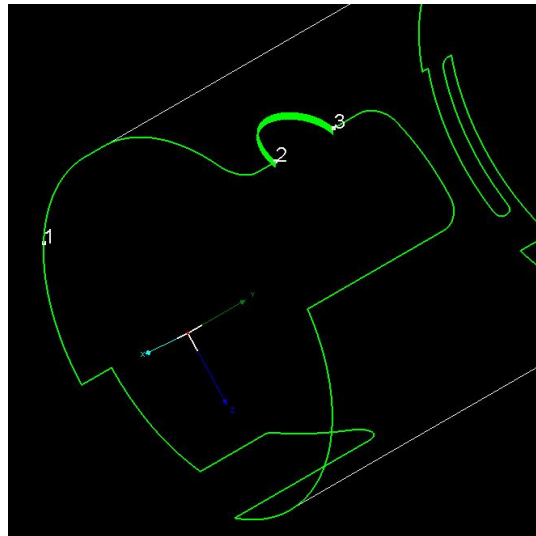
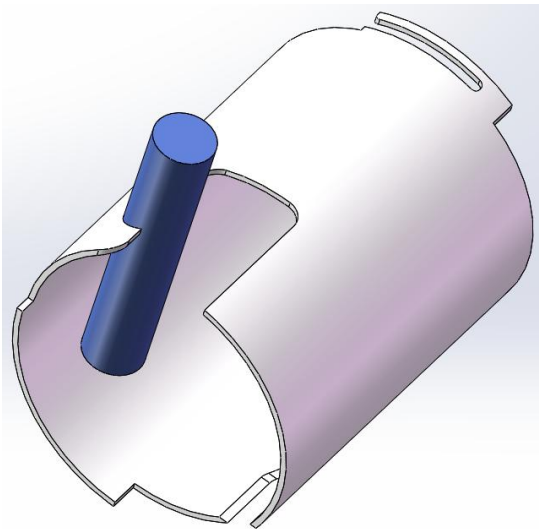


Original Layers or Intersect Hole technique etc. will not be changed if you enable the option Preserve Original Layers and Technique.

Take the tubular part in the figure as an example, it is necessary to apply Intersect Hole technique on the arc if you want to assemble this arc with the branch tube (you can separate this single arc from the whole by Break Curve and then apply Intersect



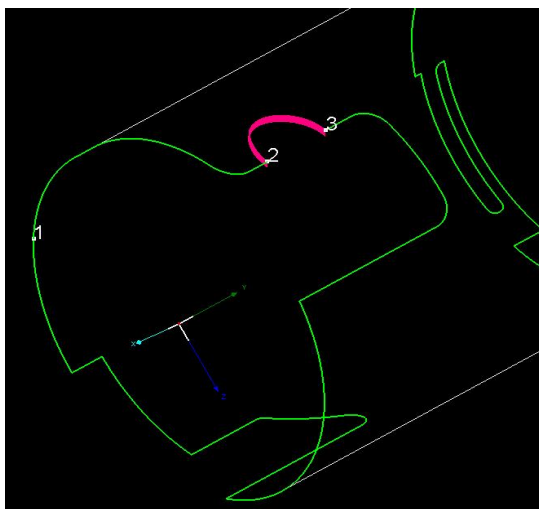
Hole technique on it).



Compared with common cutting path, the one with Intersect Hole technique will get thicker wall when cutting the very arc. To have better cutting result by adjusting the cutting parameters(as slow down the cutting speed), you can set another layer with different color on this cutting path(it is possible for TubePro to apply different cutting parameters according to the layers with different colors).

Join these three tool path segments as one in last step by Join Line.

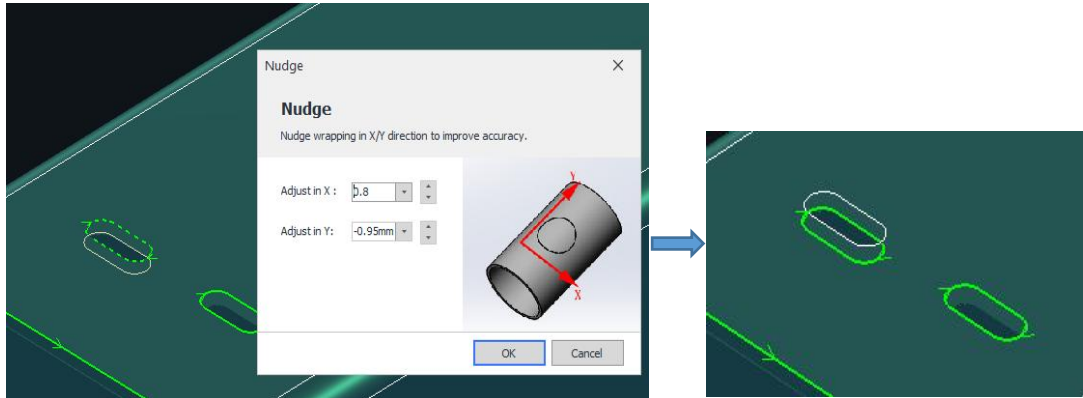
The cutting head will pierce and start cutting at Start Point 1, then continue cutting at arc by the cutting parameters in Pink Layer for Intersect Hole technique, without Stop and Re-pierce.





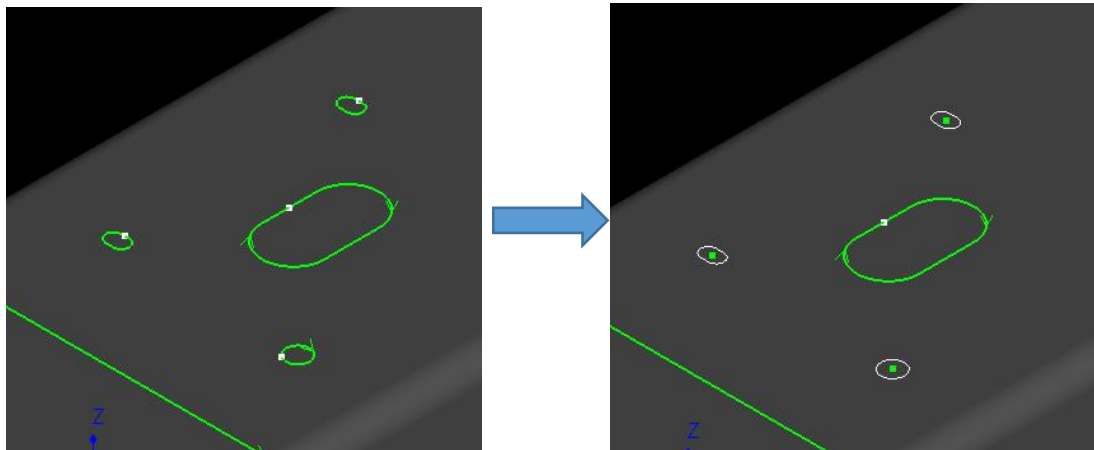
2.6.7 Fine Tune Wrapping

By this option, you can move the wrapping graphic in X or Y direction.



2.6.8 Replace as Point

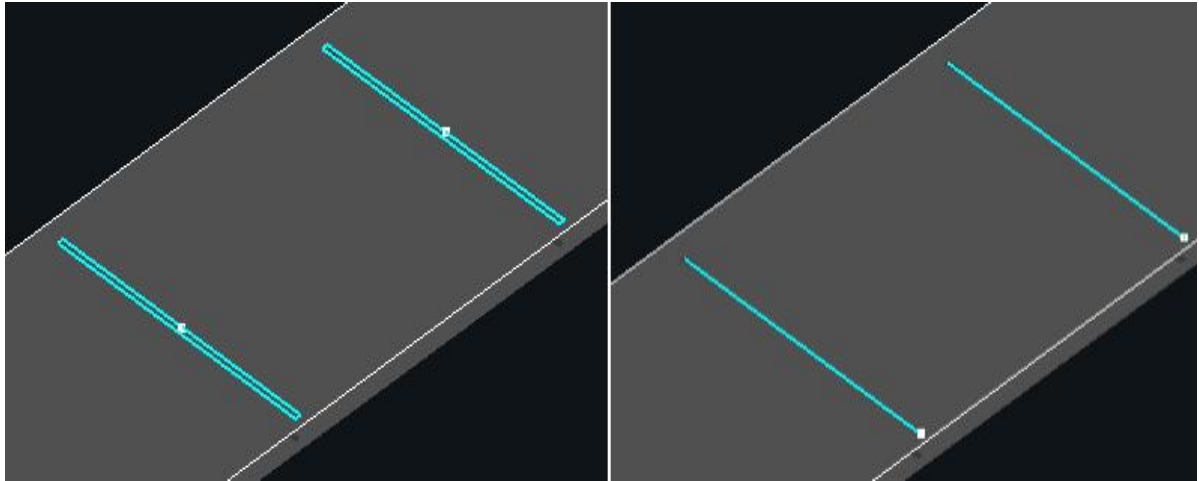
For the part with small dimension which can't be machined by laser, you can replace it with a Pierce Point by Replace as Point. As shown below, the original cutting path is changed from green to white(Not Cut Layer), replaced as a green Pierce Point in the center.





2.6.9 Replace as Line

Because a single line can't be saved in a 3D file, if you want a single cutting path in TubesT, it is necessary to use Draw Wrap or Delete the Curves to realize that. But now, you can use the “Replace as Line” function to replace a narrow strip hole with a line.

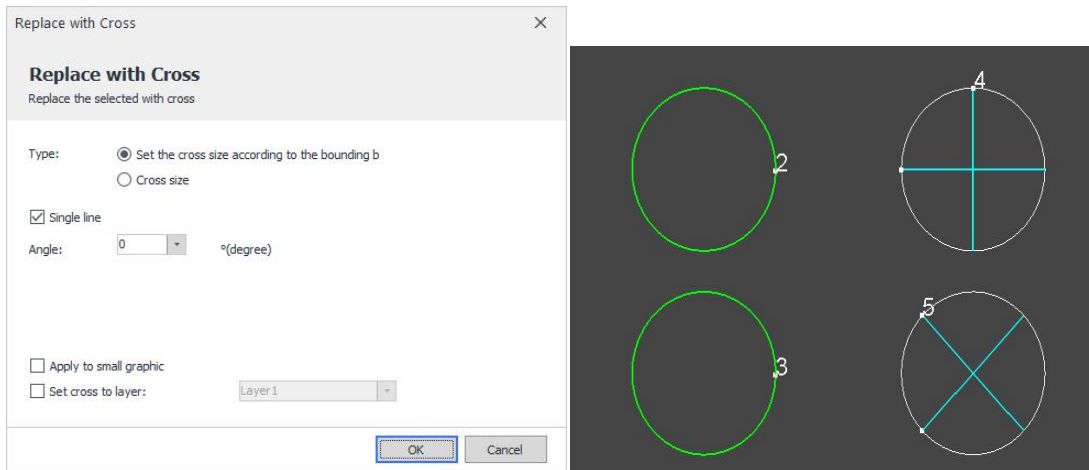


Select the graphic and set “Max Width” in the window of “Replace as Line”. Eventually, narrow holes with the width smaller than this parameter will be replaced as single lines, while the other selected shapes will not be affected.

2.6.10 Replace as Cross

Replace as Cross is applied when some holes are not cut and you want to mark the position with a cross mark. “Set the Cross Size according to the Bound”: For example, a circle with 5 mm diameter will be replaced with a cross whose length of both horizontal and vertical lines is 5 mm.

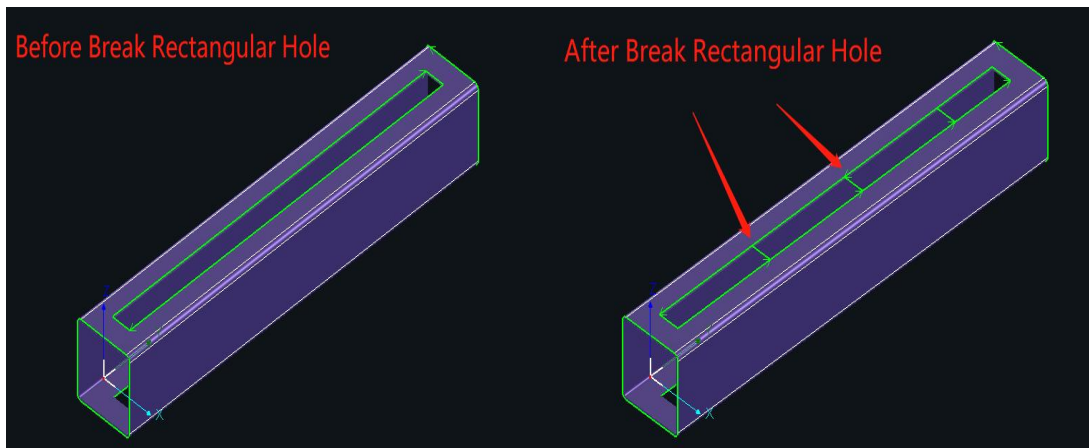
“Cross Size”: You can customize the length of horizontal and vertical lines regardless of the size of the replaced graphic. At this time, the width of horizontal line and vertical line should be set separately.



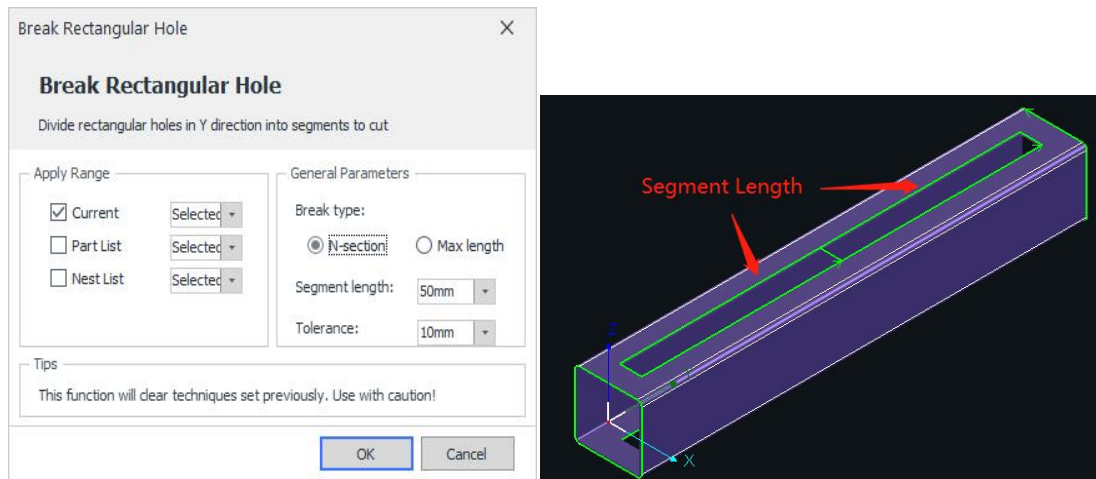
2.6.11 Break Rectangular Hole

For the rectangular holes with long distance in Y direction, it is possible to divide it into multiple holes whose cutting path can be cut by C-shaped Co-edge to avoid repeated tube pulling in Y direction.

Click Break Rectangular Hole in the drop down menu of Curve Tool to enter it. Set the Apply Range of parts that need division and also the Break Type and Segment Length in General Parameters.



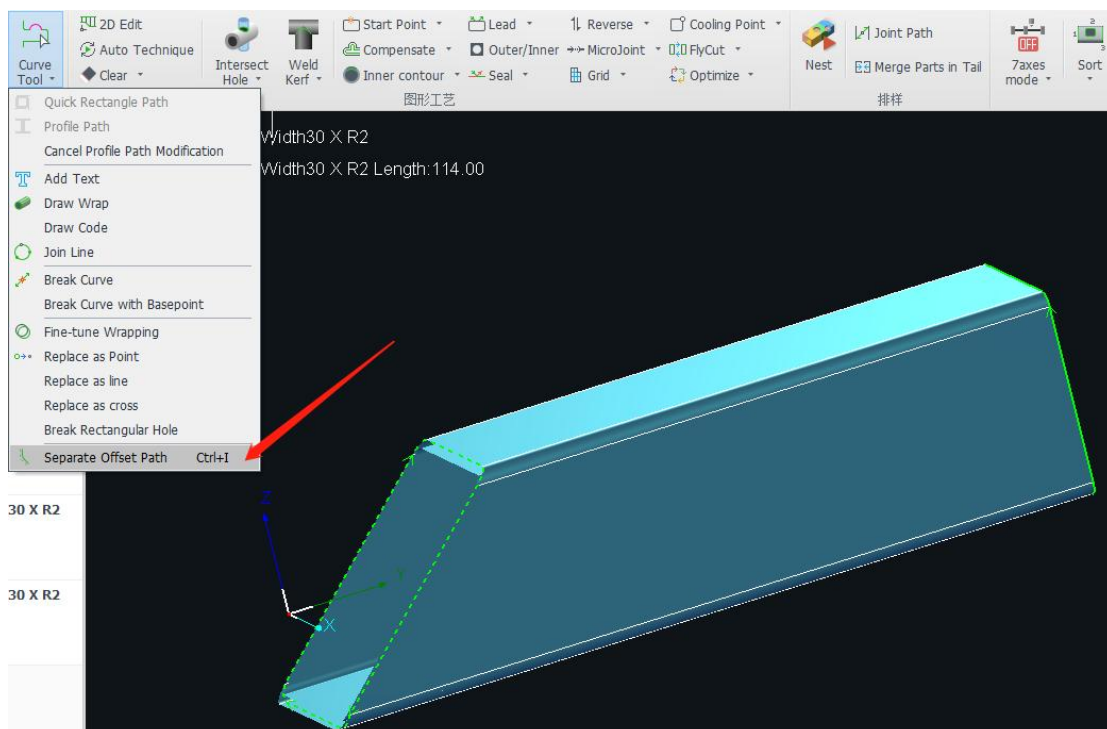
In General Parameters, Segment Length is the length of each small hole when a long hole is divided into multiple segments. When the length of the last segment is less than the set value in Tolerance, then it will be merged into the previous segment.



2.6.12 Separate Offset Path

Separate Offset Path is to transfer the cutting path generated by Weld Kerf or Compensation into separate ones, and the original cutting path will not be saved.

This function is usually used for the parts which need both Weld Kerf offset and Break Curve operation. To make it, you need to add Weld Kerf or Compensation first, then select the cutting path newly generated and click Separate Offset Path(or press quick key Ctrl+I), finally execute Break Curve or other operations after a separate cutting path is created.

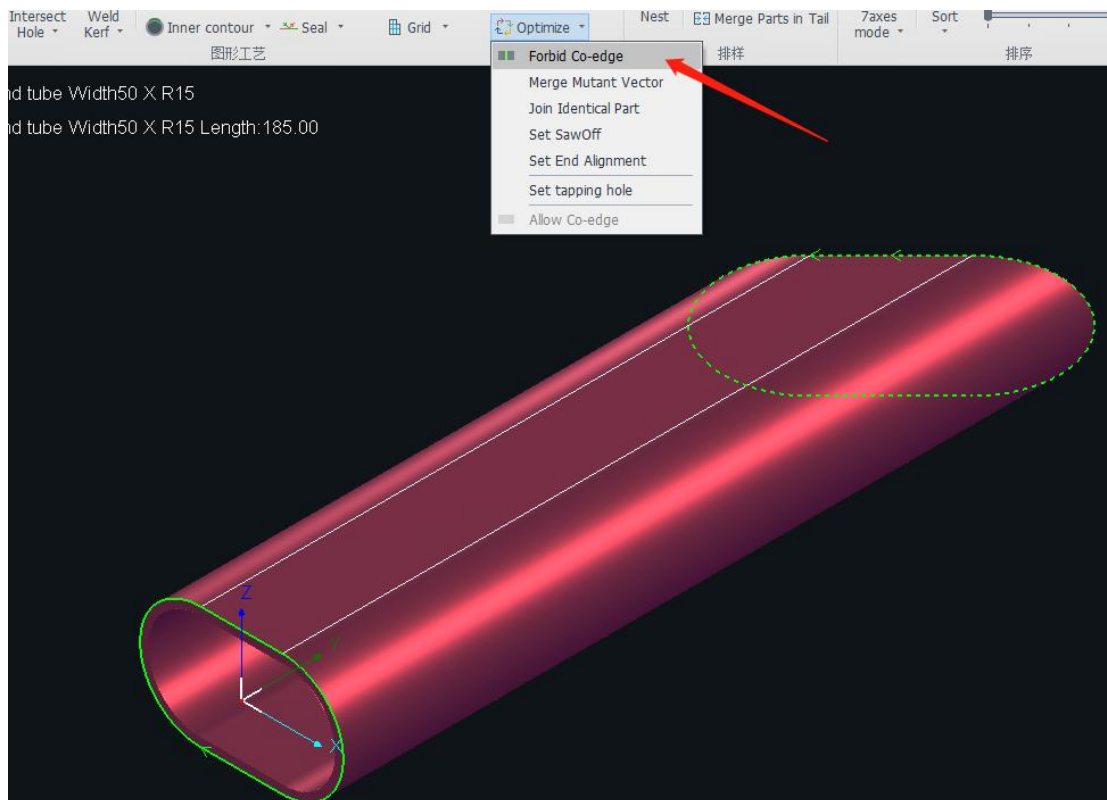




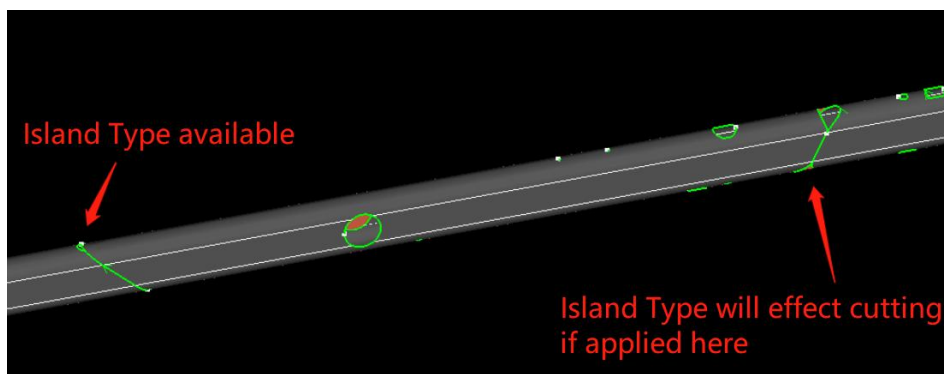
2.7 Optimize

2.7.1 Forbid Co-edge

Before parts nesting, you can select any one cut-off section of parts and click Forbid Co-edge in the drop down menu of Optimize, then the very section will not be applied Co-edge in Auto Nest. But the other cut-off section and the rest parts will not be effected in Auto Nest.

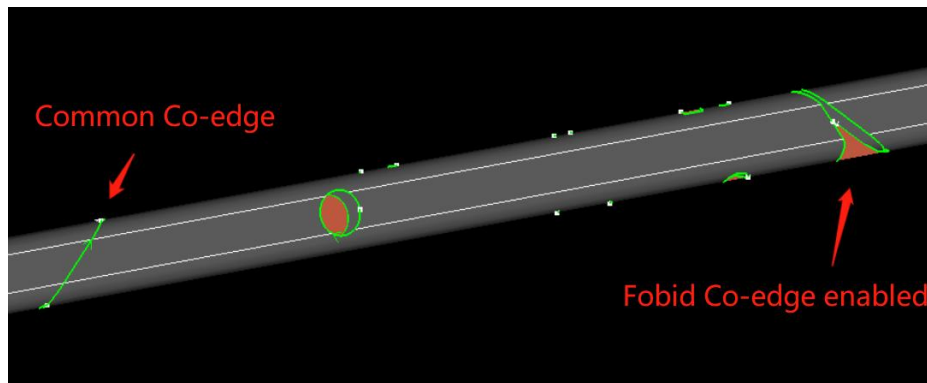


As shown in the figure, the parts applied Island Type Co-edge, will fall off before the cutting of last co-edge line is finished since the co-edge line is too short.





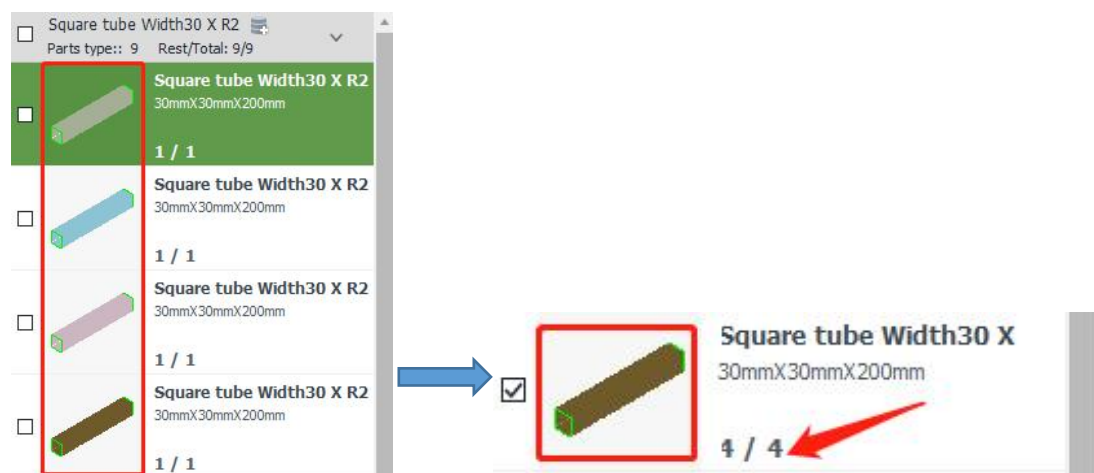
To solve this, you can set the right cut-off section as Forbid Co-edge, the result is as the following:



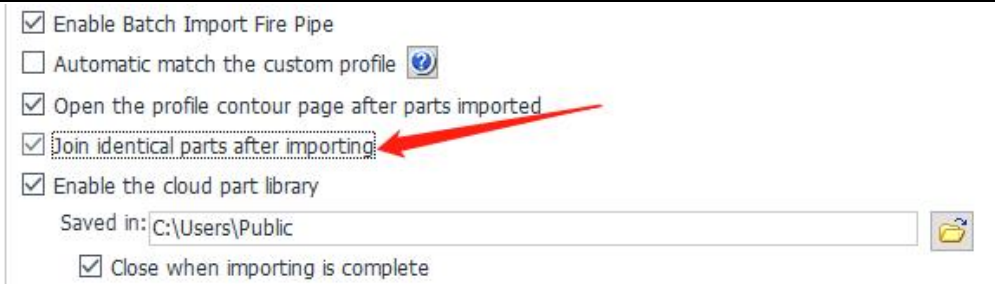
2.7.2 Join Identical Part

When there are N parts with the same cross section, cutting path, size and techniques, the same parts can be recognized and joined by TubesT, so that their amount can be accumulated.

You can check the same parts that have been imported in the Part List, and click Join Identical Part in the Optimize drop-down menu to merge the same parts together and accumulate the parts amount (Join Identical Parts result is shown as below).

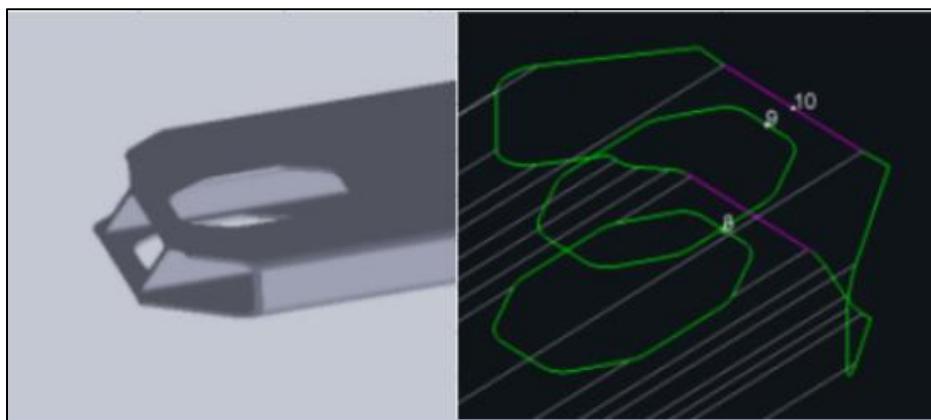


Also you can enable Join Identical Parts after Importing in User Habit Parameters before importing the identical parts (this function will only be validated after it is enabled, but not as soon as enabled).



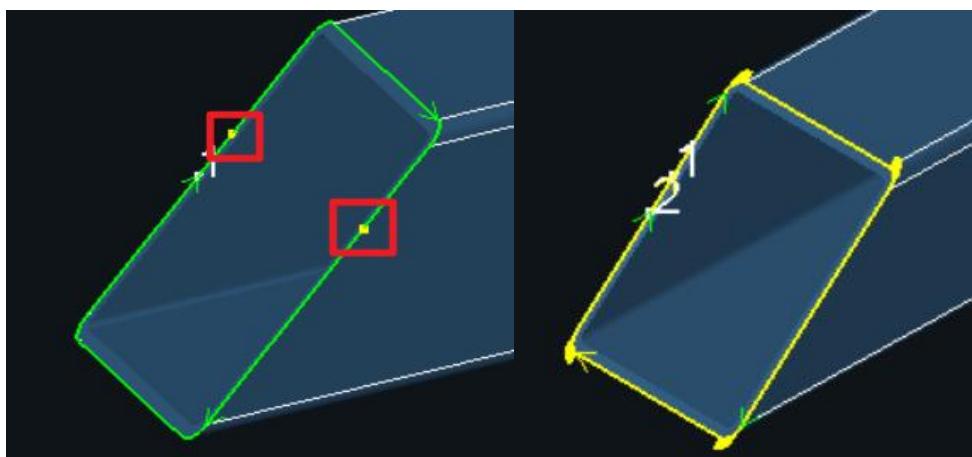
2.7.3 Set SawOff

After setting the SawOff line, it can be matched with TubePro and sawing machine to process the tubes with reinforcing ribs.



After importing the drawing, apply Break Curve on Sawing Line in cut-off line, and click Set SawOff option in the Optimize drop-down menu; Click three base points on the SawOff section in turn to determine the sawing angle for the sawing machine.

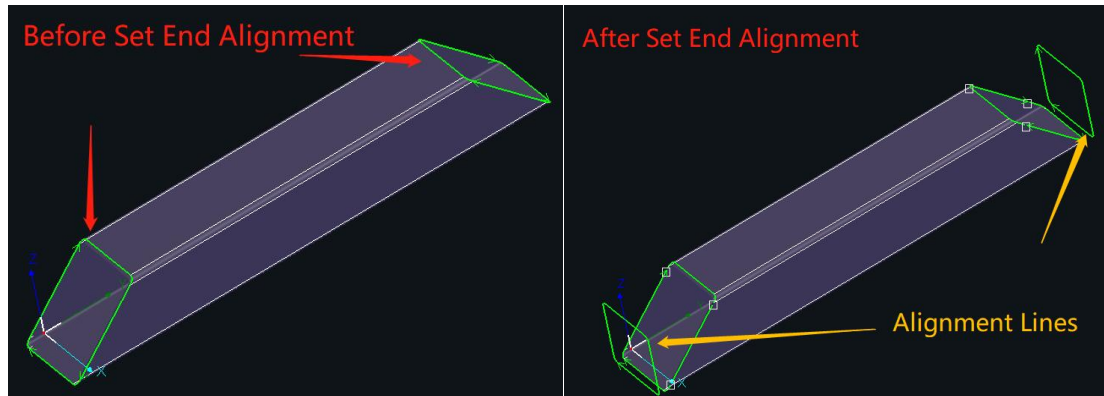
The first two points will be marked with yellow dots, and a SawOff path in yellow will be generated after the three points are set, and the green line represents the original curve. Press the ESC key on the keyboard to exit Set SawOff.



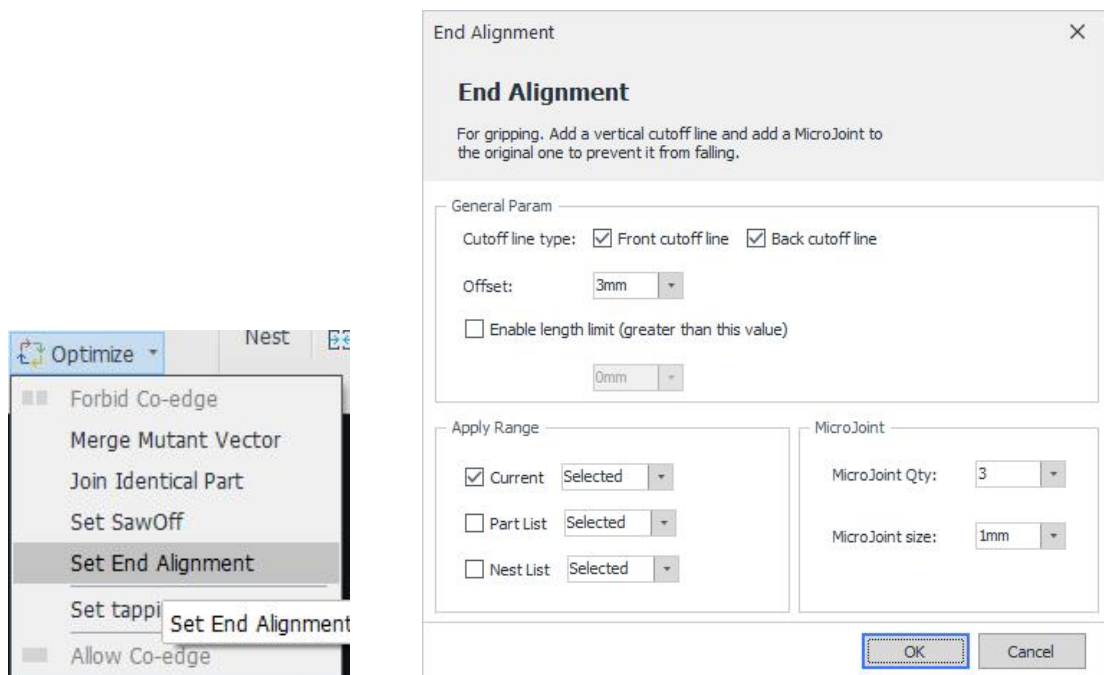


2.7.4 Set End Alignment

For parts with bevel cut-off section on both sides, you can apply Set End Alignment to fill the section cutting lines of parts, and MicroJoint will be added to cutting lines, which is convenient for clamping by the jaws.



Select the part with bevel cut-off section and click Set End Alignment in the Optimize drop-down menu (the whole part must be selected, not just the bevel cut-off line). Select the type of cut-off line corresponding to the part to be aligned, and set the offset distance between the aligned cut-off line and the original one.



When Enable length limit (greater than this value) is set, a single part whose length is greater than this value is allowed to add an aligned cut-off line, while a



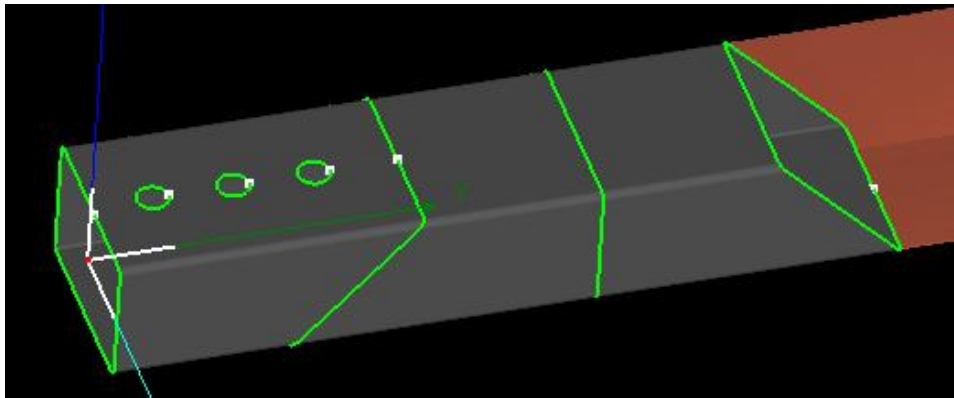
single part whose length is less than this value will not. It is possible to add MicroJoint to the bevel cut-off line so that it can be held by the claws.

2.8 Nest

2.8.1 Co-edge

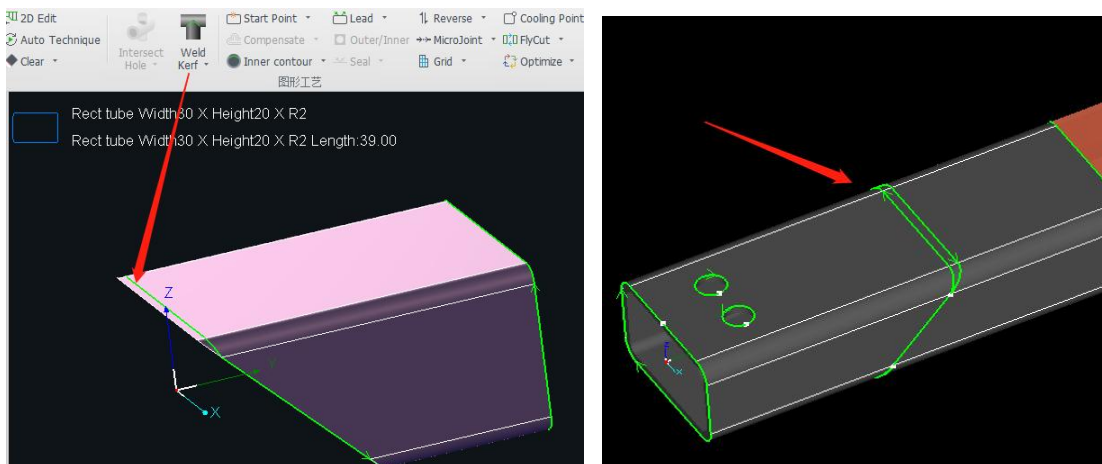
2.8.1.1 Identical Match

Identical match is always used to create co-edged line between two parts with shared cut-off sections, as shown in the picture below.



2.8.1.2 Weld Kerf Offset

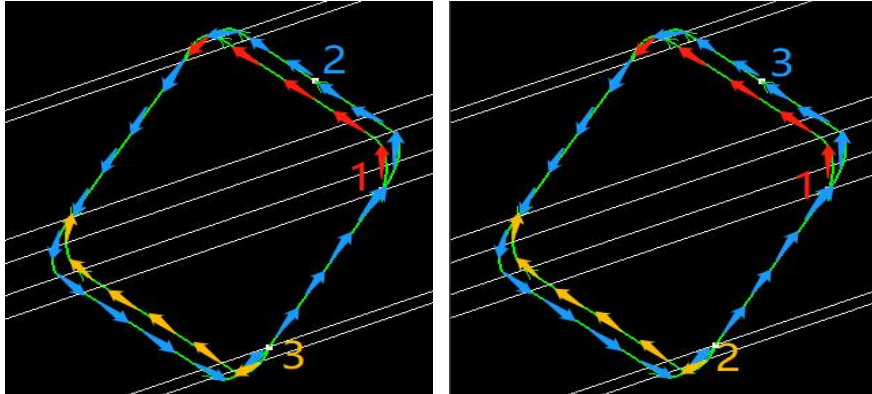
Weld Kerf Offset is used to create co-edged line for the parts applied weld kerf offset as below.



By simulated co-edged cutting path, you can see the cutting contains 3 segments

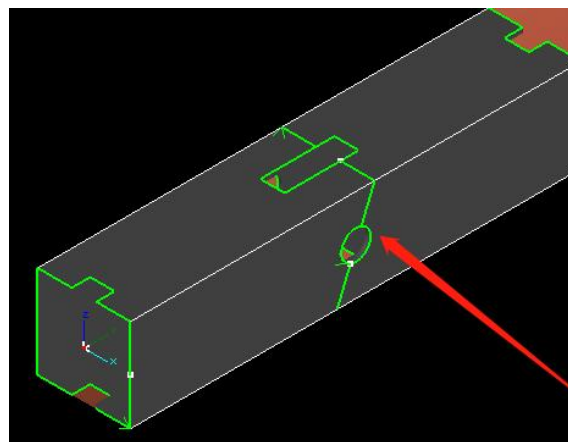


marked by red, yellow and blue respectively in below figure. The default cutting sequence is red →blue →yellow(as shown on left), while if Cut-Off Last is enabled, the cutting sequence will be red →yellow → blue(as shown on right).




2.8.1.3 Island Type


If there is a waste zone on co-edged cutting line between two parts, it's called Island type in TubesT.



Same to Weld Kerf Offset, there are different approaches to create Island Type co-edged line: Continuous Path, 3-Segment Path and Single Part Path.

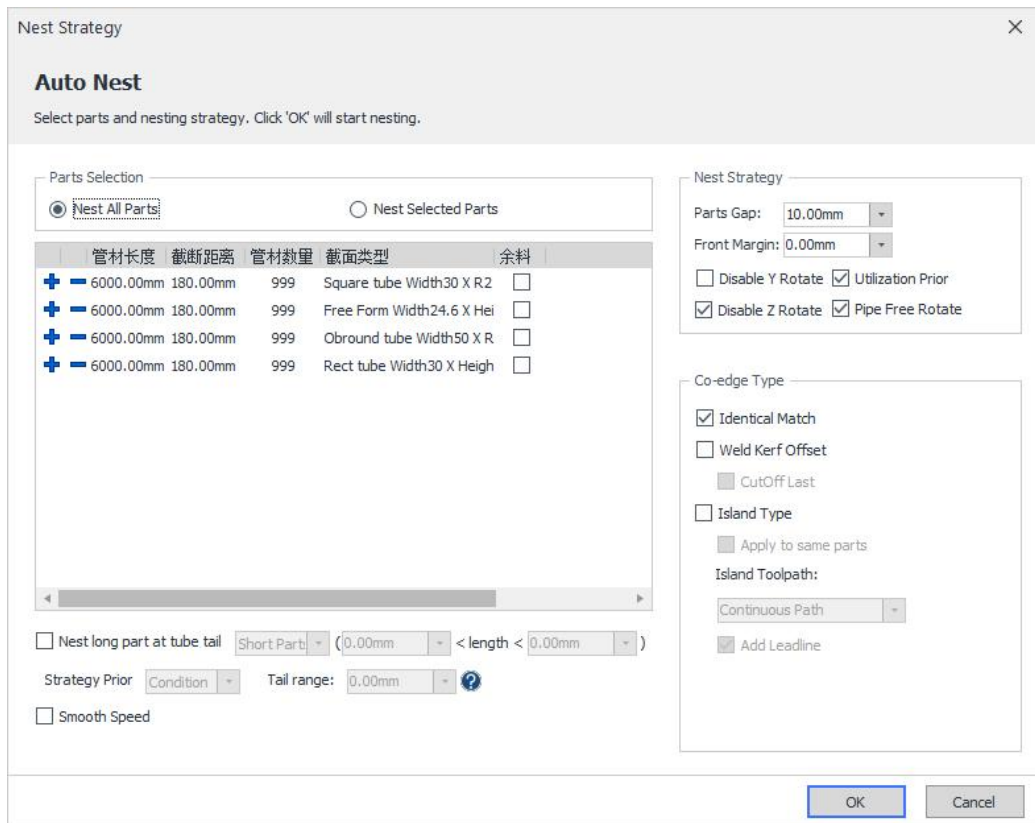
The difference on the cutting path can be viewed by  when -the nesting is finished.

2.8.2 Auto Nest

After the modification of parts amount and design is finished, click  to



start Auto Nest.

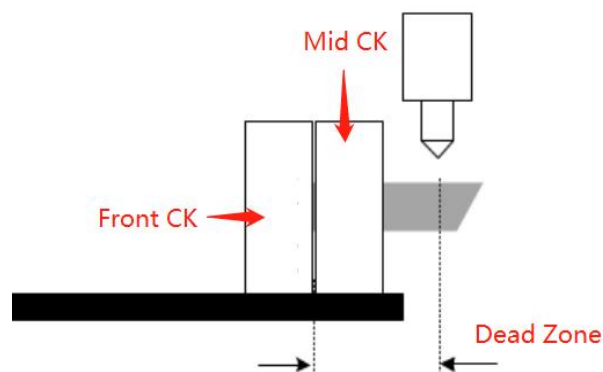


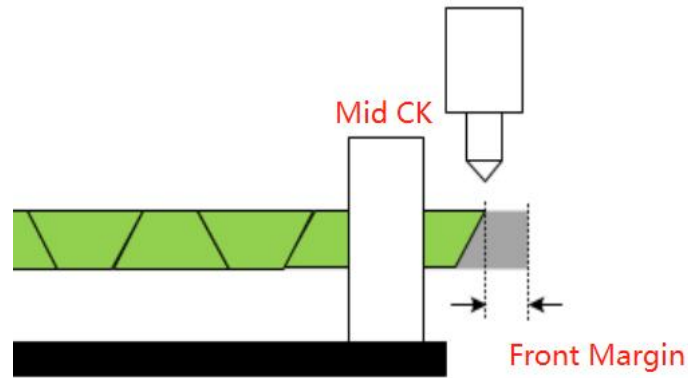
Parts Selection: choose to nest all parts in list or just selected ones.

Tube length: the total length of the tube.

Dead Zone: when feeding chuck reaches to its limit position towards fixed chuck, the tube body from chuck jaws to laser head which can not be machined is called Dead Zone.

Front Margin: the distance from the first part the tube end face.





The tube length available for nesting is decided by Tube Total Length, Dead Zone and Front Margin.

Remnant: If there are some remnant tubes existing, and you want to nest the parts on the remnant tubes in the first priority, you can enable Remnant option. For example, in the following figure, the software will automatically arrange 3 pieces of 1300 remnant tubes first, and then arrange the whole pipe of 6 meters.

	管材长度	截断距离	管材数量	截面类型	余料
+ -	6000.00mm	180.00mm	999	Square tube Width30 X R2	<input checked="" type="checkbox"/>
+ -	6000.00mm	180.00mm	999	Free Form Width24.6 X Hei	<input type="checkbox"/>

Parts Gap: it is the minimal distance between two parts when Co-edge is not enabled.

Disable Y Rotate: parts cannot rotate around Y direction in nesting procedure.

Disable Z Rotate: parts cannot rotate around Z direction in nesting procedure.

Pipe Free Rotate (Round Type): it allows round parts to freely rotate in nesting to minimize the waste.

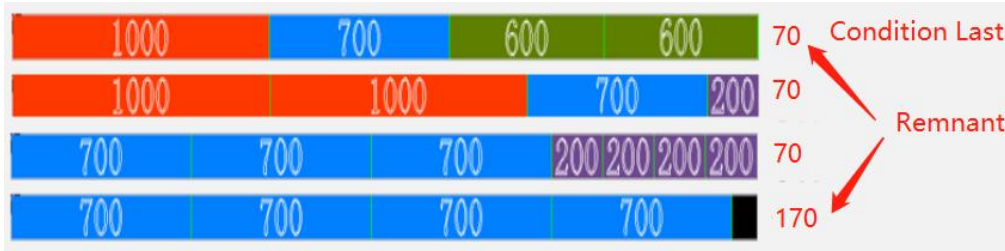
Nest Long Parts on Tube Tail: To realize short remnant or zero remnant tail like Dodge or Pull functions, it is necessary for you to nest the long or short parts on the last place in nesting result.

Take long parts as an example: as long as a length is set, it is determined how long the parts will be regarded as long parts by TubesT. After Auto Nest is finished,

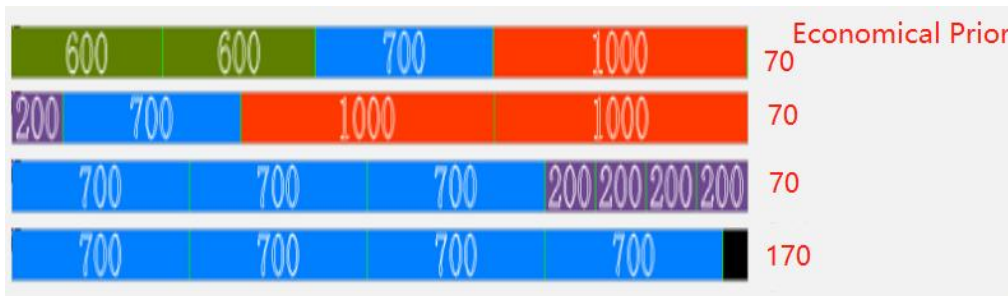


TubesT will automatically arrange the long parts to the last.

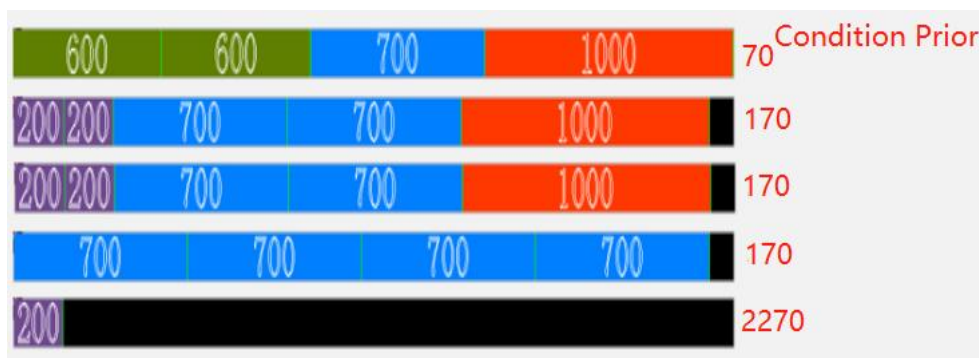
Assuming that there is such a nesting result:



Under the premise of Utilization Prior, only two tubes can meet the conditions when Nest Long Parts at Tube Tail is enabled:



If you select Condition as the Strategy Prior and want to save the materials as possible as you can, there will be three tubes which can meet the conditions but more materials used:




Smooth Speed: When you're nesting massive parts, it is possible to drag the dot on the bar to adjust the ratio of Better Utilization and Nesting Speed.

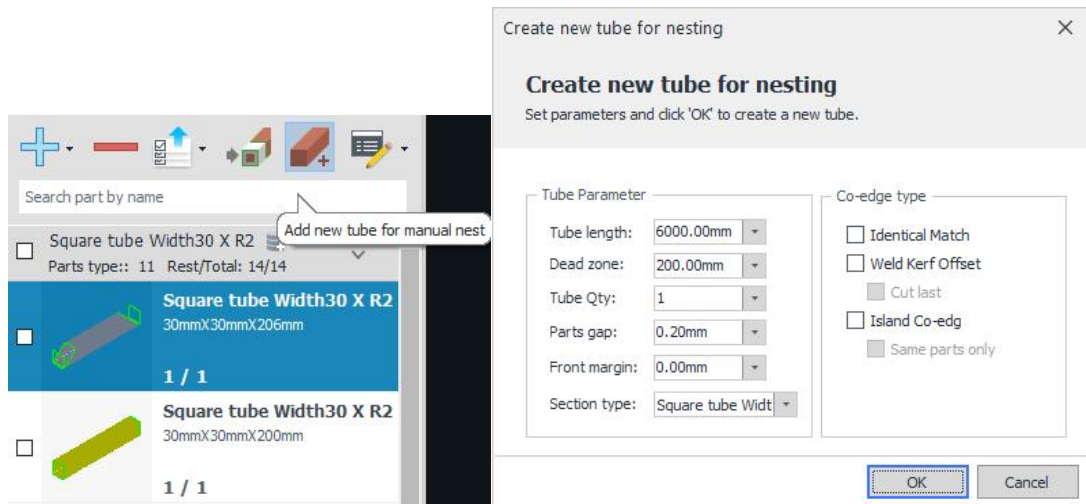




2.8.3 Manual Nest


It is allowed to manually nest parts to tube for special needs.

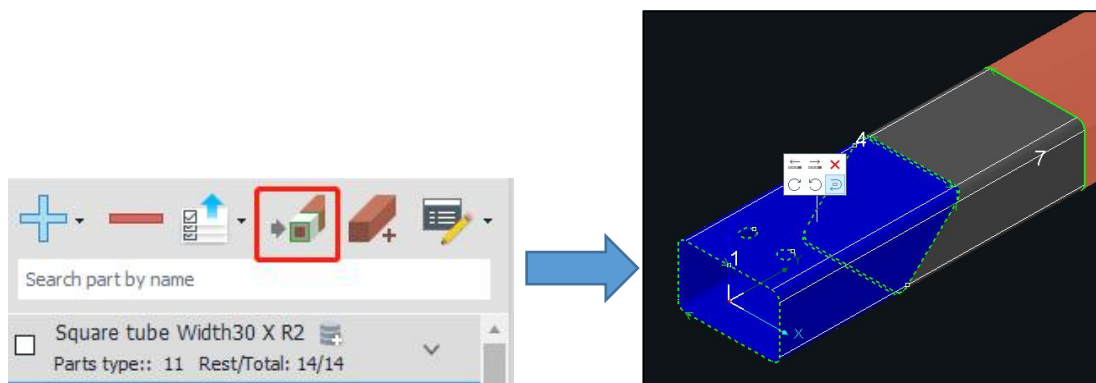
Click  above the parts list to create a new tube for manual nest.



Choose Section Type first.

Tube Parameters and Co-edge Type are definitely same to those in Auto Nest, which is allowed to be modified as needed.

Click OK to create a new available tube, and then select the parts and click  to nest them on the very tube.



When nested, the parts can also be re-placed manually (available for Auto Nest or Manual Nest). Click nested part will call out quick access box which provides options to move part forward/backward, remove part, rotate CCW/CW, flip selected part.



2.9 Sort

Various Sort Strategies are provided in TubesT, you can apply any one of them in the drop down menu of Sort.

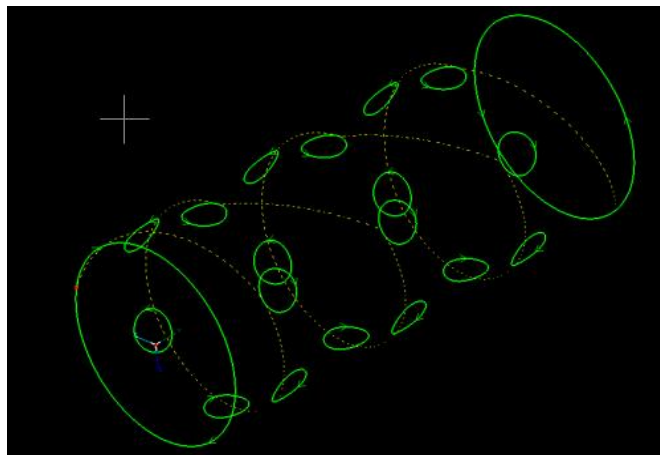
2.9.1 Auto Sort

2.9.1.1 Sort along Y Axis

Y axis is the tube extrusion direction, the sorting principle is: start from tube front to tail.

If two graphics start in same Y coordinate, then it will sort it in CW direction.

The sorting result will be spiral along the tube from the front to the end. A typical sorting result is shown in below figure.



2.9.1.2 Sort by Face

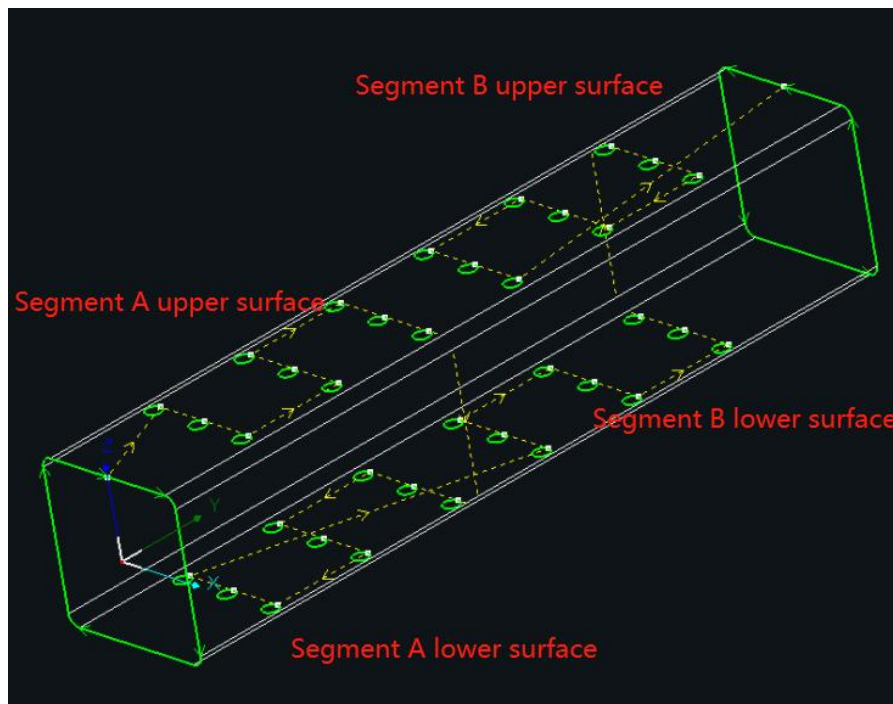
Sort by Face is suitable for Rectangular, Square and Special Circular tubes (if the surface with a fixed rotation angle is considered as a face). Under this sorting method, the graphics on one face will be sorted along Y axis first, which can realize that the parts on different faces will be machined one face by one face.

Length is needed when you're using Sort by Face, when it is set, TubesT will separate the tube into several segments with designated Length and then sort the graphics on each one respectively, but not sort them on a whole face.



The Travel Path will be optimized when Sort by Face is applied to short the unnecessary path and make it similar with a S shape.

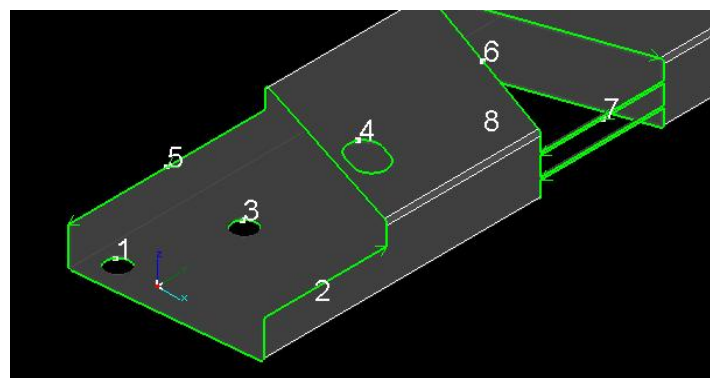
For example, the square tube is divided into Segment A and B by Length in Sort by Face. When the upper face of Segment A is finished, its bottom face will be the next; to shorten the Travel Path, the next will be the bottom face, and then upper face of Segment B respectively, whose whole trajectory is like a S shape.



If so, the path is optimized by reducing the rotation between segments.

2.9.1.3 Wrap Cut-off Section First

The holes (1, 2, 3, 4) will be cut before cut-off section (5), this can avoid the situation where the waste in cut-off zone (5) may tilt up and hit the laser head.





2.9.1.4 Rotation B Axis Rotation

If the servo motor of B axis uses Absolute Encoder, and the feedback value of encoder will overflow to a certain extent if B axis rotates in the same direction all the time, resulting in software alarm and cutting suspension.

Therefore, it is recommended to check this option if the B axis uses an absolute value encoder.

The result is, ① the cutting direction alternates clockwise and counterclockwise in turn; ② the travel direction of the holes on the tube face alternates clockwise and counterclockwise in turn, which means B Axis Rotation will be controlled as possible as it can.

2.9.1.5 Sort Change Cut-off Start

If this function is checked, the software is allowed to automatically adjust the Start Point of the cut-off line, and in some cases, a shorter Travel Path can be obtained.

2.9.1.6 Applied to All Nested

If this function is checked, the Sort command will be applied to all nesting results every time it is enabled.

2.9.2 Manual Sort

2.9.2.1 Common Sort

Also, it is possible to apply Manual Sort when the strategies of Auto Sort failed to meet your demands.


When enabled, TubesT will turn off Render Effect of parts and then get them gray, then you can sort the graphics by mouse clicking. To be distinct, the ones sorted will be turn green. When Manual Sort is finished, right click the mouse and select Finish sort to apply current sorting result.



2.9.2.2 Unfold 3D Sort

Unfold 3D Sort, an extension of Manual Sort, is always used on the parts which need special sorting. When enabled, you can sort the graphics in 2D plane by expanding the 3D tubes into a 2D one to avoid the visual noise.

2.9.2.3 Sort by Simulate Button

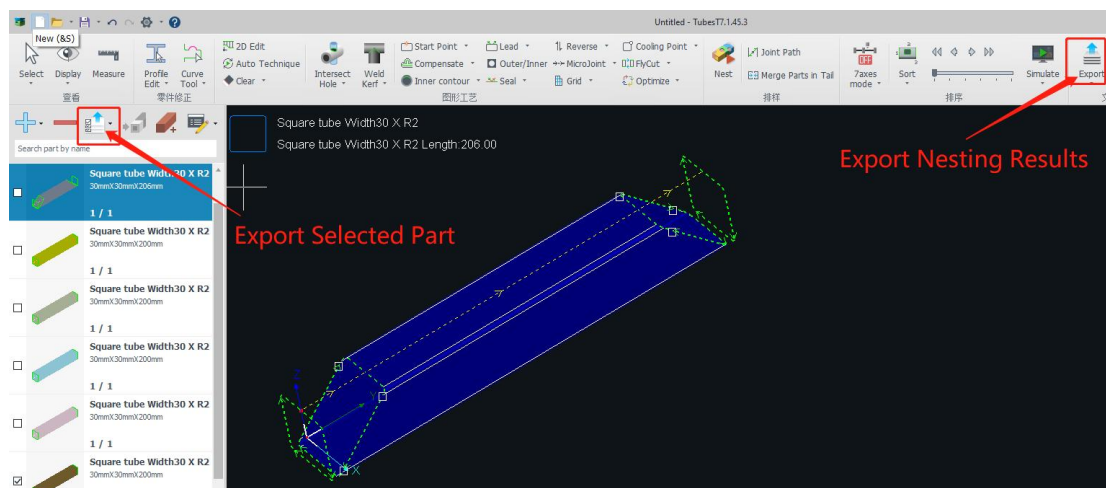
Select one graphic and click buttons from  on top menu bar to move the cutting sequence one step forward or backward, set as first one or last one.

2.10 Export

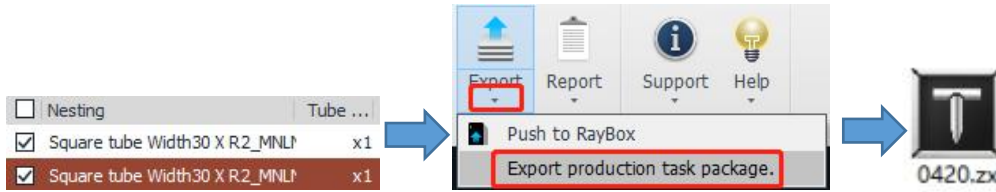
It is possible to export the parts and nesting results in TubesT, which is *.ZX file for TubePro, *.CTDS file for CypTube respectively.

2.10.1 Export Nesting Results

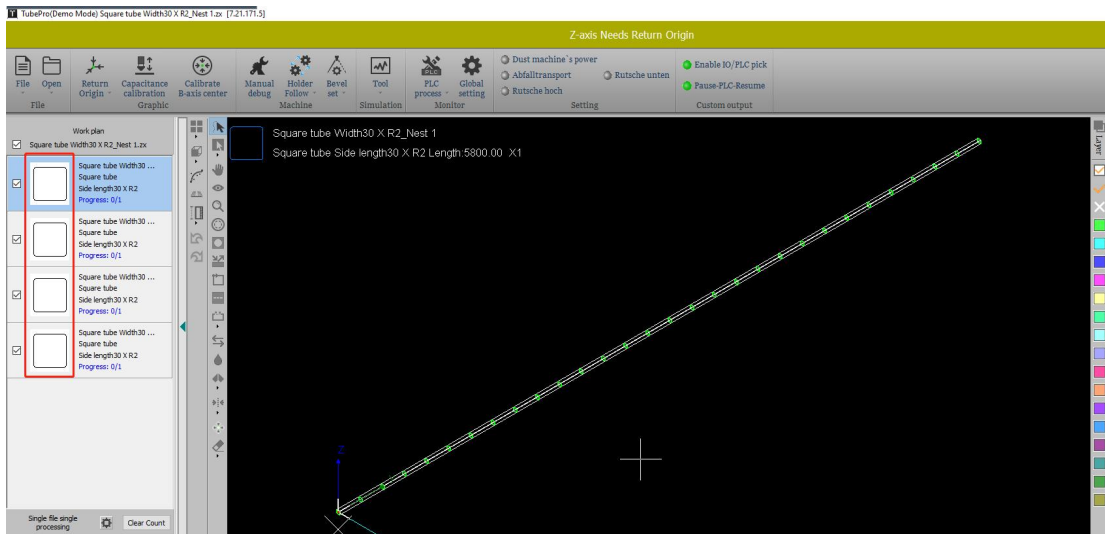
When nesting is finished, click Export on the upper left to export all nesting results in *.ZX format to TubePro, also in *.CTDS for CypTube (profile parts not supported).



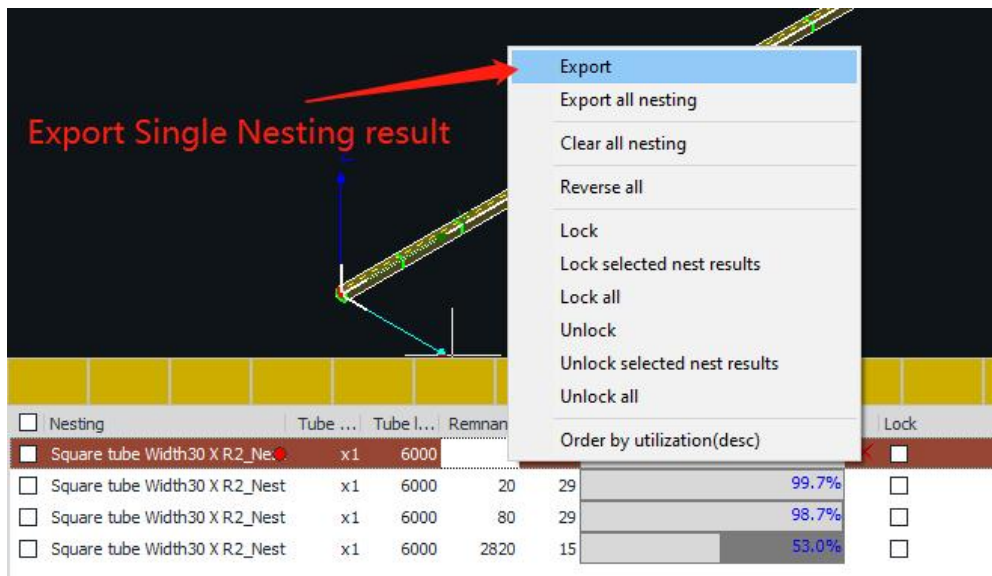
Click Export Production Task Package in the Export drop-down menu to save all the nesting results into a machining file.

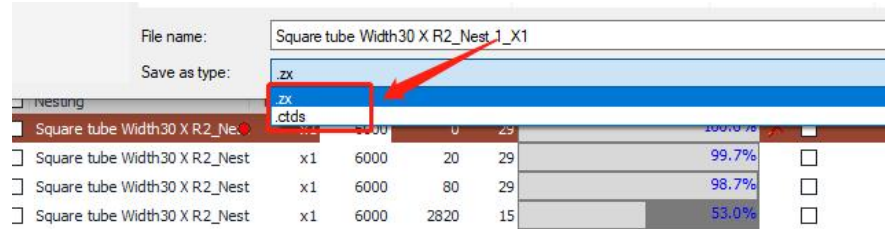


When opened in TubePro, it will automatically enter the Work Plan interface, display the Progress of each nesting result, and automatically switch to the next nesting result after processing to a predetermined number of times.



Otherwise, you can select from nesting list and export single nesting as *.ZX or *.CTDS file for cutting.

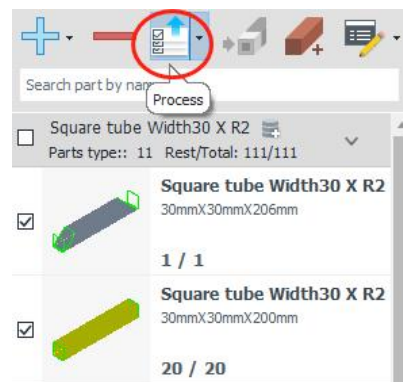




If the cutting file cannot be read by TubePro or CypTube, it may be caused by version compatibility between TubesT and cutting software TubePro or CypTube. It can be solved by Update TubePro or CypTube to latest version.

2.10.2 Export Part File

If you only want to export cutting file of a single part, just select the parts and right click the mouse, then choose Export .



2.11 Report

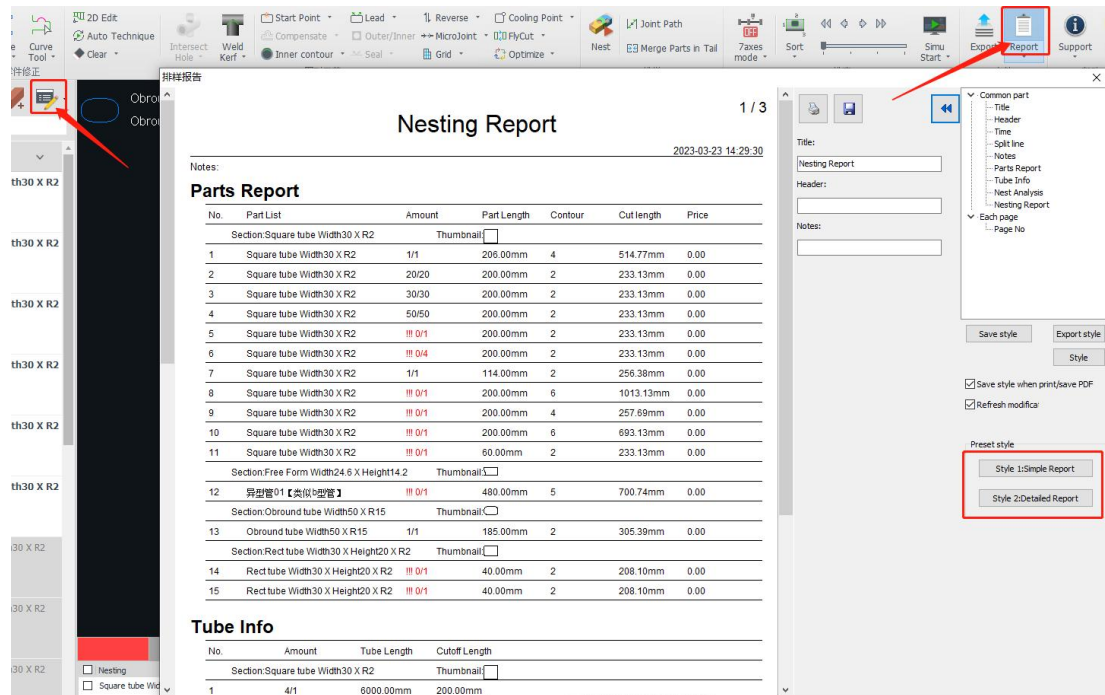
After nesting, you can view the part details in each nesting result in Report. TubesT has Preset Report and Custom Report. The built-in style of Preset Report can choose Simple Report and Detailed Report, while the Custom Report can change the report content and display style in it under the user template according to the your needs.

2.11.1 Preset Report

There are two styles in the nesting report: Simple Report and Detailed Report. Parts Report, Tube Info, Nest Analysis and Nesting Report can be viewed in the Detailed Report; while Part Report, Tube Info and Nesting Report can be viewed in

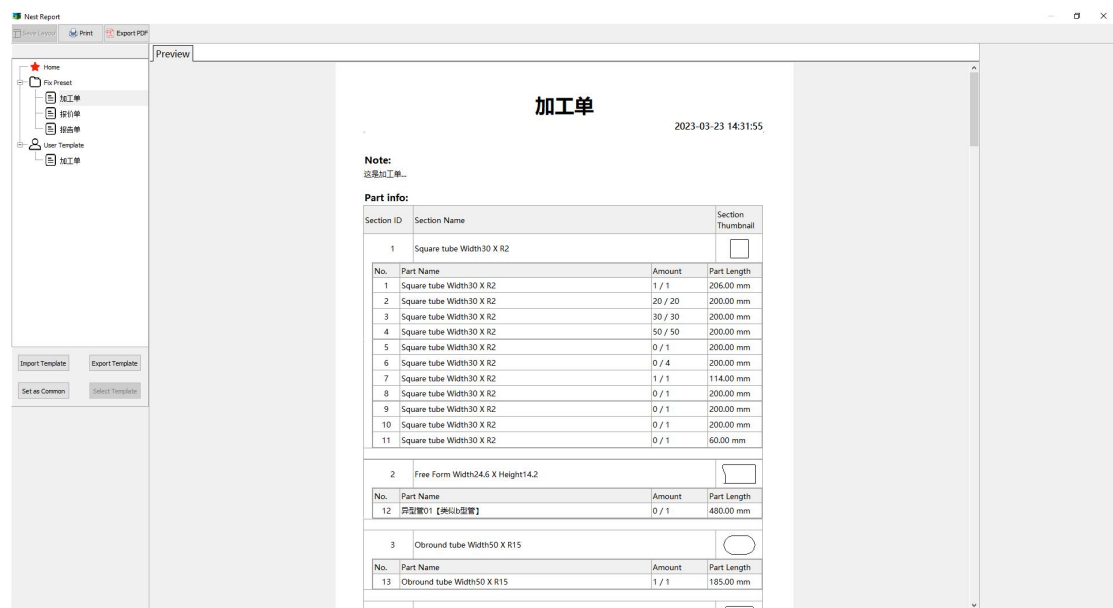


the Simple Report, in which the part price can be set in the Batch Edit and displayed in the report.



2.11.2 Custom Report

Custom Report is added newly in the drop down menu of Report. Some fixed Preset and User Template are available in Custom Report, including Work Report, Price Report and Report.



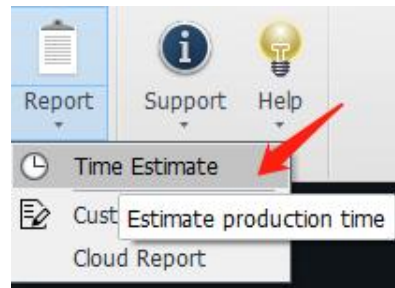


Reports in User Template can be deleted, and information in it can be added or subtracted as needed in Report Layout.

For more details please refer to the contents on our official website.

2.11.3 Time Estimate

Time Estimate can be used to estimate and count the processing time of nesting results. When the nesting is finished, click Time Estimate in the drop down menu of Report. Then you can input the cutting speed as needed, click Start to get the result, and please note that it is necessary to install TubePro in current computer. Click Export Excel to export the results of Time Estimate to a table.



Estimate Production Time

Estimate Production Time
It is a rough time estimation. Time delay, axis speed error and other exceptions are not included

Nesting	Qty.	Cut speed (M/...	Time per tube	Total time
<input checked="" type="checkbox"/> Square tube Width30 X R2_Nest 1	1	0.6		
<input checked="" type="checkbox"/> Square tube Width30 X R2_Nest 2	1	0.6		
<input checked="" type="checkbox"/> Square tube Width30 X R2_Nest 3	1	0.6		
<input checked="" type="checkbox"/> Square tube Width30 X R2_Nest 4	1	0.6		
<input checked="" type="checkbox"/> Obround tube Width50 X R15_Nest 1	1	0.6		

Total time of selected tube: 0s

0 %

More settings

Default cut speed: 0.6 Speed unit: M/min Time ratio: 1

When using Time Estimate, you need to install a TubePro (7.1.15.135_20200902 or later) on the computer where TubesT is installed; To ensure the accuracy of the prediction, please ensure that the parameters (Travel Speed, Acceleration, etc.) of this TubePro are the same as those of the TubePro on the machine.





III. Auxiliary Function



3.1 Other Functions

3.1.1 Simulate

You're allowed to simulate the cutting process to check cutting sequence and direction of nesting results. Dynamic and Static Simulate are included in TubesT.

When in Dynamic Simulate, click Simulate to start simulating the parts or nesting results in current window if no parts are selected, but if you do, it will simulate the cutting effect of the selected parts only, in which you can click

 to adjust the Simulate Speed and  to change current process.

In Static mode, click  to change sequence of selected graphics, and slide  to view the cutting sequence.

For I-steel and free-formed tubes, you can check the interference detection by Simulate, and the 3D Vector of the parts can be modified automatically when specific cutting head type is selected.

Click Path Check on the Simulate drop-down menu to check the path of the nesting results to find whether there are dangerous and unreasonable cutting path. After turning off the Path Check window, you can open it again by clicking Open Check Result Window.

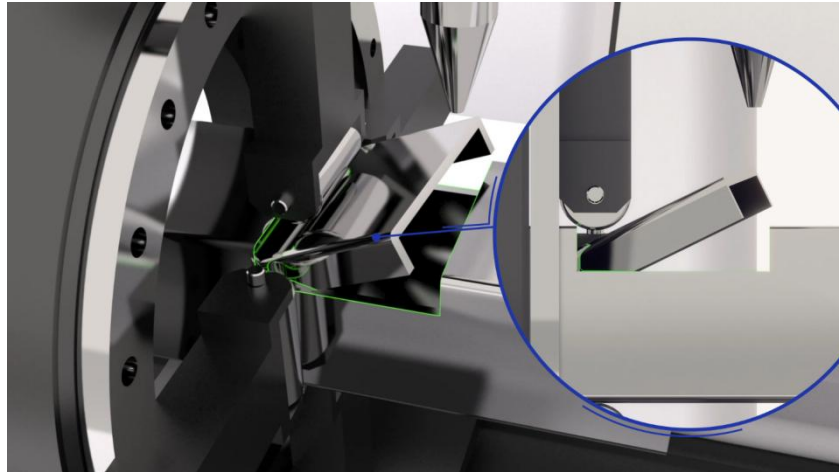
3.1.2 Path Check

3.1.2.1 Rolling Shaft Safety Check

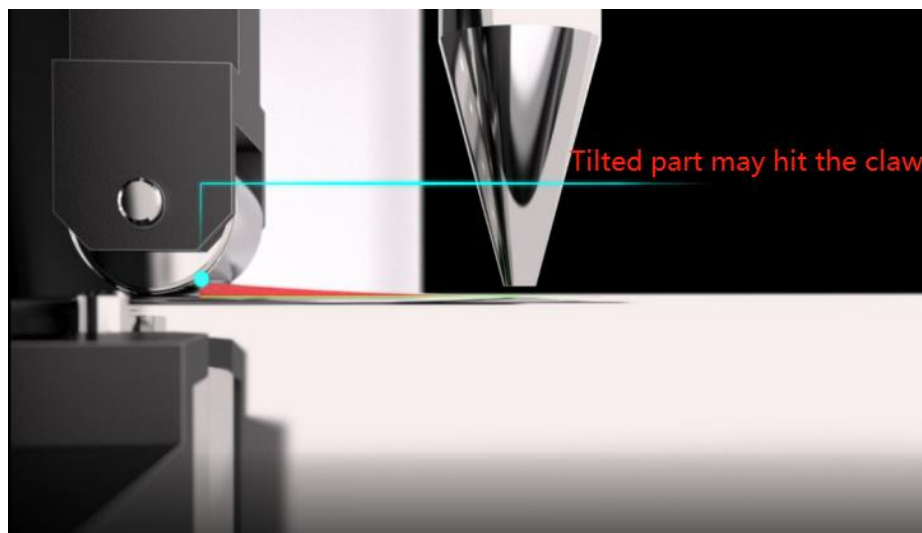
The jaw of the chuck in the tube cutting machine is generally a roller, once the waste material is tilted or the sunken part of the tube body is pulled into the roller, it will affect the normal Y-axis movement and even cause damage to the



machine. Therefore, when Rolling Shaft Safety Check is enabled in User Habit Settings, it will automatically check the cutting path according to preset parameters, and give tips and modification suggestions.



For machining some tubes with long holes on it, when the main chuck pulls the tube back, if the backward distance is long, the cut area will be pulled into the jaw of the middle chuck, to avoid this, you can enable Rolling Shaft Safety Check, then it will automatically prompt the risk according to the preset distance from the cutting head to the middle chuck and give suggestions on this.

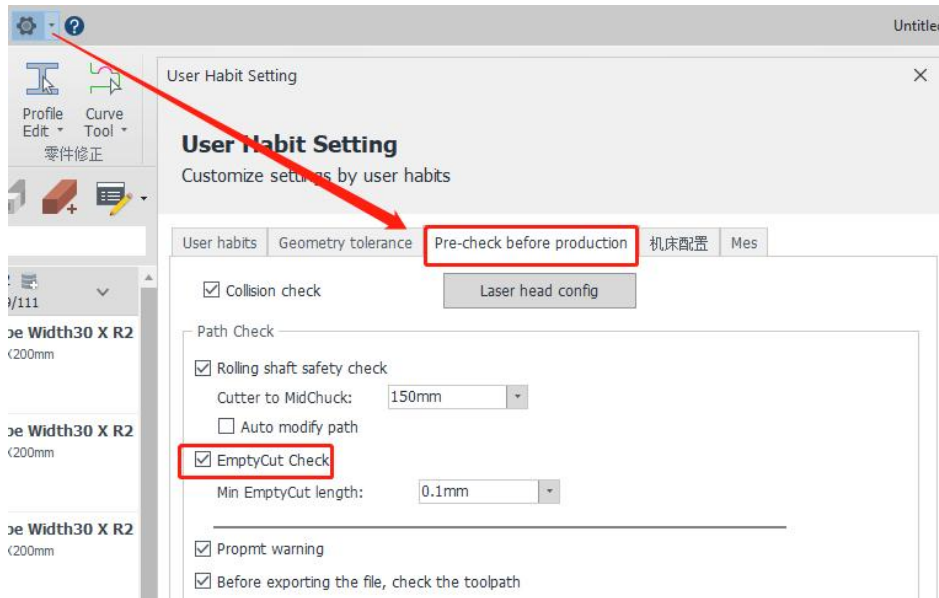


3.1.2.2 EmptyCut Check

Click the icon on the upper left to open User Habit Settings, you can enable EmptyCut Check in Pre-check Before Production. When there is a disorder as “cut off



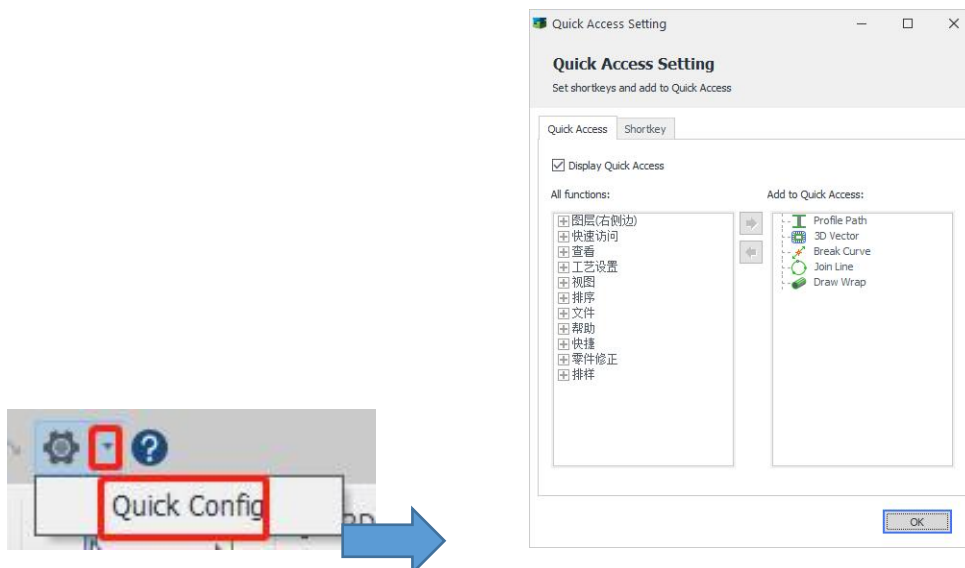
the part first, then cut the hole on the part”, TubesT will prompt the alarm “Empty Cut Path”. If so, it is recommended to adjust the cutting sequence, by Auto Sort or Manual Sort introduced above.



When the EmptyCut distance is less than the set “Min EmptyCut Length”, this alarm will not be prompted. If there is a very short EmptyCut distance which won't effect actual machining, you can adjust this allowance to get EmprtCut Check passed.

3.1.3 Custom Shortcut Keys

In the User Habit Setting drop-down menu, you can customize Quick Access Bar and Shortcut Keys.






3.1.4 Save *.YXY File Automatically When Export Files

*.YXY is sketch file created by TubesT which cannot be used for cutting software.


Format	File	Application	Source
*.IGS	Parts drawing	Nest in TubesT, TubesT-Lite ; CypTube cutting file	CAD software like SolidWorks
*.SAT		Nest in TubesT, TubesT-Lite	
*.JHB		Nest in TubesT	
*.ZX	Cutting file	TubePro cutting file	Export by TubesT/TubesT-Lite
*.ZZX			
*.CTD		CypTube cutting file	
*.CTDS			
*.YXY	Sketch File	Save the parts and nesting results for later editing in TubesT again.	Save by TubesT

You can open *.YXY file by  in TubesT, then TubesT will restore the editing status when it's saved including the parts nested, techniques applied and nesting results etc.

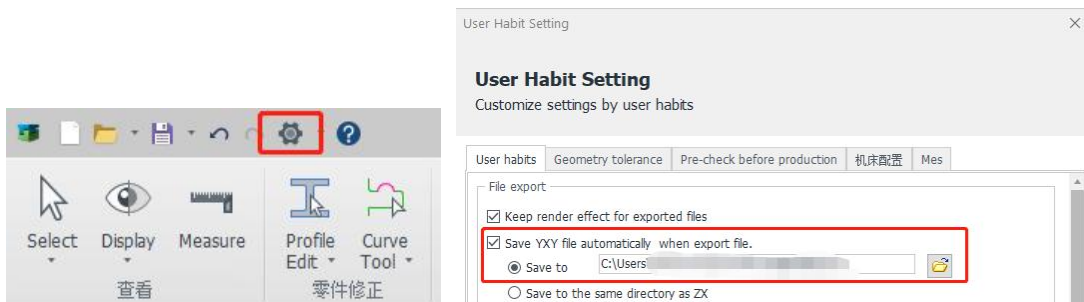
Therefore, it is recommended to save a *.YXY file when exporting *.ZX or *.CTDS file, it allows user to modify the nesting or technique directly when the



cutting in machine can not meet the demands, otherwise you have to do the nesting and technique settings again.

Click  at top left in TubesT to save the file as *.YXY file.

If Save *.YXY file Automatically When Export File is enabled in “User habit setting” window, TubesT will save a *.YXY file automatically to designated directory while the cutting file is exported.



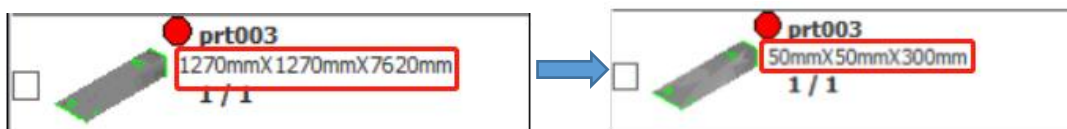
3.1.5 IGS Length Unit

There is an option named IGS Length Unit in User Habit Settings, Read Length Unit and Ignore Length Unit available.



The default setting is “Read Length Unit” which means TubesT will read parts in same length with actual drawings. If you draw the square part as 50-inch but what you really need is 50 mm, you can select 'Ignore Length Unit', by which TubesT will only read the value “50” and add default unit “mm” for it, then its dimension will be 50 mm.

The following figure shows the Import Result of 50-inch square tubes from the setting of Read Length Unit and Ignore Length Unit respectively.





3.1.6 Differentiate Part Thickness

For parts with the same diameter but different thicknesses, they are placed in the same section group by default, also they can be nested on same tube. If you need to differentiate this kind of tubes, it is necessary for you to enable Differentiate Part Thickness.

When this option is enabled, and then the imported parts will have thickness attributes, so the parts with the same diameter but different thicknesses will be placed in different section groups and nested separately.




This function needs to be enabled before importing parts, or it will turn gray.

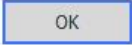
3.2 Part Sections Process

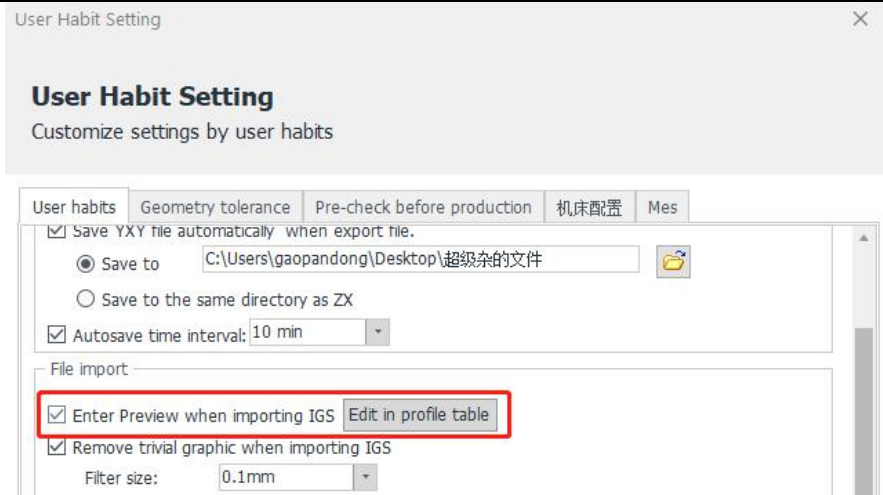
3.2.1 Parts Tolerance

TubesT is capable to classify the parts of profile dimension within given tolerance into same group. For example, the round parts of radius as 38mm and 38.2mm can be classified in same group and nested on same tube.


To enable tolerance when importing parts:

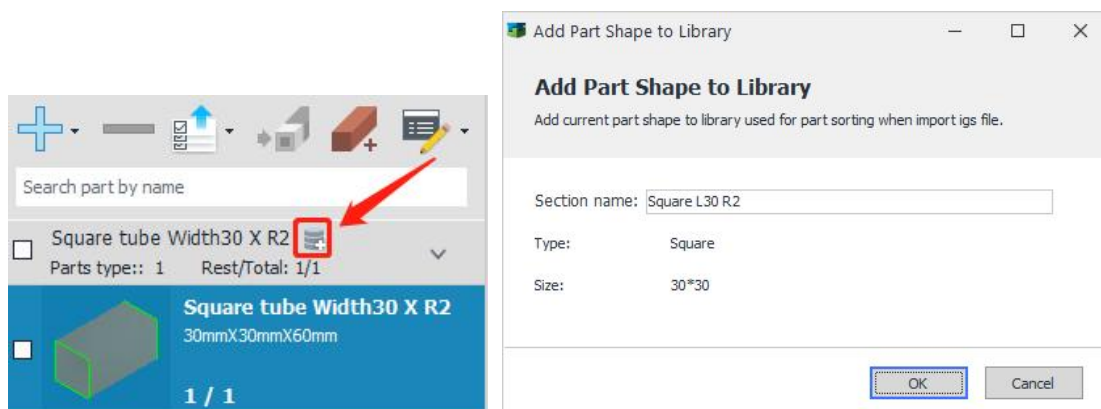
- (1). Click  to open “User Habit Setting” window, enable Enter Preview

When Importing IGS in User Habits, and click  to save it and exit.

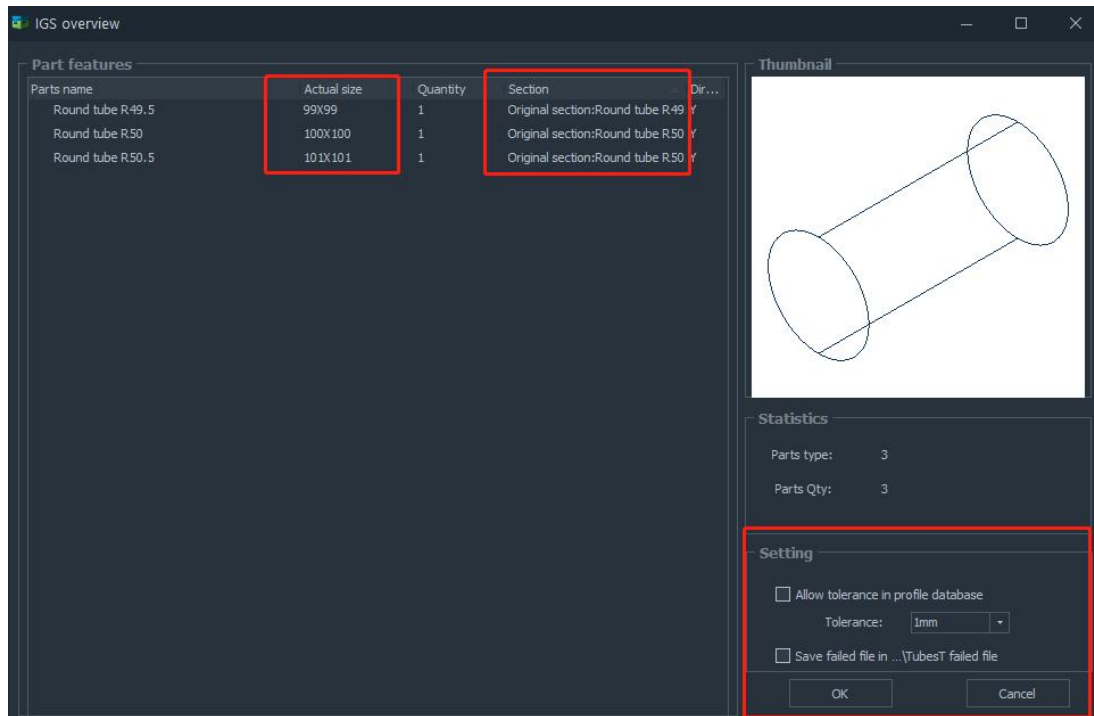


(2). Import IGS file by Import from File as a standard section, as the square tube of 38mm shown below. When finished, there will be an IGS Overview when importing, ignore it and click OK to the next.

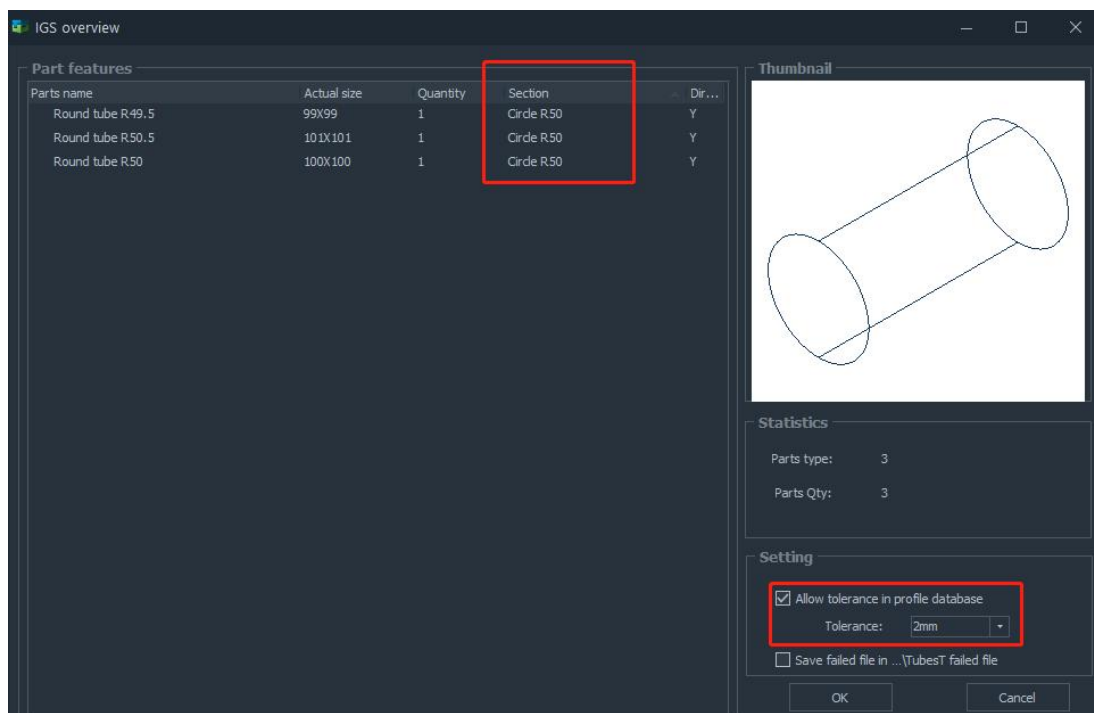
(3). When imported, click  beside the square tube of 50x50 mm, R3 to save it as a standard section to Part Shape Library.



(4). Import the parts with little dimension allowance like the square tubes of 49.5x49.5 mm and 50.5x50.5 mm by Import from File again. During it, you can see the actual dimension of each tubes in IGS Overview, also the section to-be-recognized by TubesT.

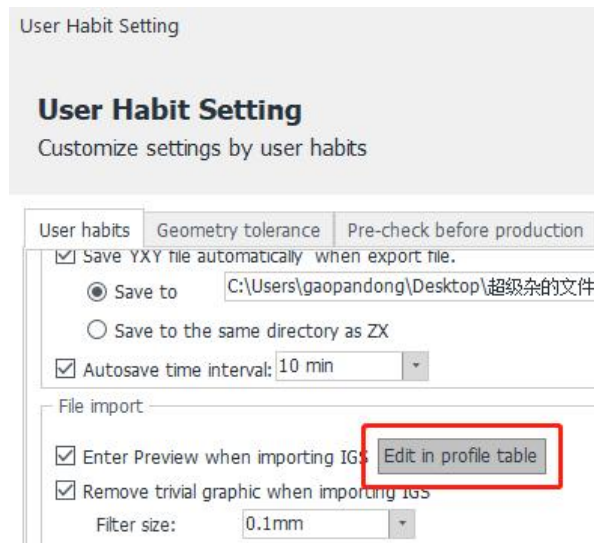



(5). At this time, check Allow Tolerance in Profile Database, and set an appropriate maximum Allowance, such as 2 mm in the example. Then, it can be found that the square tubes of 49.5x49.5 mm and 50.5x50.5 mm can be merged into the standard tube section of 50x50 mm added in Step 3, and all Section Date is Square Tube Width 50 mm, R3.

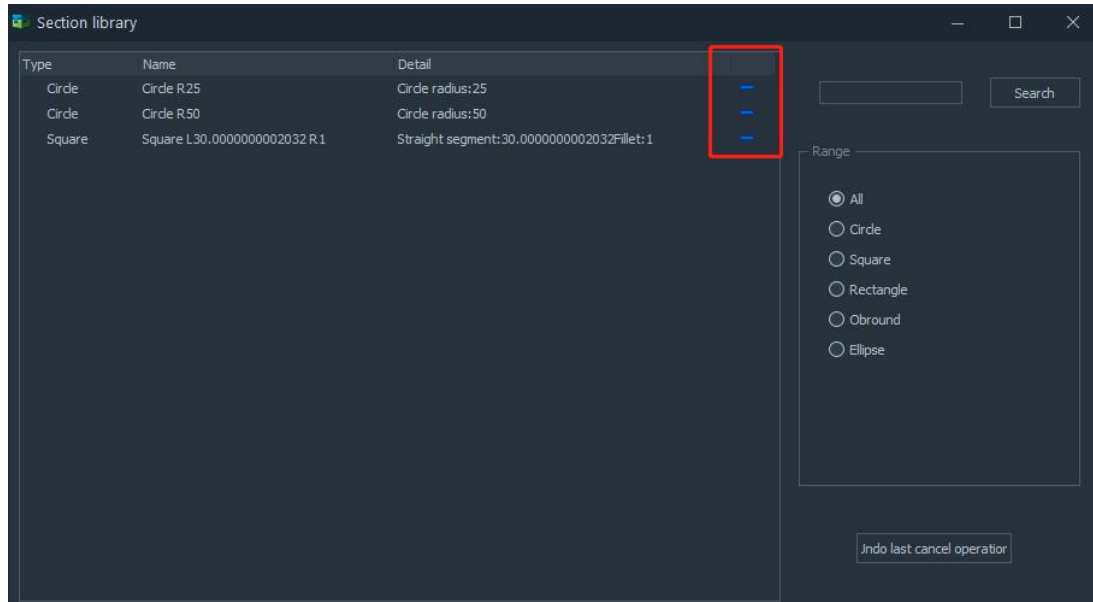




When imported, the square tubes of 50.5 mm and 49.5 mm are classified as the tube of 50 mm, capable to be nested together.



It is possible to check the Standard Sections added in Edit in Profile Table, User Habit Settings. And you can click  to delete selected section added previously.



Currently this function is available for Circular, Square, Rectangular, Obround and Ellipse tubes.

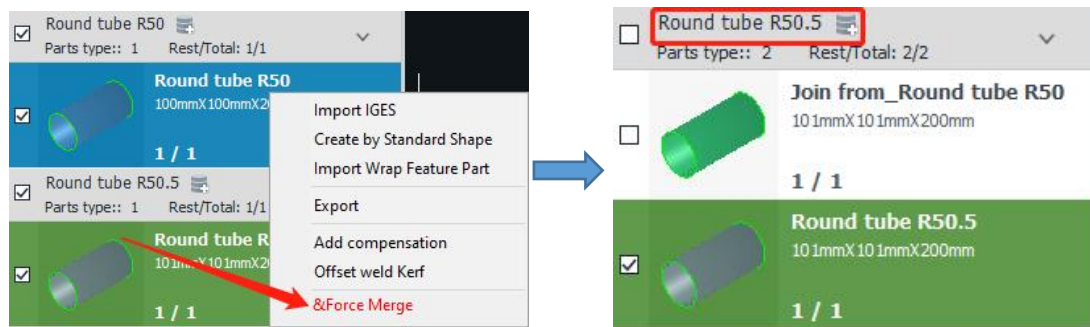
Sometimes the standard profile like circular tube, will be recognized as Free-Form shape due to the drawing error, which can also be solved by this function.



3.2.2 Force Merge

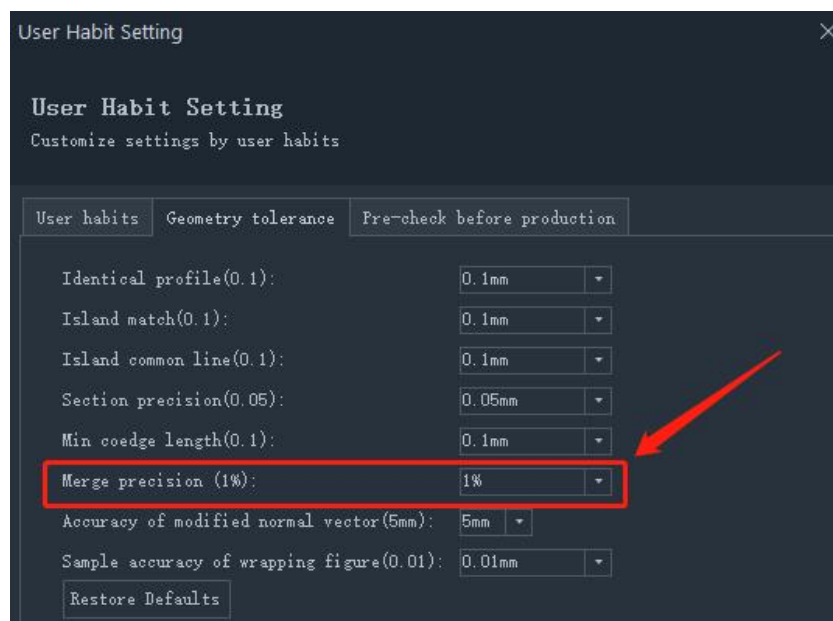
Besides importing parts with tolerance, you can also merge the parts of different dimensions in Part List into same group by Force Merge.

Select parts and right click the mouse, select Force Merge.



Then you can see the square parts selected are merged in same group for nesting.

The tolerance, Merge Precision, can be set in User Habit Settings, by default only the parts within 1% dimension difference can be merged in same group.



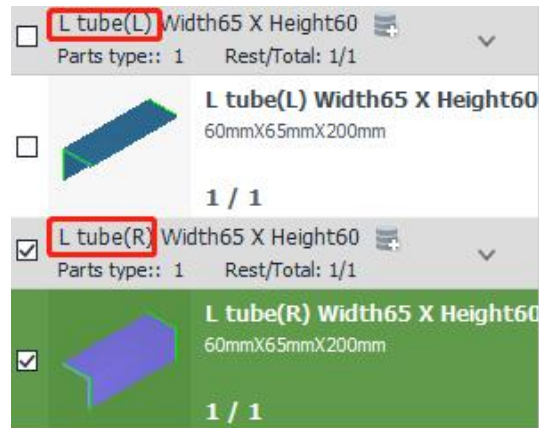
Different with importing parts with tolerance, there is no standard sections in Force Merge, and the parts will be merged into the parts with largest section.

3.2.3 Merge L Profile of Same Size

The angle steels with same dimension may also be recognized as two types, L



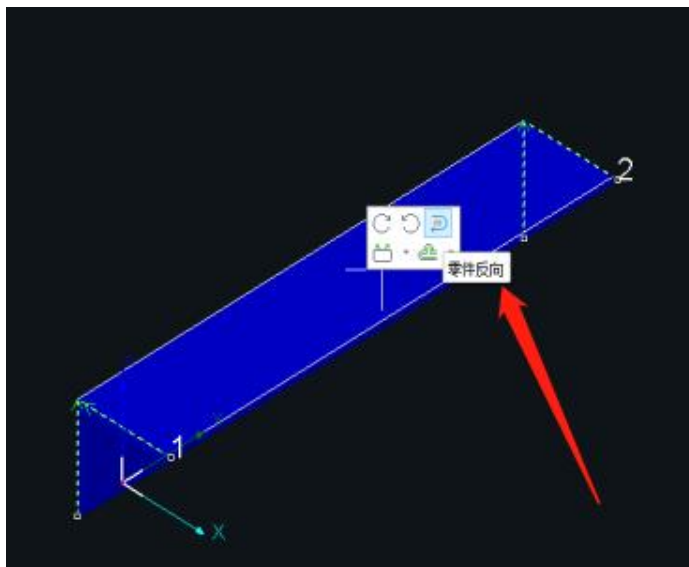
(vertical side on the left) and R (vertical side on the right). Once the angle steel with same size but different types are placed in two section groups, they cannot be nested on the same kind of tubes.



Two methods as shown below to solve this:

(1). Reverse

It is possible to use Reverse to change the L/R type of angle steel. This operation is simple and easy enough, but it can not reverse the parts by batches.

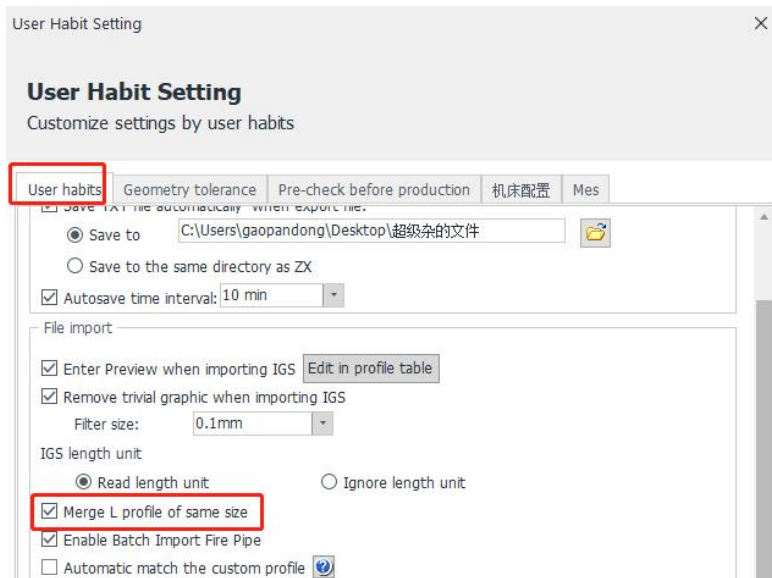


(2). Merge L Profile of Same Size

In User Habit Settings, enable Merge L Profile of Same Size. When finished, the angle steel of the same size imported will be merged into the same section group to be



nested together.

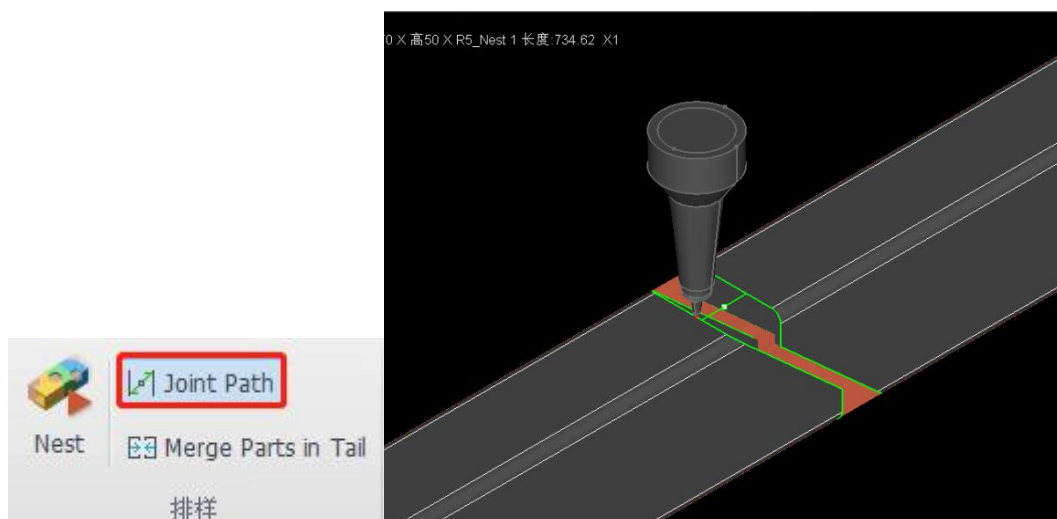


3.3 Machining Skills

3.3.1 Joint Path

If two parts share a piercing point, using the function of Joint Path can reduce the lens loss by piercing and improve the machining efficiency for the machining task in which the cut-off line must have Leadlines.

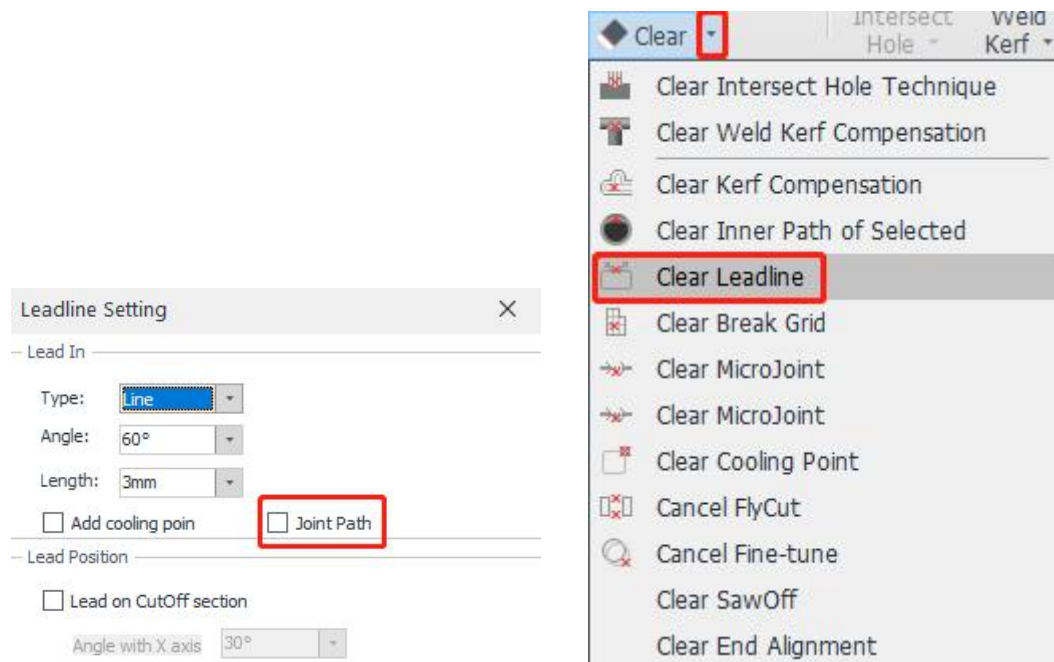
If you're using Manual Joint Path, click Joint Path icon first, and then draw a line between two parts to generate a Joint Path at current position (press ESC or right click the mouse button to exit).





If you're using Auto Joint Path, enable Joint Path when applying leadlines, then click OK to create Joint Path between the parts not co-edged, and you can just set the LeadLine length as normal.

If you need to clear Joint Path, select the cut-off line connected by it, then click Clear Leadline in the drop-down menu of Clear.



3.3.2 Merge Parts in Tail

Some special tube cutting machines featured by 7-axis Pulling and Auto Dodge, capable to realize zero-waste cutting. However, due to the mechanical structure, it is required that the parts arranged at the end of the pipe must be larger than a certain length.

If the parts to-be-nested can not meet the length condition, you can apply Merge Parts in Tail in Nest Tools Bar.

For example, if a machine requires that the parts nested at tube end must be longer than 300 mm, you can set the tail range to 300 mm as shown in the figure, and it can be seen that the parts originally nested at the end of the pipe have changed. Parts within 300 mm are merged, except that the front and end cut-off lines of the merged parts are still retained, the middle cut-off lines (whether they share the same



side or not) will be automatically set in the 10th layer (orange layer), and the inner holes of each part will still be retained as well.

When machining, the inner hole and the front and end cut-off lines of the merged parts will be normally machined, while the cut-off lines in the middle of the merged parts are only marked but not cut as 10th layer is set by marking parameters.

