

StarPIPE software user manual

V5.0

Apply to the WINDOWS operating system

Beijing Starfire Control Technology Co.,Ltd

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Table of Contents

CHAPTER ONE SOFTWARE INSTALLATION GUIDE	4
I. STEPS FOR INSTALLING STARPIPE:.....	4
II. INSTRUCTIONS FOR UNINSTALLING STARPIPE:	6
CHAPTER TWO USER MANUAL OF STARPIPE.....	7
I. INTRODUCTION TO OPERATION INTERFACE.....	7
1. <i>Functions of main toolbar:.....</i>	<i>8</i>
2. <i>Create a new project.....</i>	<i>8</i>
3. <i>Open a project.....</i>	<i>9</i>
II. DESIGN PIPE JUNCTION	10
1. <i>Feature toolbar.....</i>	<i>10</i>
2. <i>Feature ribbon.....</i>	<i>11</i>
3. <i>Modeling instructions for Feature Tool.....</i>	<i>15</i>
III. INSTANCES FOR MODELING OF PIPE JUNCTION FEATURE	20
1. <i>Feature modeling.....</i>	<i>21</i>
2. <i>Machining setting.....</i>	<i>22</i>
3. <i>Simulation machining.....</i>	<i>24</i>
4. <i>Output NC code.....</i>	<i>25</i>
IV. IMPORT CAD PIPE STRUCTURE GRAPH	26
1. <i>Draw structure graph of pipe fitting.....</i>	<i>26</i>
2. <i>Import pipe structure graph.....</i>	<i>27</i>
3. <i>Set pipe fitting parameters</i>	<i>27</i>
4. <i>Set machining parameters</i>	<i>28</i>
5. <i>Simulation machining.....</i>	<i>28</i>
6. <i>Output NC code.....</i>	<i>29</i>
V. INSTANCE OF IMPORTING PIPE STRUCTURE GRAPH	29
1. <i>Import pipe structure graph.....</i>	<i>29</i>
2. <i>Set pipe fitting parameters</i>	<i>29</i>
3. <i>Modify pipe junction</i>	<i>30</i>
4. <i>Simulation machining.....</i>	<i>30</i>
5. <i>Output NC code.....</i>	<i>30</i>
VI. IMPORT DESIGN DRAWING	30
1. <i>Import design drawing</i>	<i>31</i>
2. <i>Set pipe fitting parameters</i>	<i>31</i>
3. <i>Modify pipe junction</i>	<i>32</i>
4. <i>Simulation machining.....</i>	<i>32</i>
5. <i>Output NC code.....</i>	<i>32</i>
VII. VIEW MANIPULATION.....	32
VIII. SET SYSTEM PARAMETERS	33
1. <i>Set parameters of automatic path</i>	<i>33</i>
2. <i>Set parameters of machine tool.....</i>	<i>36</i>
3. <i>Set parameters of code</i>	<i>37</i>
IX. SIMULATION MACHINING.....	38
X. OUTPUT NC CODE	40
XI. TOOL.....	40
APPENDIX I: THREE BASIC MODES OF INTERSECTING LINE MACHINE TOOL.....	41
1. BI-AXIAL MACHINING MODE (GROOVE MACHINING IS NOT SUPPORTED).....	41
2. THE SWINGING GUN HAS A PARALLELOGRAM STRUCTURE (PARALLELOGRAM MODE)	41
3. THE SWINGING GUN HAS A STRUCTURE WITH INFINITE SWIVELING HEAD (FITTING ANGLE MODE)....	41
APPENDIX II: REFERENCE FOR COORDINATE SYSTEM.....	43
APPENDIX III: INTRODUCTION TO DRAWING MODULE.....	44
1. BASIC OPERATION	44

2. INTRODUCTION TO DRAWING TOOL.....	45
3. EDIT AND MODIFY	50
SALES AND TECHNICAL SUPPORT	53

Chapter One Software Installation Guide

Notes:

The dongle delivered with the software’s installation disk must be inserted in the USB port before installation. The software should be installed by following the steps below. If the dongle is not inserted during installation, the software will be unavailable upon the completion of installation. In this case, it must be reinstalled.

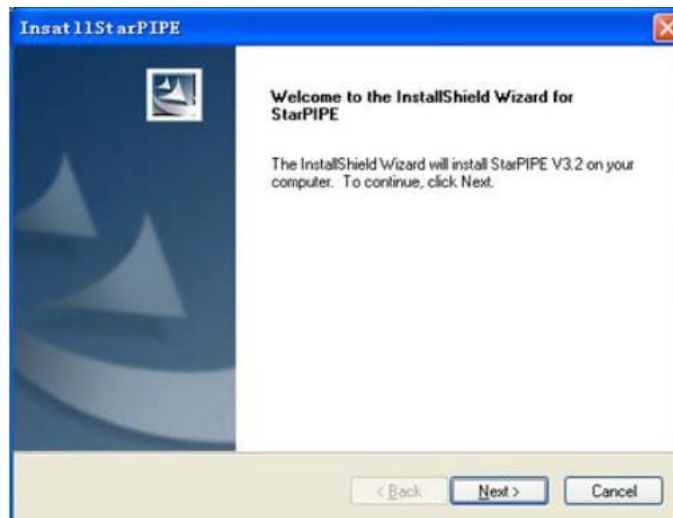
To read the electronic instructions of the software, Adobe Reader must be installed.

The software only supports the Chinese and English versions of Windows operating system. A compatibility problem may occur if it is installed on other language versions of Windows operating system. In this case, please install the English language package.

StarCAM 3D Intersecting Line Cutting Software (“StarPIPE”)

I. Steps for installing StarPIPE:

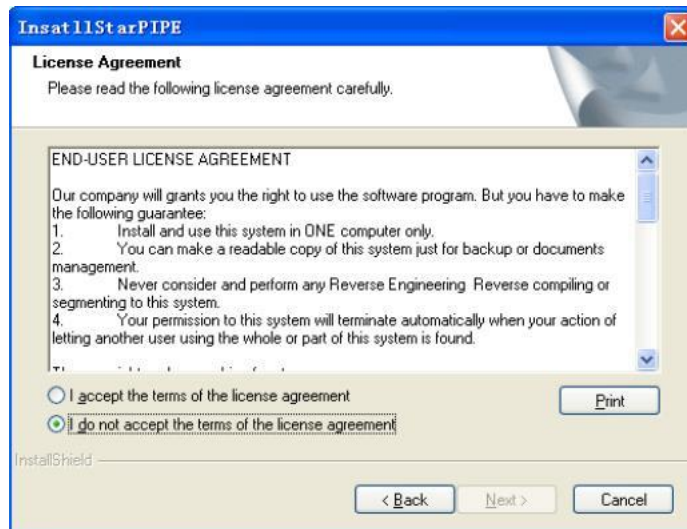
1. Double-click “Setup.exe” under the root directory of installation disk to start installation, as shown in the following picture:



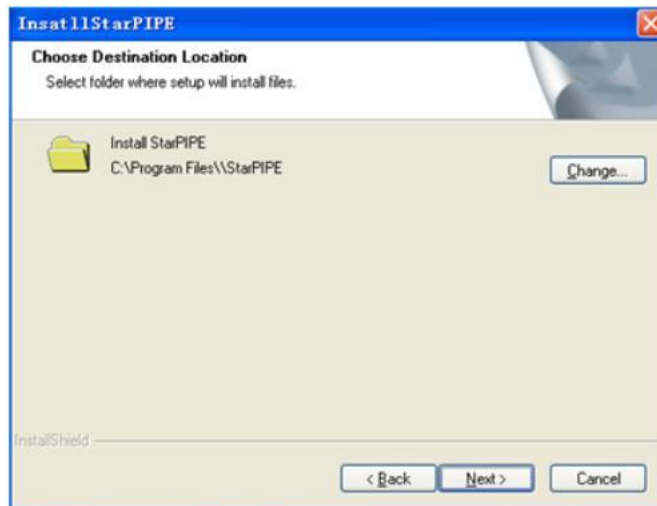
2. Insert the attached dongle as instructed.



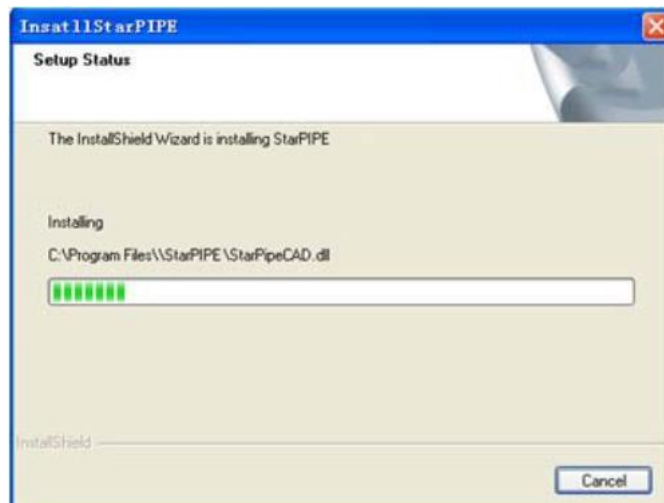
3. Click “Yes” to enter the License Agreement interface. If you agree the License Agreement, select “I accept the terms in the License Agreement”, and click “Next”.



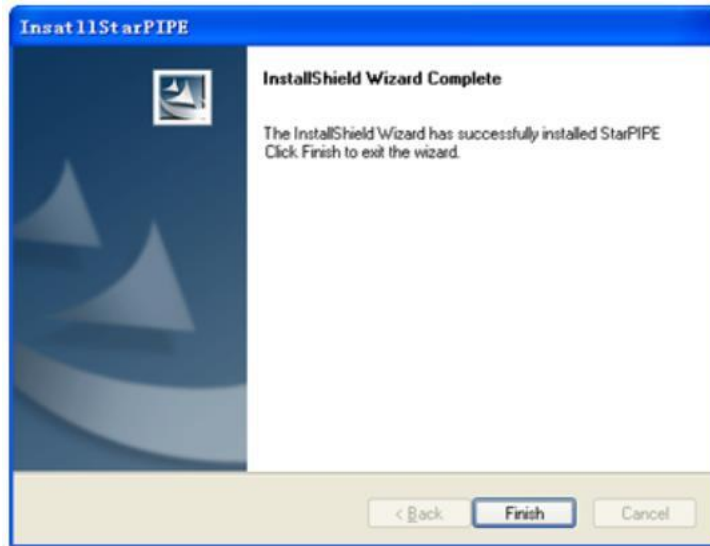
4. In the tip page of installation directory, you can click “Change” to select installation path. Otherwise, it will be installed according to default path.



5. Click “Next” to enter the interface for confirmation. If everything is correct, click “Install”. The system starts to copy and install the software.



6. Upon the completion of copying files, an interface appears to indicate the installation is completed. Click “Done”.



To start the software, click the “StarPIPE” short-cut icon on desktop.



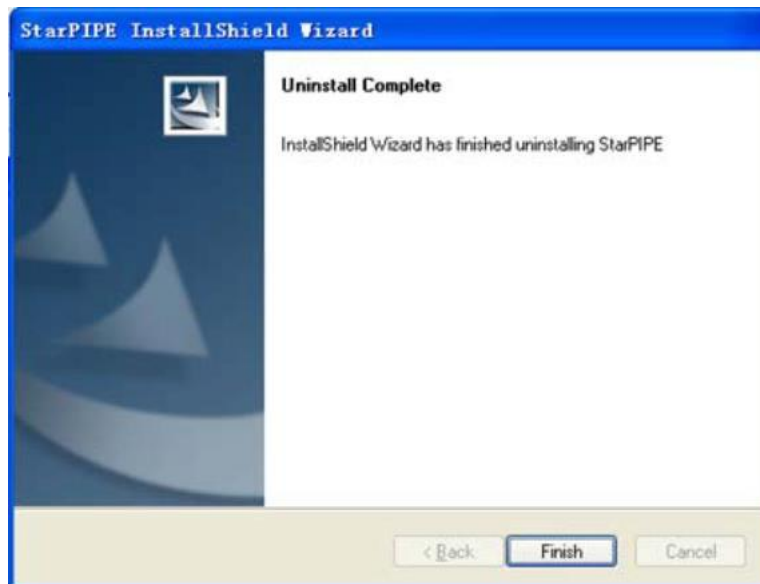
Note: If the dongle is not inserted during installation, the software will be unavailable upon the completion of installation. In this case, the dongle must be inserted to reinstall the software.

II. Instructions for uninstalling StarPIPE:

1. If it is necessary to uninstall the software, click “Start” > “All Programs” > “UnInstall” under “StarPIPE” to start the software’s Uninstall Wizard.
2. Select “Yes” when the inquiry dialog box appears, as shown in the picture below.



3. The following interface appears after the software is uninstalled.



Click “Done”, and the uninstalling process ends. The software can be also uninstalled by “Add/Remove Programs” on Windows’ control panel.

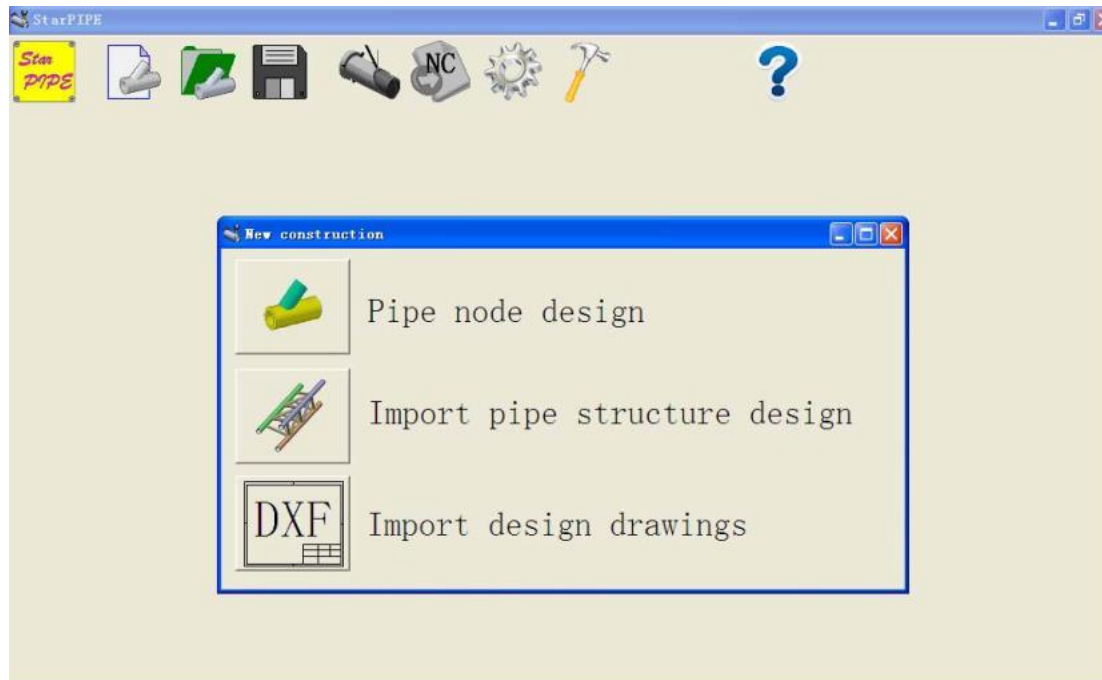
Chapter Two User Manual of StarPIPE

You must be familiar with your intersecting line cutting equipment before using StarPIPE. Therefore, it is necessary to read the user manual of machine tool. The content in Appendix I of our manual is for your reference.

If you are familiar with your intersecting line cutting equipment, you should set the basic machine parameters in “System Parameter Settings” when you first use the software. For detailed setting methods, refer to “VIII. System Parameter Settings”.

I. Introduction to operation interface

The operation interface of StarPIPE can be manipulated by icon selection, features and parameter modeling. You can complete the machining design, machining simulation and NC code output of single pipe fitting simply by clicking several tool buttons and entering characteristic parameters. Easy operation helps you get started quickly. The main operation interface is shown as below.



1. Functions of main toolbar:



Create a new design project of pipe fitting. It supports the modeling design of junction features of single pipe fitting and the modeling by importing CAD 3D image and CAD plan.



Open the saved design project task.



Save the current design project task.



Simulate machining for the current design task.



Generate the NC code of the current task.



Set cutting parameters, including automatic machining path, machine tool and format of NC code.



Utility Manager




Go to the next step

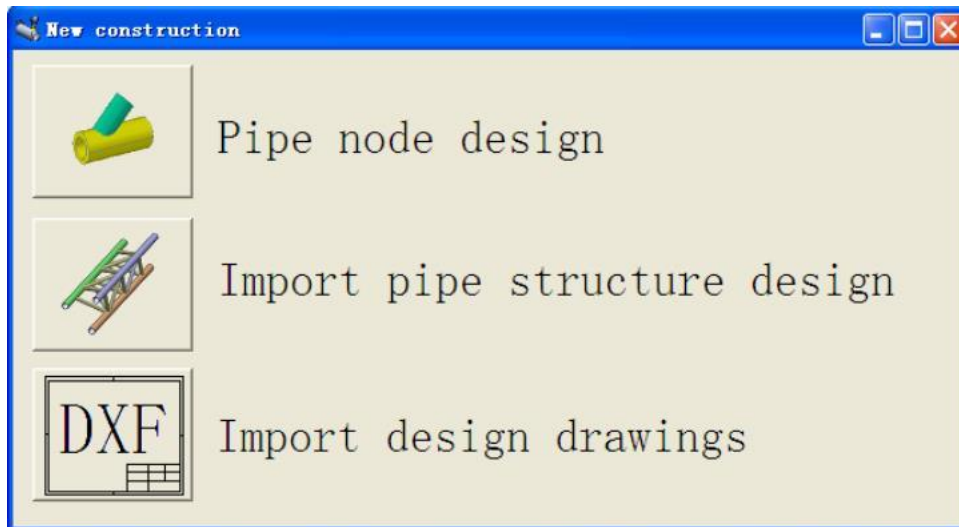


Return to the last step

2. Create a new project



Click  on the toolbar of main window, and a selection box of task type appears, as shown in the picture below:



Where:



Design interactive pipe junction. Feature modeling and parameter interaction are used to complete the design of pipe fitting junction. In this way, intersecting line cutting design, cutting simulation, nesting and NC code output of single pipe can be achieved rapidly.



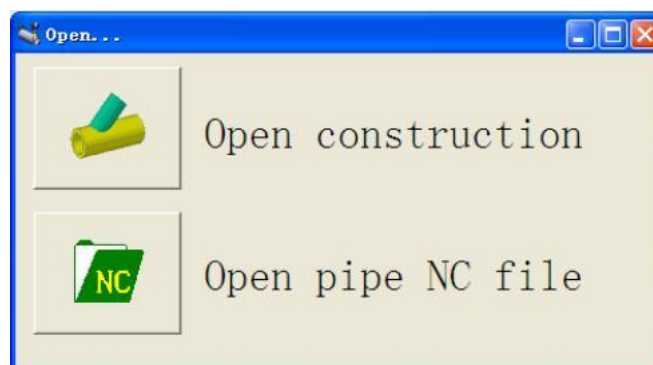
Invoke and import the pipe structure design graph that the user produces via the third party's design software (in 3D DXF format). After users select the pipe fitting to be machined, the system automatically generate the intersecting line for cutting, groove angle and 3D solid graph of the pipe fitting, as required by users' parameter settings. Different pipe fittings can be selected rapidly for intersecting line cutting, cutting simulation, nesting and NC code output.



Import the plan of designed pipe structure in DXF format. Users can generate the graph of pipe structure on the plan rapidly. It can be modified and edited on the interface for pipe junction design. Different pipe fitting can be selected rapidly for intersecting line cutting, cutting simulation, nesting and NC code output.

3. **Open a project**

You can open the saved design project and the generated NC code in the project. Select "Open" on the main toolbar and the selection box below appears:





Open the saved pipe machining project. Users can select the project path and file name on the dialog box for opening files. The system automatically enters the interface for junction design and display the design features for editing and modification by users.

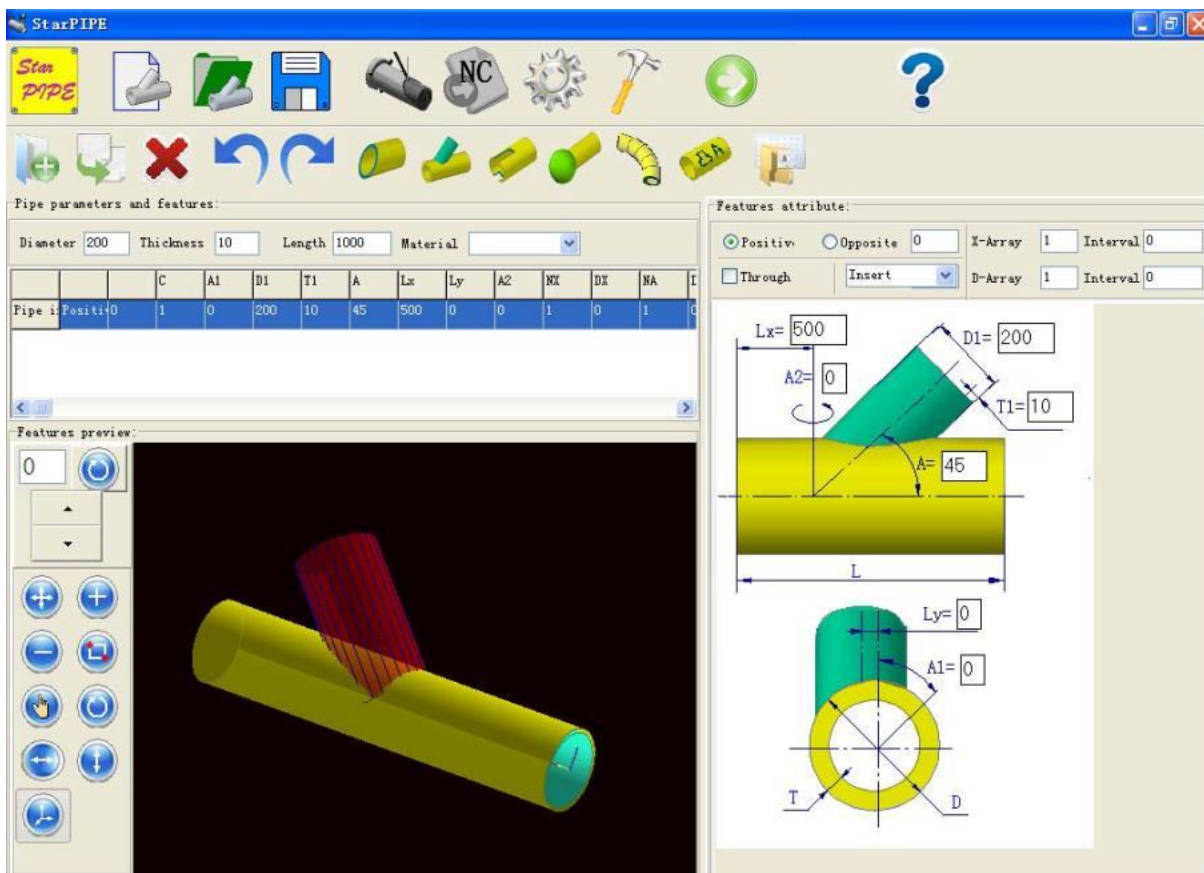


Open the generated NC code file. Users can select the project path and file name on the dialog box for opening files. The system automatically enters the interface for code simulation and displays the NC codes for simulation and modification by users.

Instructions: The format of the NC code to be opened shall be the same as that set in System Settings. Otherwise, simulation may fail. Users can edit and modify codes by using “Edit NC Code”, so as to match the current format.

II. Design pipe junction

When users select “Design Pipe Junction” in the “New” dialog box, the system automatically enters the mode for designing feature parameters of pipe junction, as shown in the picture below. In this mode, the intersecting structure design and nesting of single pipe fitting can be completed. It applies to the users who are not familiar with CAD drawing. Under this interface, users can complete 3D pattern formation, intersecting line formation, computer cutting simulation, nesting and NC code output of single pipe fitting rapidly according to its intersecting characteristics and geometric parameters. The design interface is as follows:



1. Feature toolbar

The functions of all tools in the secondary toolbar are as follows:



Start the design of new pipe junction, and clear all contents of the current design.



Copy the list template line currently selected by the user to the end of list line. In this way, a new feature can be copied rapidly.



Delete the features selected in the feature list.



Undo the last action.



Restore the last action deleted.



Add a truncating or slope feature in the designated position of the current pipe fitting. It is mainly used to generate the end cutting features at the left and right end of pipe fitting. By modifying feature parameters, different end-slope cutting or end cutting can be achieved.



Add an intersecting feature of pipe fitting in the designated position of the current pipe. It is mainly used to generate the relationship of intersecting features between pipe fittings. By modifying parameters, the intersecting features and intersecting relationship from different angles and in different positions can be achieved.



Add a cut feature in the designated position of the current pipe fitting. It is mainly used to generate the cut features in different positions of pipe. Multiple cut features like round, oval, rectangle, triangle and polygon are supported.



Add a ball-cut feature in the designated position of the current pipe. It is mainly used to generate the cut features for pipe welding involving ball structure parts.



Generate a “small shrimp waist” bend feature and automatically perform nesting cutting.



Generate the machining features of graph and text on the current pipe fitting. It is mainly used to process graph and text on pipe fitting.





Import a reference machining from users' part library for reference during design. Feature parameters can be modified to meet the design requirement.

2. Feature ribbon

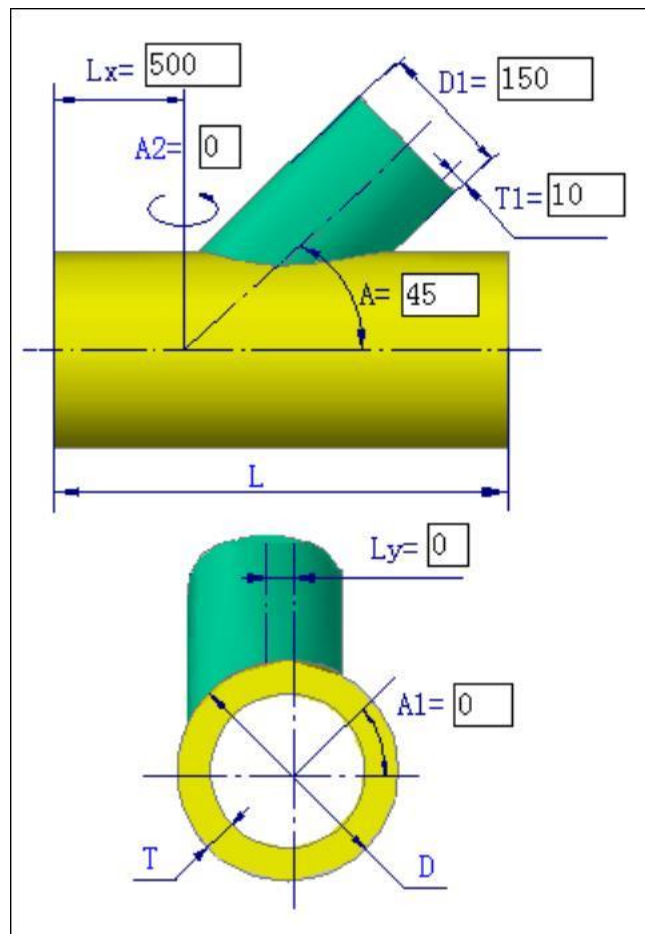
The function of all feature ribbons is as follows:

- 1) **List of feature parameters:** when users added the intersecting feature supported by the system to the current pipe fitting, the table will display the type and parameters of the feature added. The 3D image of the feature can be displayed in the Feature Preview Area. Users can delete a feature line, copy a feature or add a new feature by Feature Tool.

			C	A1	Type	La	Lb	A	Lx	Ly	A2	cr	NX	DX	NA
Pipe in	Positi	0	1	0	200	10	45	500	0	0	1	0	1	0	
End cu	Positi	0	2	0	45	0	0								
Incisi	Positi	0	1	0	2	100	50	0	50	0	0	0	1	0	1

Users can delete the selected line by choosing , or copy the selected feature line to the end of the list by choosing .

- 2) **Setting and legend of feature parameters:** in this area, users can observe the feature parameters that they selected in the list of feature parameters, and modify parameter values as required by construction drawings at any time to meet the requirements for machining. The modified feature parameters can be updated and displayed immediately in the Feature Preview Area.

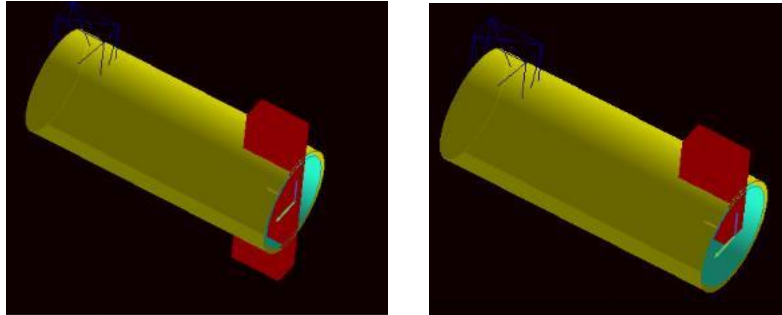


- 3) **Common properties setting of feature**

Features attribute:					
<input checked="" type="radio"/> Positive	<input type="radio"/> Opposite	<input type="text" value="0"/>	X-Array	<input type="text" value="1"/>	Interval <input type="text" value="0"/>
<input type="checkbox"/> Through	Node gap	<input type="text" value="----"/>	D-Array	<input type="text" value="1"/>	Interval <input type="text" value="0"/>

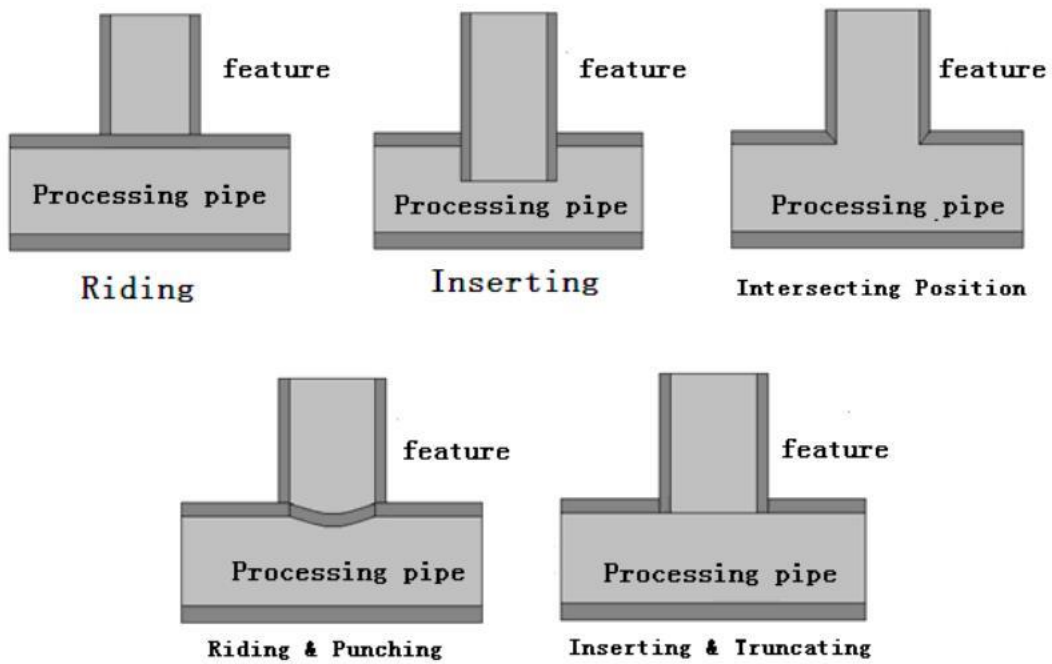
Position: the current feature can be set at the front end or back end of the pipe fitting.

Radial penetration: the current feature can be set as penetrating or not penetrating the pipe fitting. The schematic diagram below shows a rectangular cut that penetrates and does not penetrate the pipe, respectively.



Intersecting mode: when the intersecting feature is added, the intersecting mode between pipe fittings can be selected according to construction requirements. It reflects the cutting relationship between the pipe to be machined and the feature. Users can select different intersecting relationships according to actual demand. For example, structural weld assembly usually uses “Riding Position” or “Inserting Position”; oil pipeline usually uses “Riding & Punching Position” or “Inserting & Truncating Position”; and “Intersecting Position” applies to butt welding of process pipe fittings. The cutting schematic diagram of the intersecting relationship between pipe fittings is as follows:

Special tips: the intersecting relationship in the schematic diagram refers to the intersecting relationship between the machining feature and the pipe fitting to be machined, so it is not always equivalent to the relationship between secondary pipe and primary pipe on the design sketch. For example, when a pipe-intersecting feature is added, the feature will not lead to machining on the pipe to be machined if “Riding Position” is selected, but it will lead to machining on the pipe to be machined if “Inserting Position” is selected. In this case, the pipe to be machined can be regarded as “primary pipe”, and the feature can be regarded as “secondary pipe”.

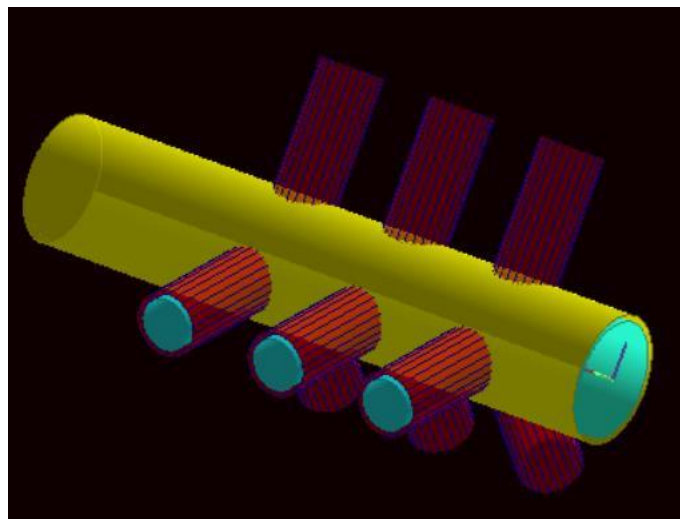


Axial array: multiple arrays (default: 1) can be arranged axially on the pipe fitting for the selected feature.

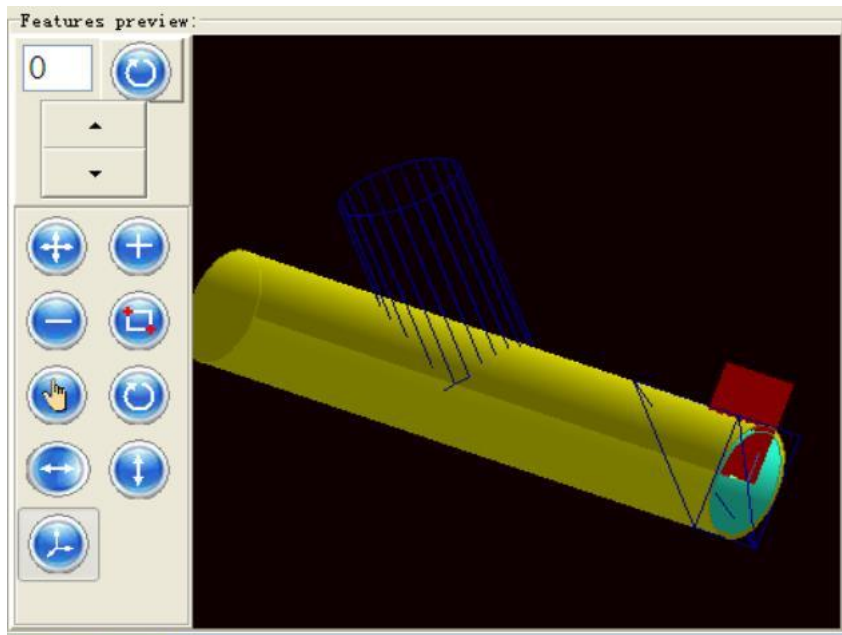
The axial interval on pipe fitting during feature arraying can be set as the interval.

Radial array: multiple radial arrays (default: 1) can be arranged on the pipe fitting for the selected feature.

The degree of radial interval on pipe fitting during feature arraying can be set as the interval (degree). The picture below shows an instance of round cut feature consisting of three axial arrays and three rows of radial arrays with an interval of 120 degrees:



- 4) **Feature Preview Area:** the intersecting relationship between the feature added and the pipe fitting is displayed three-dimensionally in this area, and can be updated and displayed immediately.



The function of relevant tool buttons is as follows:



Control automatic rotation of pipe fitting around its axis to observe the design effects. The rotation angle can be also adjusted by “+/-” button.



Display all images.



Display the window in a partially enlarged view.



Enlarge the preview. The preview can be also enlarged by scrolling the mouse wheel.



Shrink the preview. The preview can be also shrunk by scrolling the mouse wheel.



Display the pipe fitting vertically.



Display the pipe fitting horizontally.



Display the pipe fitting equilaterally.



Move the image via the mouse.

3. Modeling instructions for Feature Tool

Feature Tool can help you realize the modeling and machining process for pipe fitting cutting rapidly:

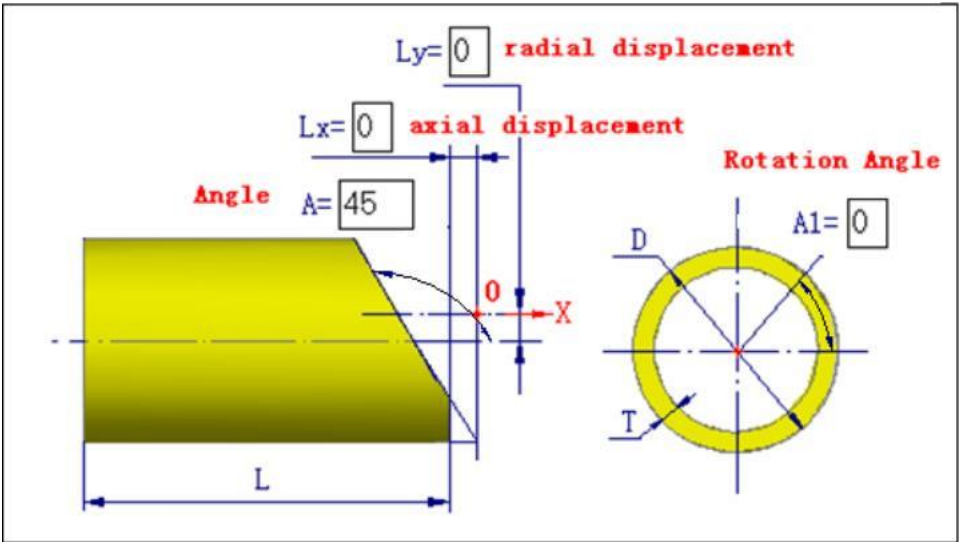
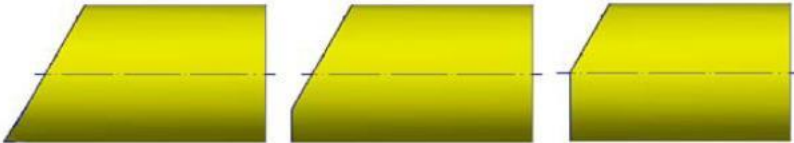
Users can add different feature relationships to the left end, the right end and the middle part of basic pipe fitting by using its Feature Tool. All feature relationships can be used in combination repeatedly. Different intersecting cutting conditions can be met by parameter setting, so as to satisfy the requirements for complex intersecting cutting.



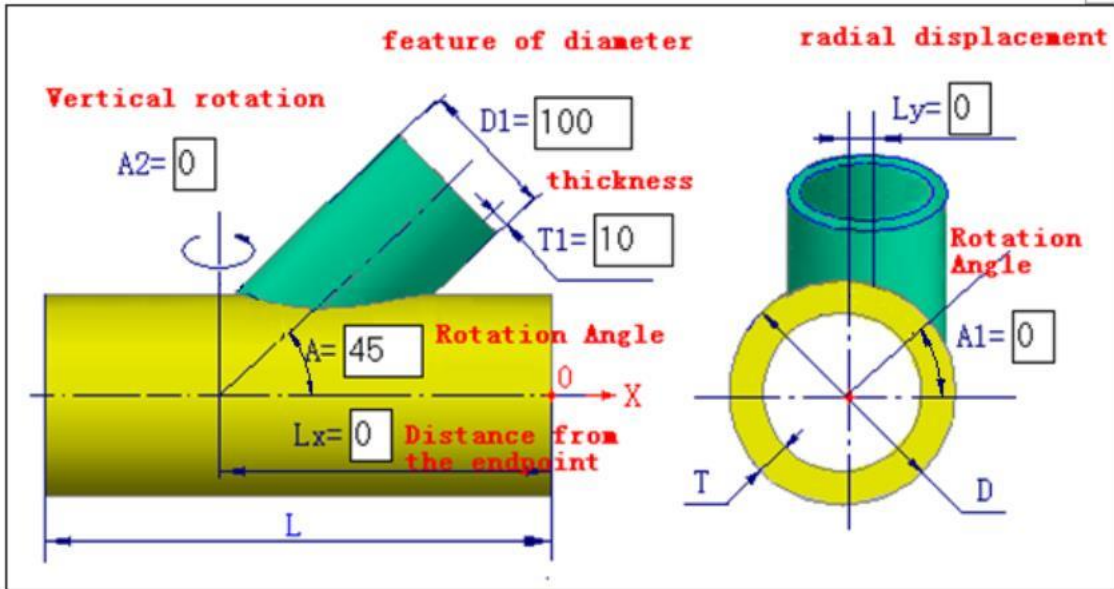
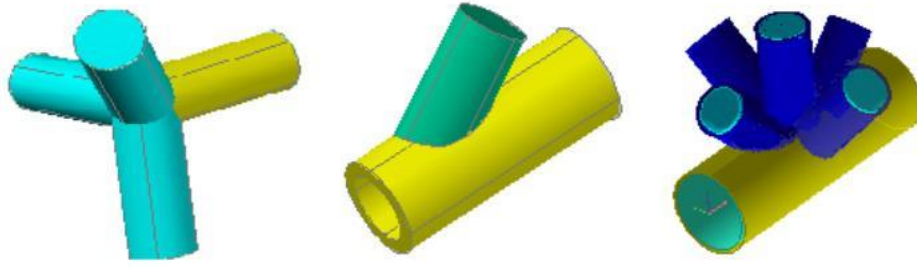
The examples of modeling type and parameter specification of all feature tools are as follows:



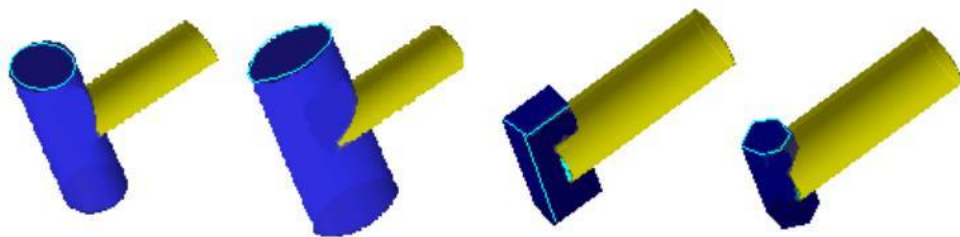
End cutting feature. With this feature, users can add a truncating or slope feature in the designated position of the current pipe fitting. It is mainly used to generate the end cutting features at the left and right end of pipe fitting. By modifying parameters, different truncating cutting or end-slope cutting can be achieved. The picture below shows the examples of some truncating types and parameter setting:

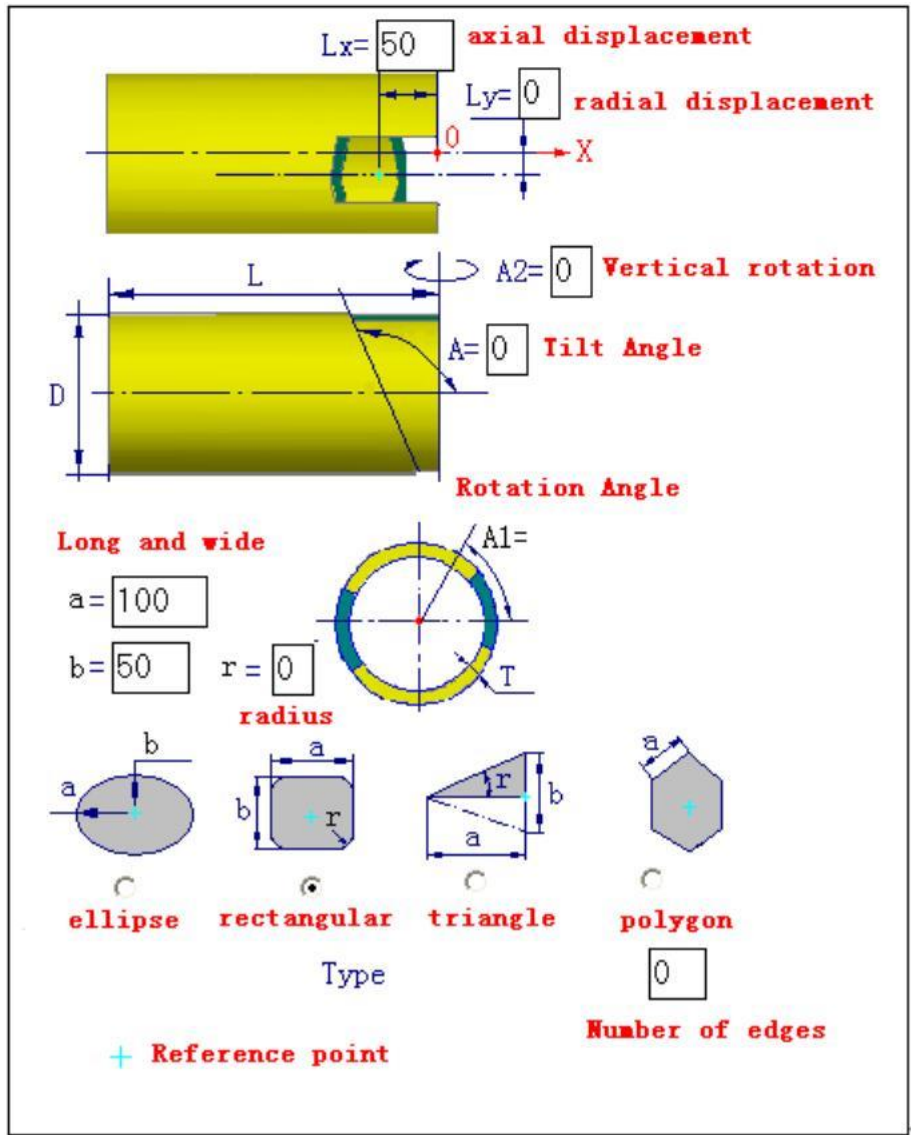


Intersecting feature. With this feature, users can add an intersecting feature of pipe in the designated position of the current pipe fitting. It is mainly used to generate the intersecting relationship between pipe fittings. By modifying parameters, the intersecting features from different angles and in different positions can be achieved. The picture below shows the examples of some intersecting types and parameter setting:

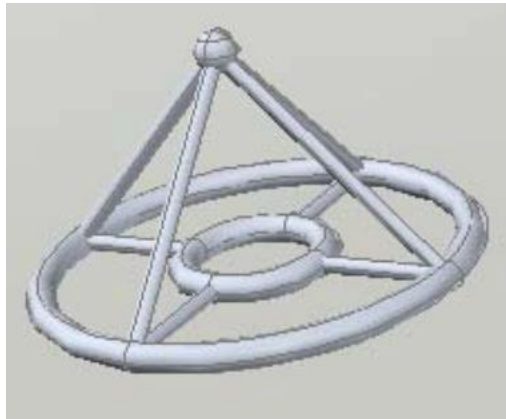


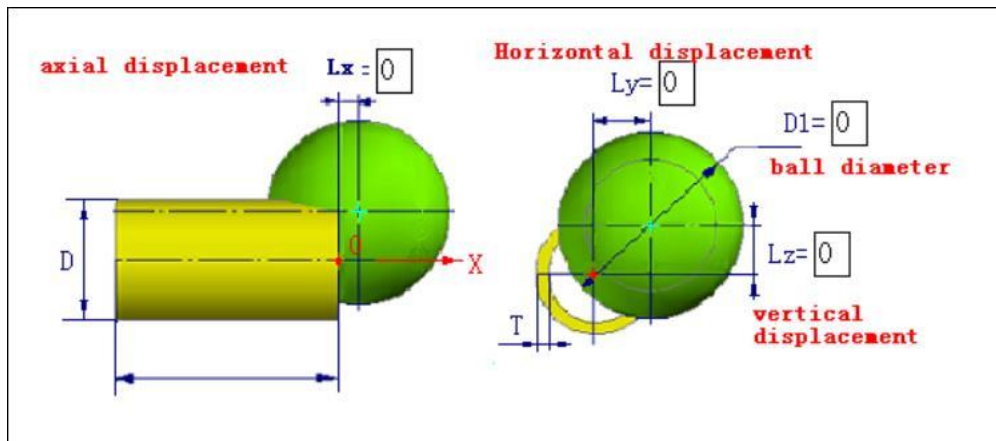
Cut/hole feature. With this feature, users can add a cut or hole feature in the designated position of the current pipe fitting. It is mainly used to generate the circular or non-circular cut features in different positions of pipe fitting. Multiple cut features like round, oval, rectangle, triangle and polygon are supported. Cut-through or no cut-through is optional. The picture below shows the examples of some cut types and parameter setting:



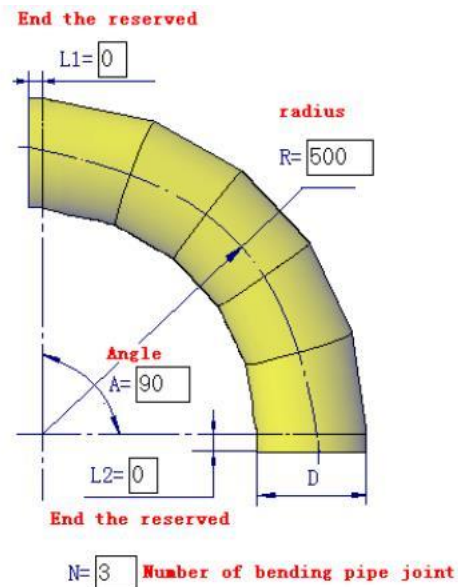
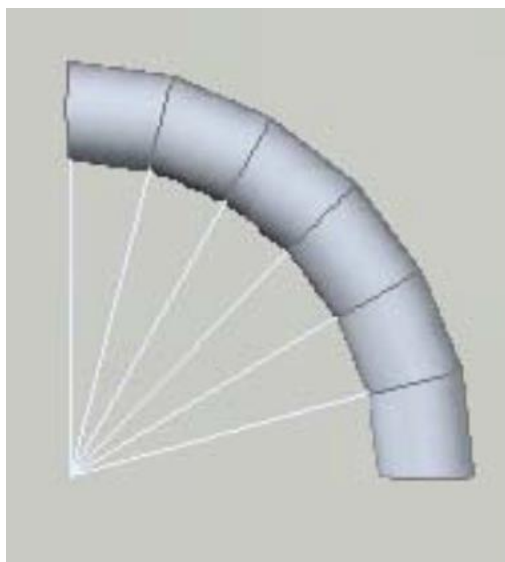


Add a ball-cut feature in the designated position of the current pipe fitting. It is mainly used to generate the cut features for pipe fitting welding involving ball structure parts. The picture below shows the example of a ball-cut and parameter setting:





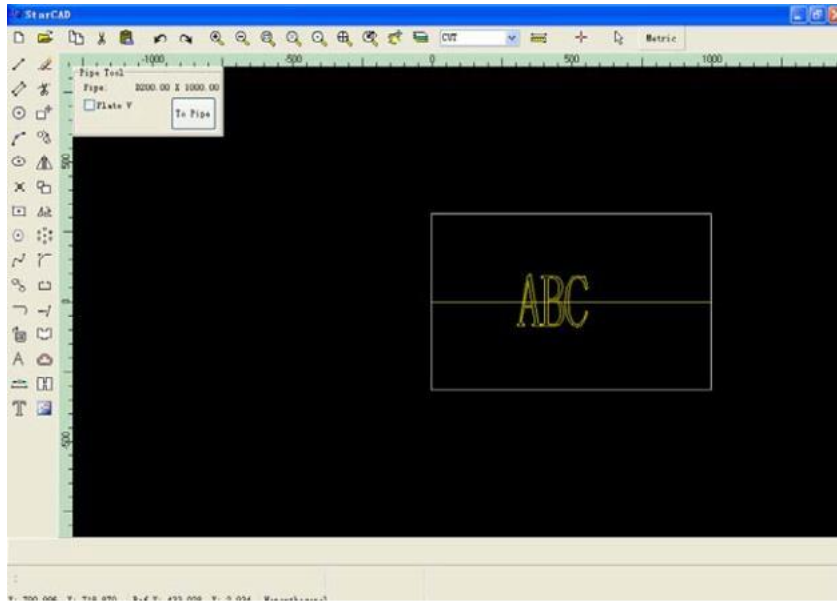
Bend feature. With this feature, users can design and generate a “small shrimp waist” bend feature rapidly and perform nesting cutting. The picture below shows the example of “small shrimp waist” bend and parameter setting:



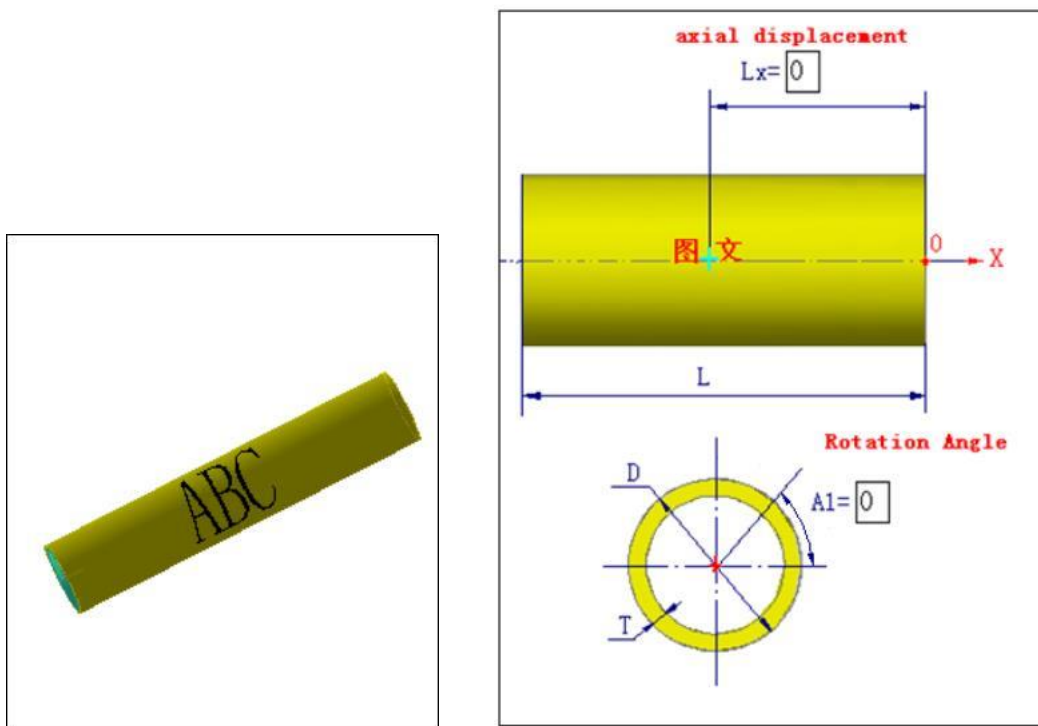
Graph-text feature. With this feature, users can generate the machining features of graph and text on the current pipe fitting. Groove machining is not available during graph and text cutting. When the machining features of graph and text are added, the system enters the CAD drawing module automatically. With the module, users can draw the machining graph or insert text and graph from gallery. Moreover, users' picture can be also converted to graph automatically.

The CAD operation interface is as follows. Its operation mode is similar to other common CAD software. The graphic file drawn by the third party's software in the DXF format can be also added. To add a graph, the “Move” tool can be used for positioning the base point (a specific point that users designate on the graph). After the drawing process is completed, click “Output” to add the feature of the drawn graph to the current pipe fitting.

Refer to Appendix III for learning how to use the tools of drawing module.



The picture below shows the example of adding graph and text on pipe fitting and relevant parameter setting:

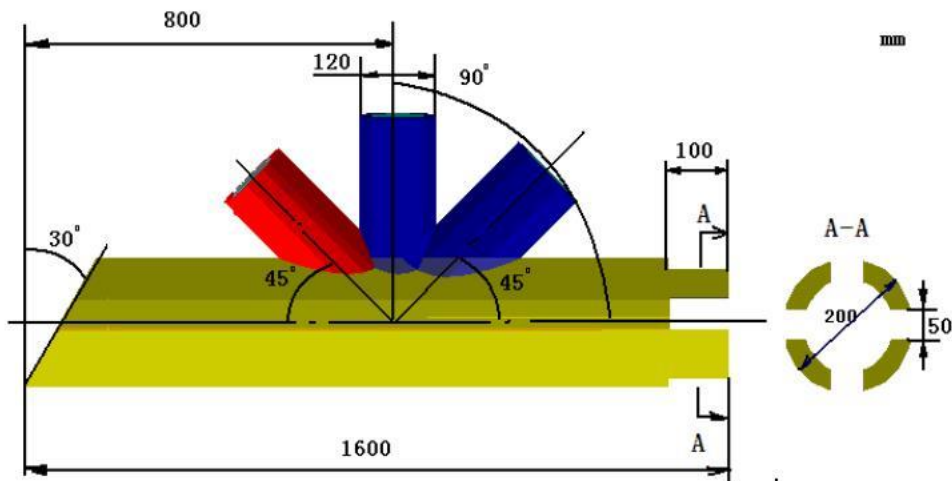


Complex junction feature. With this feature, users can rapidly import a reference design project from their part library. Users can generate new machining tasks by modifying and editing parameters after importing.

III. Instances for modeling of pipe junction feature


For example, it is required to cut a single pipe fitting now, and its size is shown in the figure below: a 30-degree truncating end-cutting is performed on the left side, four rectangular cuts at 90-degree angles from each other are made at the right end, and the middle part is penetrated by three circular


pipes.





1. Feature modeling


The design process of feature modeling is as follows:


- 1) Click  on Main Toolbar, and select “Design Pipe Junction” on the subsequent selection box of task type.
- 2) Input the basic size of pipe fitting in the dialog box for pipe junction design: length – 1,600mm, pipe diameter - 200mm, and pipe thickness - 10mm.
- 3) Add the following features in the pipe, and modify the parameters as follows:


Add a  feature, select the left end as the position, $A=30$ degrees, and leave the rest of the parameters set to the default values.

Add a  feature, select the left end as the position, $D1=120$ mm, $Lx=800$ mm, $A=90$, and leave the rest of the parameters set to the default values.

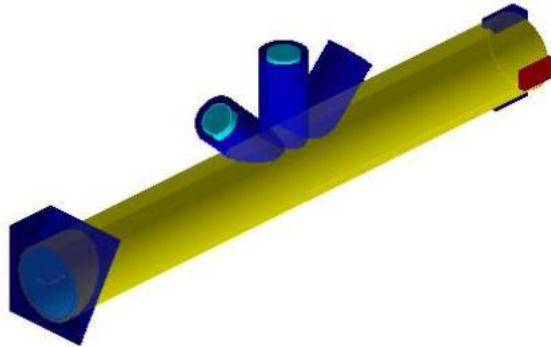
Add a  feature, select the left end as the position, $D1=120$ mm, $Lx=800$ mm, $A=45$, and leave the rest of the parameters set to the default values.

Add a  feature, select the left end as the position, $D1=120$ mm, $Lx=800$ mm, $A=135$, and leave the rest of the parameters set to the default values.

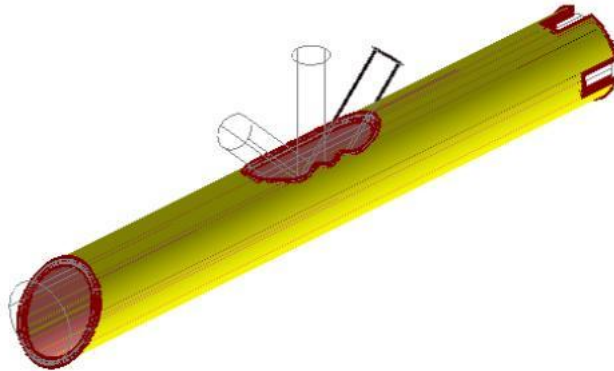
Add a  feature, select the right end as the position, $La=100$ mm, $Lb=50$ mm, $A=0$, and leave the rest of the parameters set to the default values.

Add a  feature, select the right end as the position, $La=100$ mm, $Lb=50$ mm, $A=90$, and leave the rest of the parameters set to the default values.

4) The completed feature graph is as follows:




The generated features represent the machining features on the pipe fitting, and provide a basis for intersecting in future machining process. The actual cutting graph is as follows:

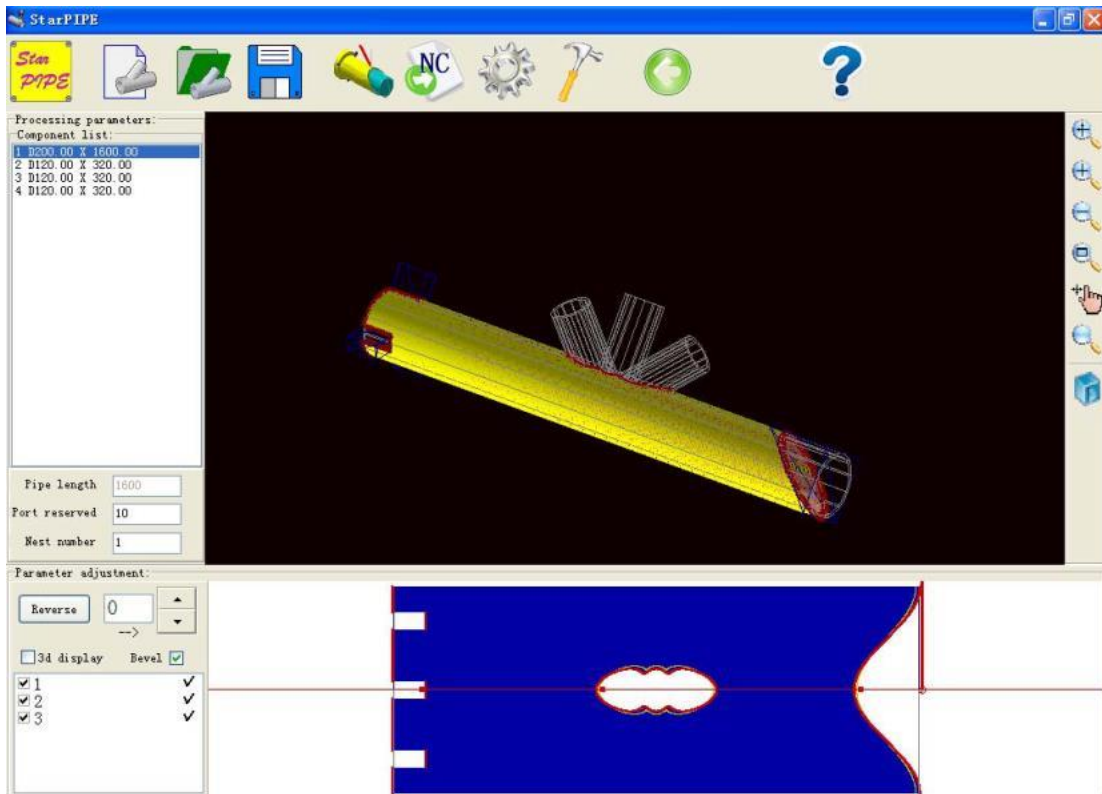


By proper combined use of basic feature tools, a more complex intersecting cutting relationship can be generated. Besides, it also provides a basis for subsequent machining path and groove cutting.


2. Machining setting


Machining setting can be conducted upon the completion of junction feature design.

- 1) Click  (“Next”) to enter the interface for machining setting. Parameters such as machining direction, starting-tool point, position of lead-in, and quantity of nesting can be set, as shown in the picture below:



The 3D graph of the pipe fitting to be machined is displayed on the upper right of the page, and the expanded view of the pipe fitting is displayed on the lower right corner. Users can set machining direction, starting-tool point and position of lead-in on the expanded view.

In the parameter selection area titled “Adjust machining parameters”, reverse cutting of pipe fitting can be conducted in the expanded view by pressing the “Pipe Reverse” button. By using , the starting angle of lead-in and lead-out on axis A (the starting-tool point is determined by the red dot – the point of intersection between the red horizontal line and the profile of pipe fitting) can be adjusted. Users can drag the red dot to any position. “3D Display” determines whether the pipe is displayed in the expanded view or 3D view.

The cutting list on the lower left corner can be used to determine whether certain machining process should be conducted (all machining processes should be conducted by default). If some cutting process needs not to be machined, click the check box corresponding to the process to uncheck it. “√” and “×” after each machining process indicate whether the groove is machined or not. Whether it is machined or not can be selected individually. Or press the  selection box on its top to machine all grooves or machine no grooves.



The size of the display area of pipe and expanded view can be changed by selecting the

Quantity of nesting: it indicates the quantity of nesting for the pipe fitting to be machined.

Length of pipe: it automatically displays the length of pipe necessary for machining based on the quantity of nesting set by users. The length is the product of the length of single pipe and the number of pipe to be machined, based on which the users can prepare materials.

Port reservation: it is used to set the reservation length of pipe port when the pipe's edge is not trim.

You can check the structural design on this page. If you are not satisfied, click the Back button

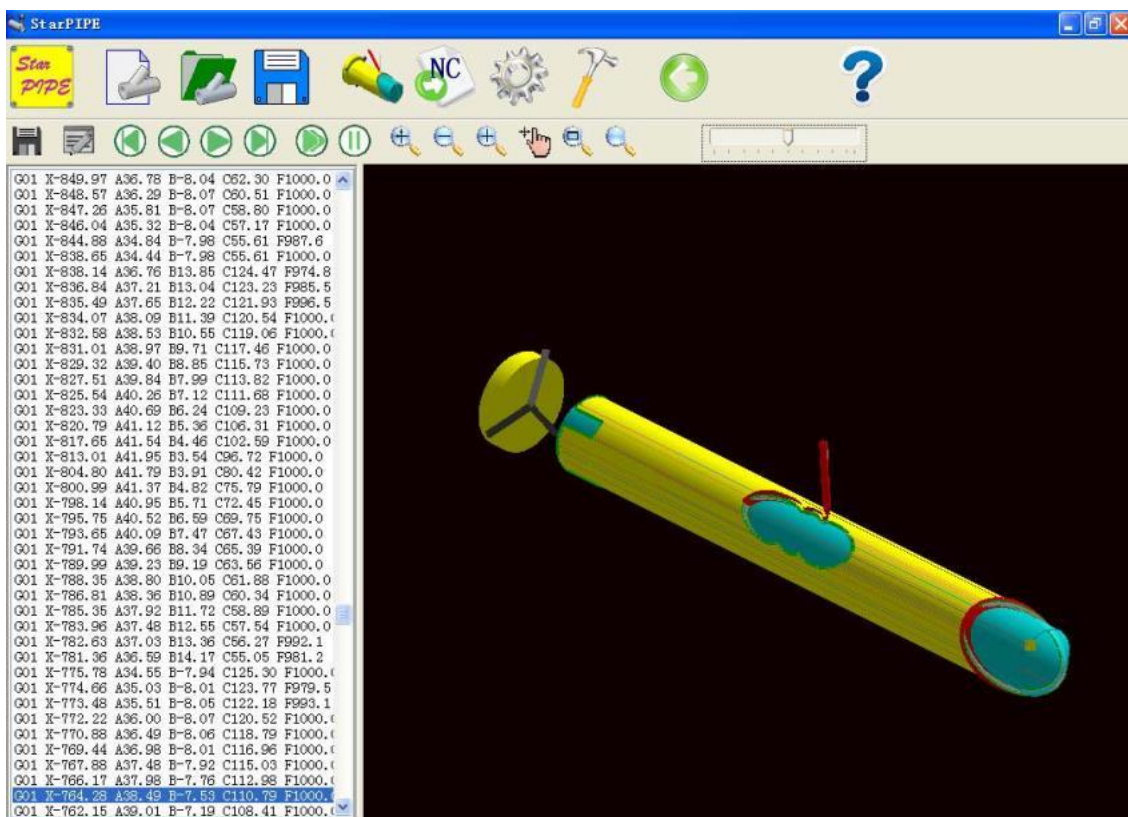


and return to the feature design page for redesigning. If you are satisfied, click the simulation tool for simulation machining.

3. Simulation machining



Click the Simulation Machining tool to enter the interface for simulation machining, as shown in the picture below:



The NC code for cutting the pipe fitting is displayed on the left side of the window, while the dynamic demo for simulation machining is displayed on the right side. The secondary control tools for simulation window are introduced as follows:



Let the pointer of NC code point to the first line of NC code.



Let the pointer of NC code move up by one working step. Meanwhile, the track simulation area displays the graph of the previous working step.



Let the pointer of NC code move down by one working step. Meanwhile, the track simulation area displays the graph of the next working step.



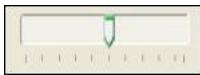
Let the pointer of NC code point to the end line of NC code.



Start continuous simulation of working steps according to automatic simulation interval set by users.



Pause continuous simulation of working steps.




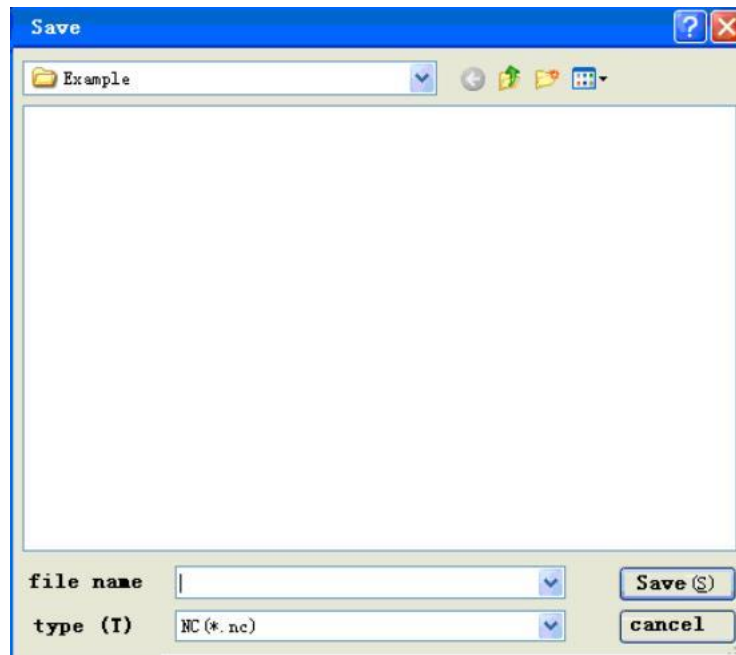
Adjust simulation speed. The speed of simulation machining can be changed by dragging the progress bar.

During simulation, the parameters including punching, lead-in and lead-out, machining path and angle of groove are displayed dynamically on the right side of the screen. During dynamic display, users can control the direction, size and angle of the display area by using the viewport control tool, so as to observe whether it is machined properly.

4. Output NC code



If the simulation machining is appropriate, output the NC code for machining. Click the  button on the main toolbar to output the NC code necessary for machining with respect to the current junction design of pipe. After clicking, the system will ask you to confirm the location of NC code where it should be saved and its file name. Click “Save” to complete the output process. The interface is shown as below:



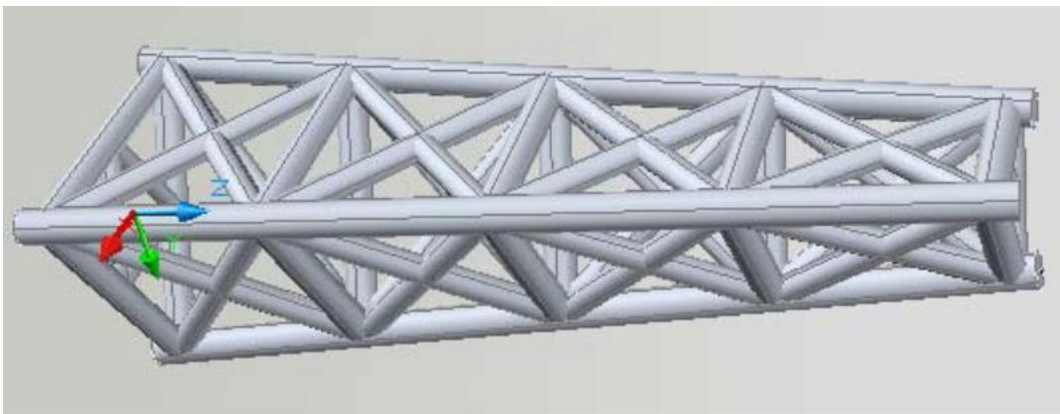
IV. Import CAD pipe structure graph

Import the 3D solid structure graph of pipe in the DXF format designed by AutoCAD. Users can select the corresponding pipe in the structure graph for pipe cutting.

1. Draw structure graph of pipe fitting

The instructions below should be followed when users draw the structure graph of pipe fitting with AutoCAD:

When users draw the pipe structure graph on the Solid Edit Interface of AutoCAD, the center line of pipe fitting should be used as the baseline to draw the relative angle, distance and connection between pipe fittings. Users can simply use solid cylinder for drawing, and draw the outside diameter only (the inside diameter can be provided after pipe cutting is imported). Do not conduct the Boolean operations including union, intersection and difference for the pipe fittings in the structure graph upon the completion of drawing, so as to maintain the independence of pipe fitting size. The picture below shows an example of pipe structure graph drawn by AutoCAD:




Go to “Modify” > “3D Manipulate” > “Extract Edge” on the Solid Edit Interface of AutoCAD to generate the boundary lines of pipe for drawing solid graph. To save the pipe structure graph,

If users decide to draw the graph of pipe's structural parts with SolidWorks, do not check the "Consolidate Results" option when drawing part features, so as to maintain the relative independency of parameters of all pipe fittings. If the parts have been consolidated already, right click each feature separately on feature tree, and select the "Edit Feature" tool in the subsequent shortcut menu of feature. Next, uncheck "Consolidate Results" in the feature dialog box, and click "√" to cancel the consolidation of features. The graph should be saved as a file in ACIS format. Enter AutoCAD again, and create a new file. Select "ACIS file" under the "Insert" menu to insert the ACIS file. Go to "Modify" > "3D Manipulate" > "Extract Edge" to extract solid edges and save them as the DXF-formatted file. If users draw the graph of pipe's structural parts with ProE, it can be also saved in ACIS format by selecting "Save a Copy" under "File".

2. Import pipe structure graph

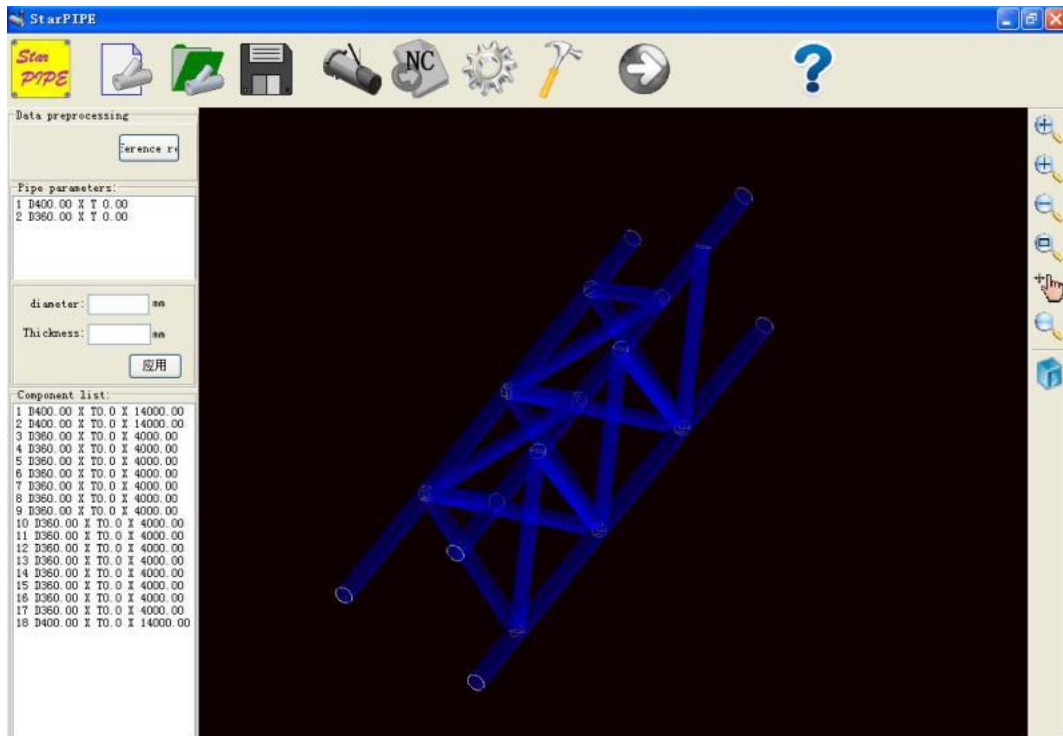


Choose the  function in the selection box of "Create a New Project", and the dialog box for opening the file of DXF structure graph is then displayed as below:

Select the solid pipe structure graph saved in DXF format, and click "Open".

3. Set pipe fitting parameters

The structure graph can be displayed in the 3D pipe structure window (as below), when users open the graphic file of 3D pipe structure. If no graph is displayed, it is possible that the graph has been treated with the Boolean operation of "union", so the system fails to identify it. In this case, it is necessary to explode the structure graphs of all pipe fittings in the solid interface of AutoCAD, and import them again.



Click a pipe fitting in the “Pipe Fitting Parameters” column above, the pipe fitting will be highlighted on the right side. Select it and its length and wall thickness are given in the left parameter input field. If the thickness of the pipe fitting has been given during drawing, it is unnecessary to input thickness. “Outside diameter” is determined by the outside diameter of the pipe fitting in the solid graph. If it is necessary to modify the outside diameter, it can be also modified here.

If the pipe structure consists of multiple pipe fittings with different diameters and thicknesses, it is required to set the wall thickness of raw material of all pipe fittings successively. Otherwise, the system will warn you:



If the graph displayed when opening the DXF solid file has excessive faces or lines, click the “Clear Interference” button to eliminate interferences.

4. Set machining parameters

After the parameters of all pipe fittings have been input, if users only select one pipe fitting to be machined in the list on the lower left (users can also click the pipe fitting to be machined on solid graph, and select the pipe fitting to highlight it in yellow), click “Next” to enter the machining design interface of pipe fitting; if multiple pipe fittings are selected, the “Next” button will be disabled. The next operation mode is the same as junction design. Please refer to Section II “Design pipe junction”.

5. Simulation machining

Refer to “3. Simulation machining”, Section III.

6. Output NC code

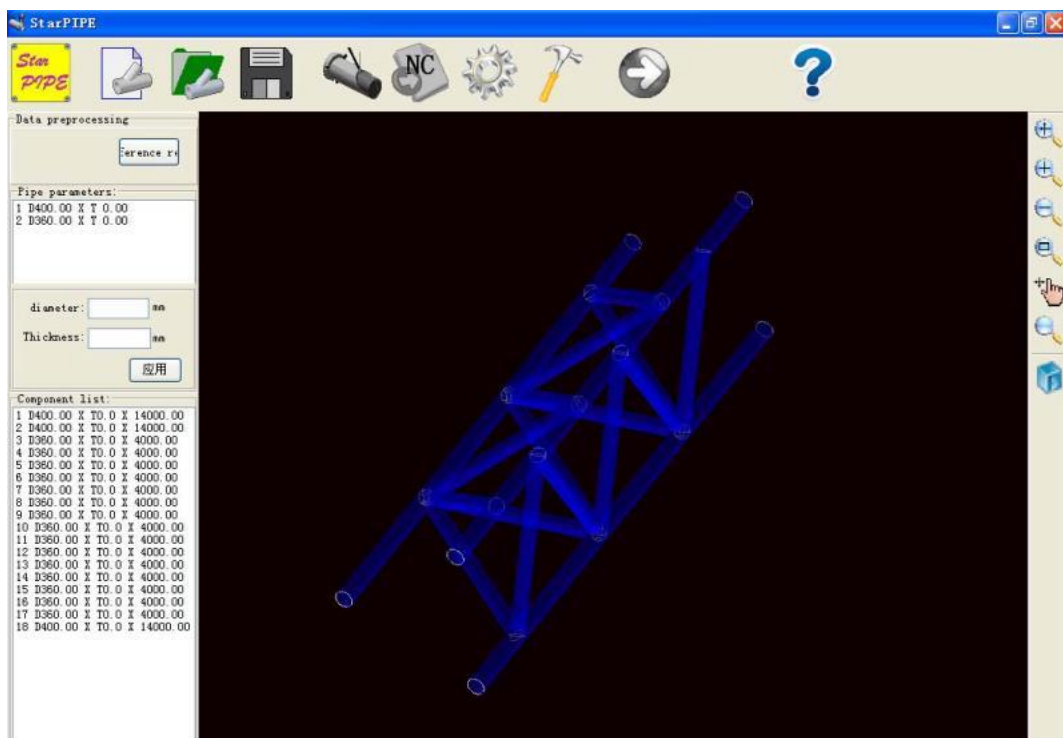
Refer to “4. Output NC code”, Section III.

V. Instance of importing pipe structure graph

Instance: let’s import the pipe structure graph “Triangular Truss Structure” from “Example” of installation directory. The graph has been drawn three-dimensionally using AutoCAD, and manipulated by going to “Modify” > “3D Manipulate” > “Extract Edge” on the Solid Edit Interface of AutoCAD as required.

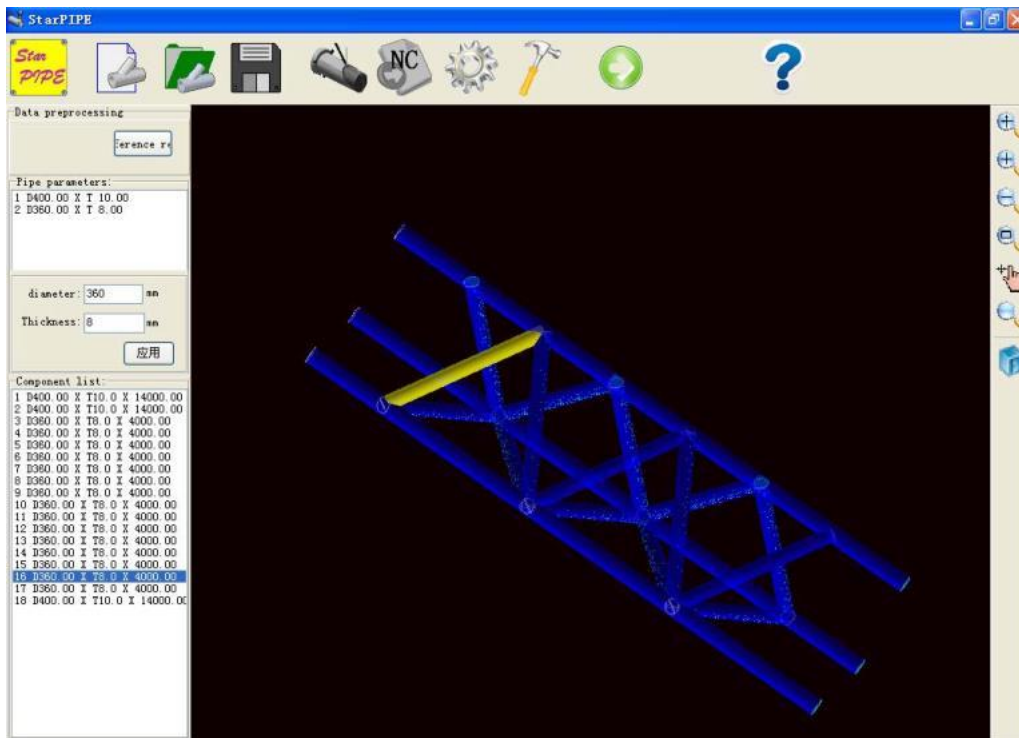
1. Import pipe structure graph

Select “New” on the main toolbar, and select “Import pipe structure graph” in the subsequent dialog box. Next, in the dialog box for opening files, select the “Example” directory from installation directory. Select the .dxf file of Triangular Truss Structure, and click “Open”, as shown in the picture below:



2. Set pipe fitting parameters

Select one pipe material in the list on the top, and enter the outside diameter and thickness of pipe fitting in the input field below. Go to the next step until all parameters of the pipe material are input. After all parameters of the pipe fitting are input, if users only select one pipe fitting to be machined in the list on the lower left (users can also click the pipe fitting to be machined on solid graph, and select the pipe fitting to highlight it in yellow), click “Next” to enter the machining design interface of pipe fitting; if multiple pipe fittings are selected, the “Next” button will be disabled.



3. Modify pipe junction

If it is necessary to modify pipe machining, users can modify and edit it on the pipe junction design page. The operation mode is the same as junction design. Please refer to Section II “Design pipe junction”.

4. Simulation machining

Refer to “3. Simulation machining”, Section III.

5. Output NC code

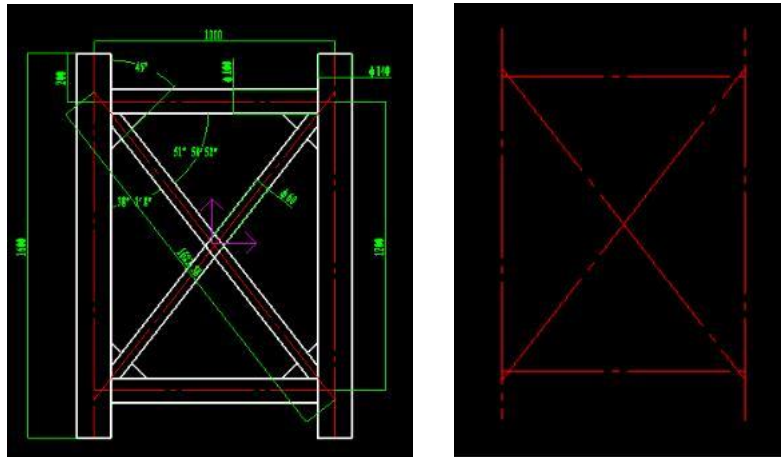
Refer to “4. Output NC code”, Section III.

VI. Import design drawing

With this feature, the pipe structure graph (in DXF format) designed by users can be imported directly. Geometric parameters on the graph can be used to generate rapidly the 3D graph of intersecting relationship of pipe fittings, so as to reduce the amount of computation and improve the design speed during pipe fitting design. The current version of the software is only able to generate the plane structure graph, but unable to generate the 3D pipe structure graph.


Tips: only the center line of pipe fittings can be drawn in the plane geometric graph drawn by users. The length of the center line should be the same as that of pipe fittings. Users provide the pipe diameter and wall thickness of pipe fittings after the graph with the center line is imported. If marks, text, and other layers are drawn on the graph machined by users, it is necessary to ensure the length of center line of pipe fitting is the same as the length of pipe fitting, delete the solid of other layers, and save the graph as a DXF file. If other layers are not deleted, there will be more lines after the graph is imported. Users shall determine the position of center line by referring to the original drawing. Set forth below is an illustration

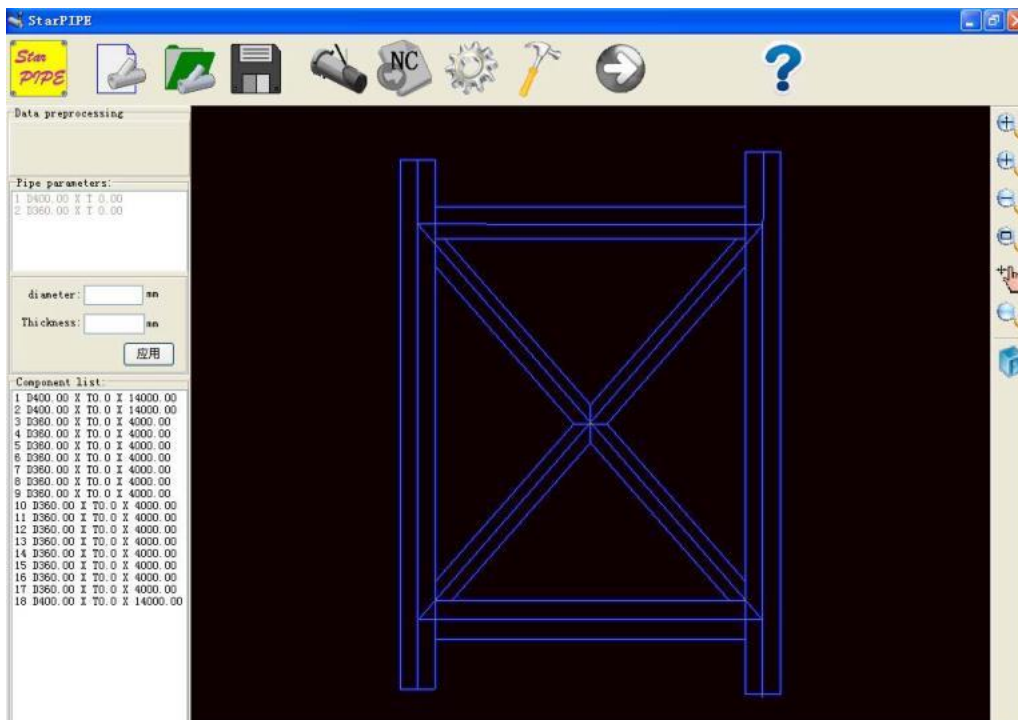
of the foregoing:



Operating procedure:

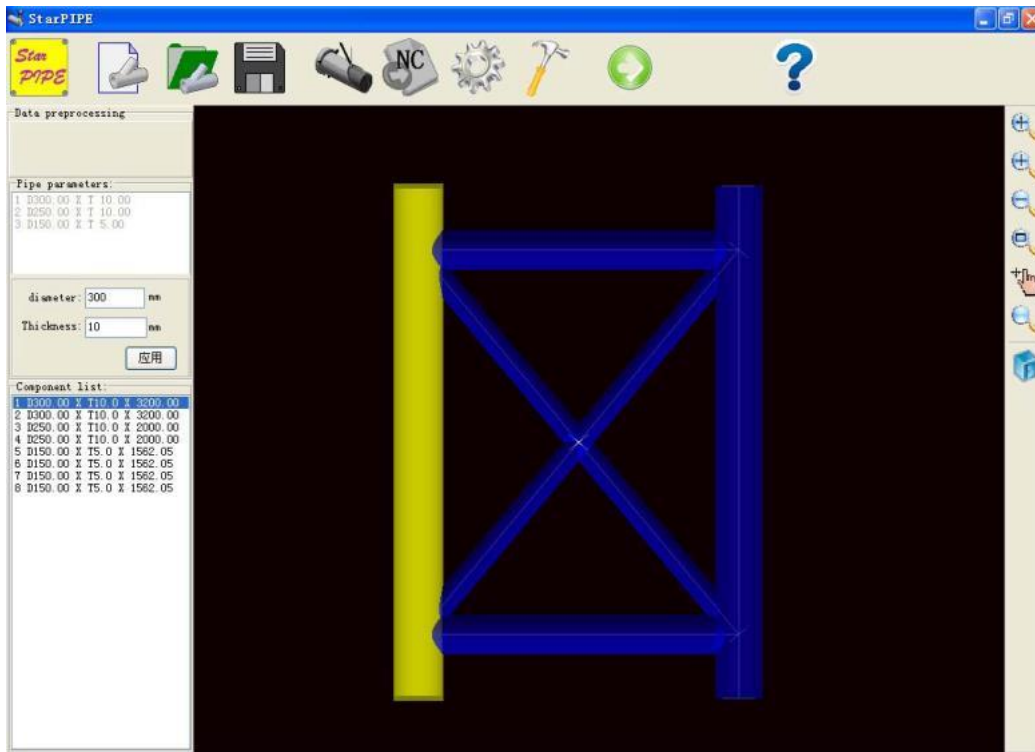
1. Import design drawing

Click the “Import Design Drawing” tool () , and the system will ask you to open the plan you have designed (the drawing must be a plan in DXF format). Select the file name to be machined from the dialog box for opening files and click “Open”. The following picture shows an example of the screen displayed after a file is opened. You can see that the solid of parts layer in the graph is not deleted.



2. Set pipe fitting parameters

Click one or multiple center lines (axes) of the graph to machined in Graphic Preview Area. Click again to unselect a center line if necessary. A solid cylinder will be displayed on the center line selected. You can enter its outside diameter and wall thickness in the left parameter area. The parameters of the solid set by users will be displayed on the left side. Input the parameters of all pipe fittings successively, as shown in the picture below:



After all parameters of the pipe fitting are input, if users only select one pipe fitting to be machined in the list on the lower left (users can also click the pipe fitting to be machined on solid graph, and select the pipe fitting to highlight it in yellow), click “Next” to enter the machining design interface of pipe fitting; if multiple pipe fittings are selected, the “Next” button will be disabled.

3. Modify pipe junction

If it is necessary to modify machining, users can modify it on the pipe junction design page, or add other machining features. The operation mode is the same as junction design. Please refer to Section II “Design pipe junction”.

4. Simulation machining

Refer to “3. Simulation machining”, Section III.

5. Output NC code

Refer to “4. Output NC code”, Section III.

VII. View manipulation

In all the 3D preview interfaces, users can control how the 3D graphs can be displayed by using view control tools and the mouse. The functions of all tools are described below:



Full Screen: it is used to display all contents of the graph in full screen in graphic display area.



Enlarge Display: it is used to display the contents on current screen in an enlarged view. This function can be also achieved by clicking the mouse wheel and moving up.



Shrink Display: it is used to display the contents on current screen in a shrunk view. This function can be also achieved by clicking the mouse wheel and moving down.



Show in Window: it is used to determine the position that shall be partially enlarged. You can select a rectangular window by dragging with the mouse to determine the position. The system will automatically display the content in the selected window.



Move Screen: it is used to move the graph by dragging it directly. This function can be also achieved by pressing and holding the right mouse button and moving the mouse.



3D/Display: it is used to display the view in frame or solid.




Rotate Graph: the graph will rotate dynamically around the pipe center to show the full view for users. In order to observe the graph dynamically in all directions, you can also press and hold the middle mouse button and move it on the preview graph.

VIII. Set system parameters

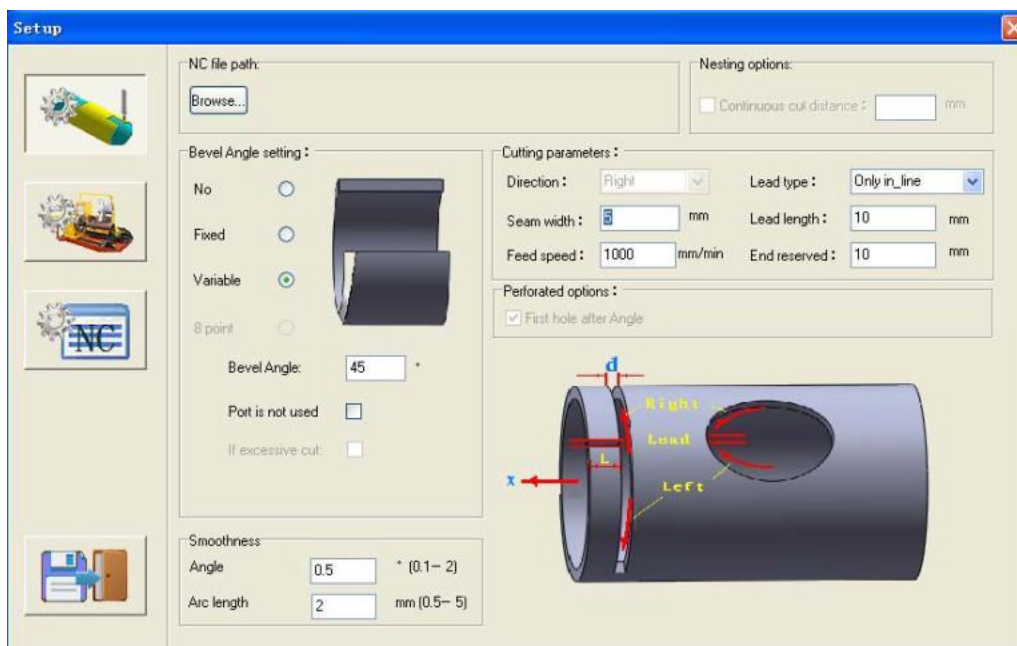
It is mainly intended to set the software's application environment, including setting parameters such as NC cutting machine, NC code format and automatic machining path, and supporting users' machine and control machine.



Click the  tool on main toolbar, and the system will open the dialog box for parameter setting.

Tips: you'd better click "Save" to save the modified setting to system profile before parts design. If you modify any setting during parts design, the simulation interface will not update the modification. In this case, you have to return to the previous step and enter the simulation interface again.

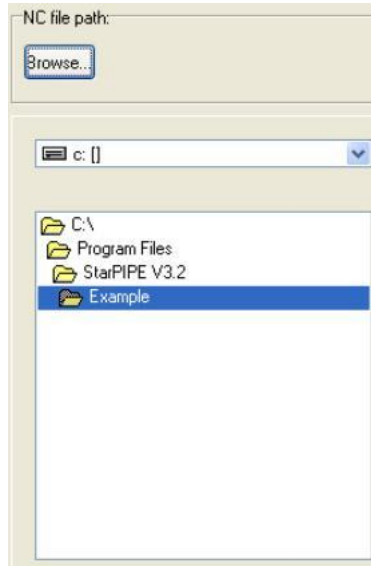
1. Set parameters of automatic path



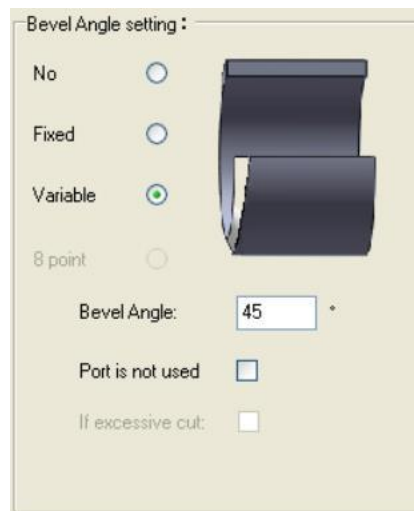
This function is mainly intended to help users set cutting parameters during cutting, ensure the generated cutting code adapt to users' cutting control system, and guarantee the accuracy of code.

1) Saving path of NC file:

To set the position where the generated NC file is saved, click "Browse", and the dialog box for selecting directory (as shown in the picture below) appears. You can select the saving path of NC file in the picture below.



2) Set groove



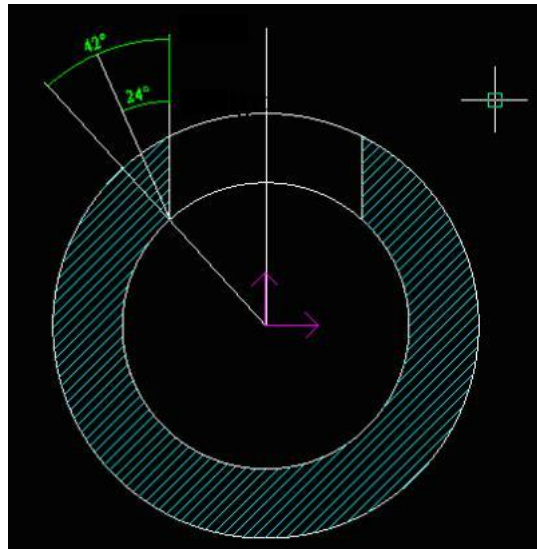
Category of groove:

No groove: no welding groove is generated in cutting code, but the actual intersecting angle of pipe fitting can be generated. The intersecting angle is either positive or negative. For example, the picture below shows a -42° angle.

Fixed groove: the groove angle of all machining lines in cutting code is identical; this function is not available currently.

Changing groove: the groove angle generated in cutting code is the algebraic sum of the intersecting angle of parts and users' welding groove. The picture below is the schematic

diagram of intersecting angle and welding groove. It can be seen that there are both positive number and negative number, so the final groove angle is the algebraic sum of intersecting angle and machining groove. In the picture, the intersecting angle is -42° , and the welding groove is set at 24° . The actual groove angle generated is the algebraic sum of the two factors, i.e. $24^\circ - 42^\circ = -18^\circ$. It should be noted that the actual groove angle generated shall not exceed the maximum positive or negative angle since the groove angle of machine tool has its limit (e.g. $\pm 45^\circ$); otherwise, the maximum positive or negative angle will be regarded as the groove angle.



Smoothness of output: users can adjust the smoothness of output according to the diameter of the pipe fitting to be machined, so as to ensure the linear velocity of machining is balanced and the cutting face is smooth. It can be set in corner and arc length. The larger the diameter is, the smaller the smoothness is.

Width of slit

The software does not support G-code profile compensation. Users need to set the width of slit. When generating NC code, the system will compensate cutting pipe for its overall size automatically, so as to guarantee the accuracy of part size. The width of slit can be given according to users' experience in cutting and the parameters such as thickness of pipe, material, type of cutting torch and pipe material.

Linear velocity of machining

The linear velocity of machining (i.e. the cutting speed for linear feeding in the case of no groove) can be given according to users' experience in cutting and the parameters such as thickness of pipe, material, type of cutting torch and pipe material. During 3D cutting, the system will set the actual feeding speed relative to linear velocity automatically based on factors such as groove angle, machining direction and cutting thickness.

Reserved position from the outermost end

The default starting position of cutting is determined by users' tool setting point. If it is necessary to reserve a certain length at the end of pipe fitting, you can set at "Reserved

position from the outermost end". The reserved length by default is 10mm.

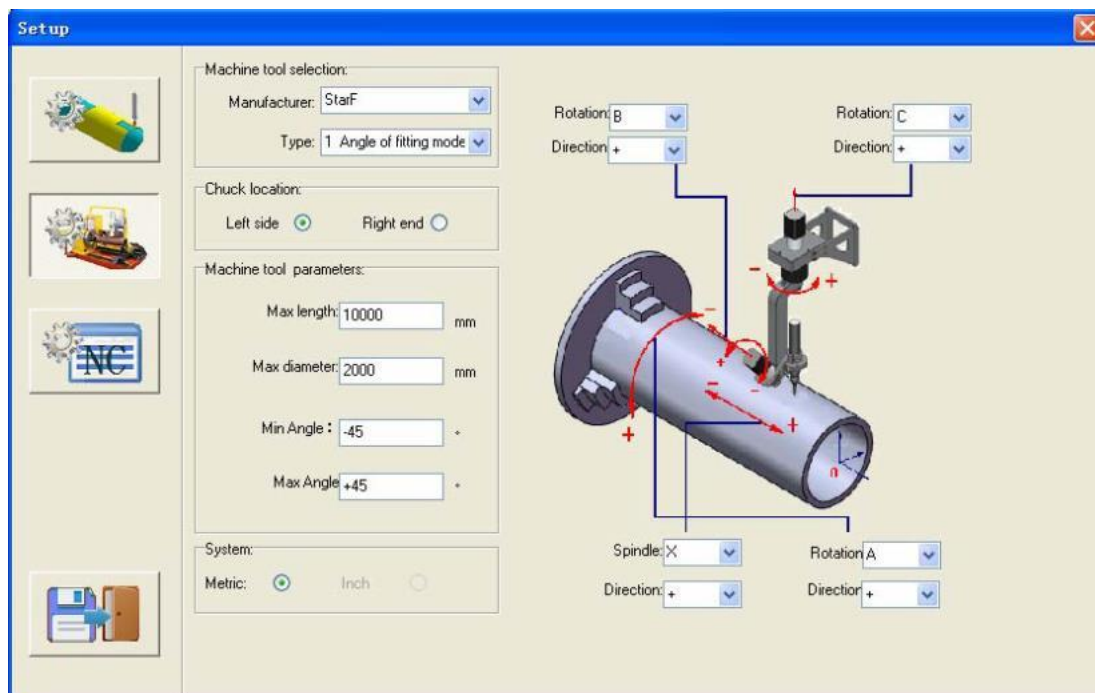
Length of lead

Set the length of lead for cutting to prevent the cutting damage to parts at cutting point during cutting and punching. Appropriate length of lead can be set according to thickness of pipe and initial angle of groove. The default angle is in the direction of X-axis. The arc lead-in is not supported currently.

2. Set parameters of machine tool

It is intended to set the corresponding cutting parameters according to the type of users' intersecting line cutting machine. It can be set according to the cutting machine tool and type of control machine during the initial use of software.

Note: modifying the parameters of machine tool inaccurately may lead to wrong cutting and waste of pipe, so the parameters must be modified carefully. Please consult professionals for help if you have any question or doubt regarding the modification.



Manufacturer of control machine: it is used to select the name of manufacturer of control machine adopted for users' cutting machine (e.g. StarF: the default name is Beijing Starfire Control System).

Type of swinging gun: it is used to select the type of swinging gun at the groove of cutting torch adopted by users' cutting machine. Available options include bi-axial mode, parallelogram mode and fitting angle mode. For details of type of swinging gun at groove, please refer to **Appendix I** attached to the User Manual.

Position of chunk: it is used to set whether the chunk is in the direction of operator's left hand or right hand when the operator faces the machine tool.

Machining parameter of machine tool: it is used to set the extreme limit of machining length, machining pipe diameter and groove angle for the cutting machine tool. When the parts to be

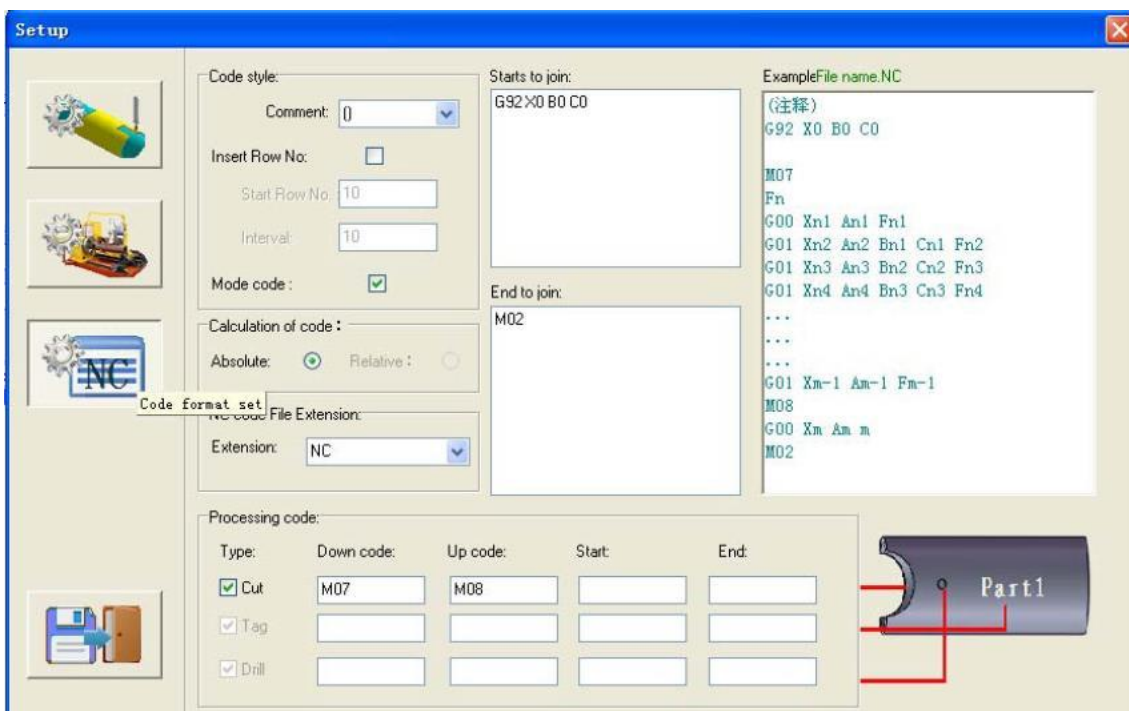
Number system: it determines whether parts size and number of NC code adopt metric units or British units. The system default is metric units. Output in British units is not supported.

Set machining axis: relevant parameters include coordinates of principal axis, feeding direction of principal axis, rotation identification name and rotation direction of principal axis; the other two coordinates are the name of rotation axis and rotation direction of groove cutting head in the parallelogram mode and fitting angle mode. Whether the direction is positive or negative is determined by the right-handed coordinate system. Specifically, the direction that satisfies the right-handed coordinate system is positive; otherwise, it is negative. For different cutting machines, the number of coordinate axis and rotation axis, identification name and direction may vary. Thus, please set the corresponding parameters according to the coordinate of cutting machine. In the picture above, X-axis is the principal axis, the main rotation axis is A, the rotation direction is positive (satisfying the right-handed coordinate system), the two rotation axes of swinging gun are B and C, and their rotation directions are both positive (satisfying the right-handed coordinate system).

3. Set parameters of code

Users should set the parameters of code in the NC code format supported by their machine tool.

Notice: The code parameters should be set by professionals, and carefully reviewed before machining to prevent the occurrence of cutting error.



Comment: it refers to the identifier of non-executable statement in NC code, which is used to add identifications in NC code, such as name of machining task, parameters and specification of raw material, instruction and other contents.

Add line number: it is used to meet the demand of adding line number for the line of code of

Code coordinates: there are two options – “absolute coordinates” and “relative coordinates”. It determines whether the generated NC code adopts absolute coordinates or relative coordinates.


Suffix of NC code file: it is used to set the extension of the output NC code file, so different users can adopt different extensions.

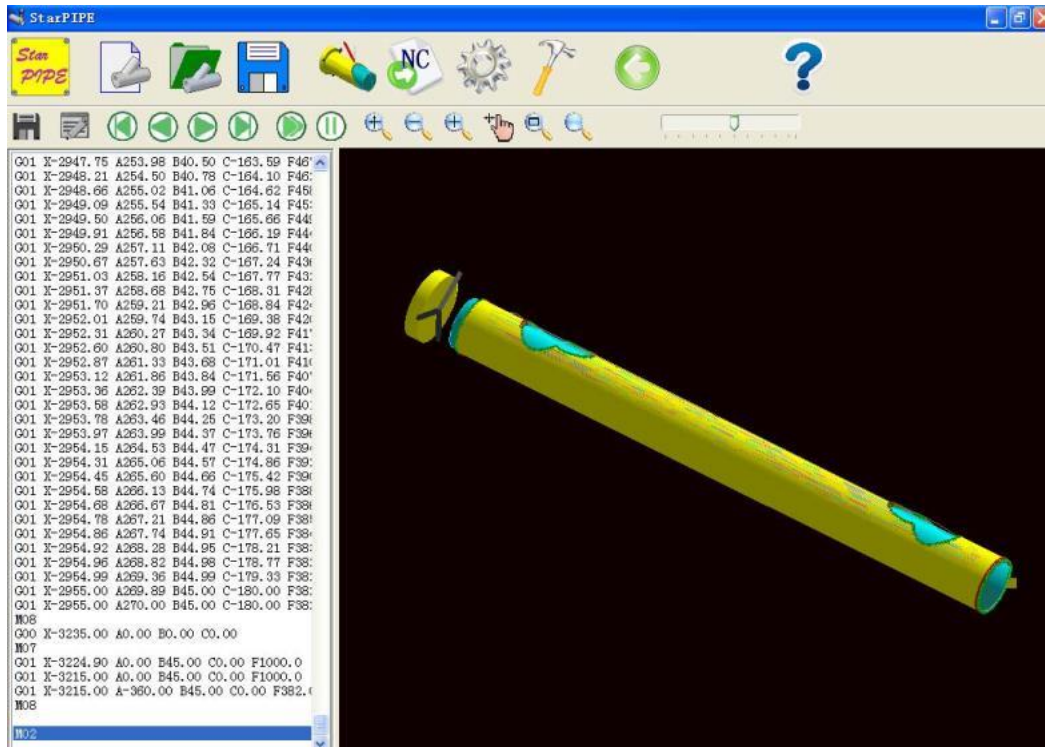
Add at the beginning of program: it is used to set the content of line of code which is added at the beginning of NC program by default, e.g. G92X0B0C0.

Add at the end of program: it is used to set the content of line of code which is added at the end of NC program by default, e.g. M02.

Machining code including: only the option “cutting” is supported currently. Users can fill the code format when the system’s cutting torch igniter falls and lifts in the subsequent cutter setting and cutter lifting codes. If the cutter setting and cutter lifting codes consist of multiple M codes, they can be separated by comma.

IX. Simulation machining

Click the  tool on main toolbar to conduct simulation machining for the generated NC code or the current machining task, so as to verify the accuracy of machining code. In addition, the NC code generated by the third party can be also opened for editing and simulation (in the case of compatible code format).



It is used to open the NC code file. Click the button, and the system displays the dialog box

for opening file. Select the desired NC code file, and click “Open”.



It is used to save the current NC code. Click the button, and the dialog box for saving NC code appears. Click “Save” to save the NC code, or rename and save it.



It is used to edit the content of NC code. Click the button, and the interface for editing code appears. Modify the NC code as required, and click “Save” to save the NC code you just edited. Exit from editing, and the machining track diagram modified will be displayed in the corresponding track simulation area.



Let the pointer of NC code point to the first line of NC code.



Let the pointer of NC code point to the end line of NC code.



Let the pointer of NC code move up by one working step. Meanwhile, the track simulation area displays the graph of the previous working step.



Let the pointer of NC code move down by one working step. Meanwhile, the track simulation area displays the graph of the next working step.



Start continuous simulation of working steps according to automatic simulation interval set by users.



Pause continuous simulation of working steps.



Display the cutting track in full screen in the track simulation area.



Shrink the graph in the track simulation area by 0.8 times (80%).



Enlarge the graph in the track simulation area by 0.8 times (80%).



Move the machining track diagram in the track simulation area by using the mouse.




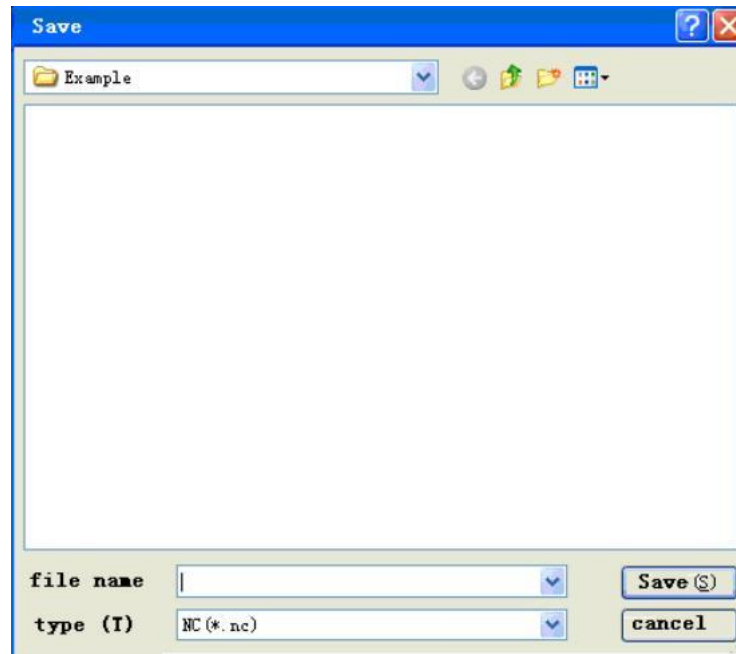
It is used to determine the position that shall be partially enlarged. You can select a rectangular window by dragging with the mouse to determine the position. The system will automatically display the content in the selected window.



Adjust simulation speed. The speed of simulation machining can be changed by dragging the progress bar.

X. Output NC code

Click the  button on the main toolbar. The system will generate the NC program corresponding to the current machining parts of pipe based on the parameters set by users. A dialog box for saving file will be then opened, asking you to give the location and type in which the NC code file should be saved, as shown in the picture below.



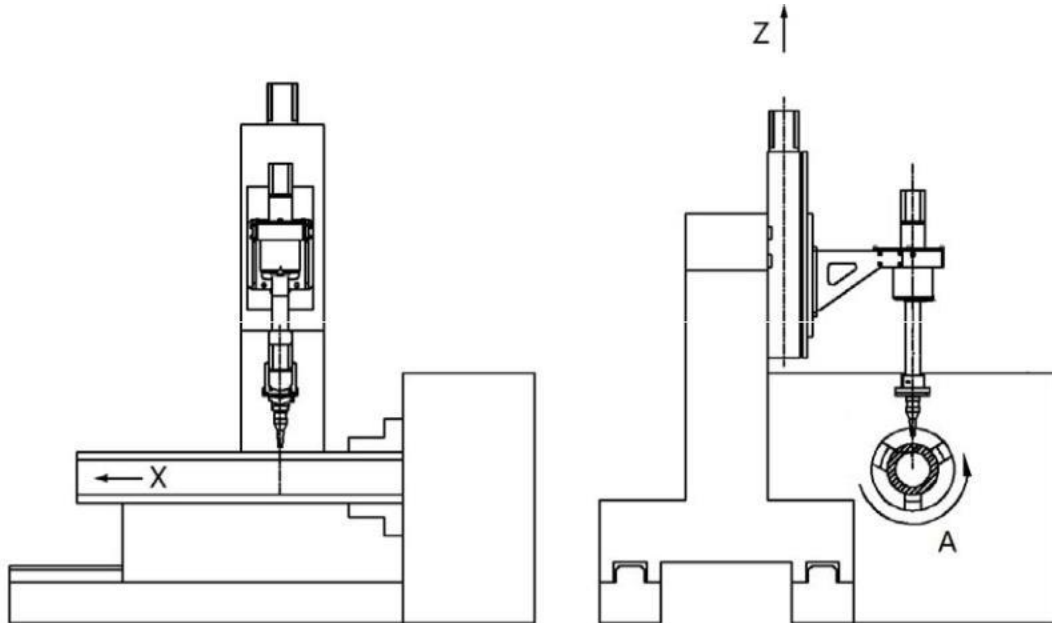
XI. Tool

It intends to provide gadgets for the system. (To be added in future)

Appendix I: Three basic modes of intersecting line machine tool

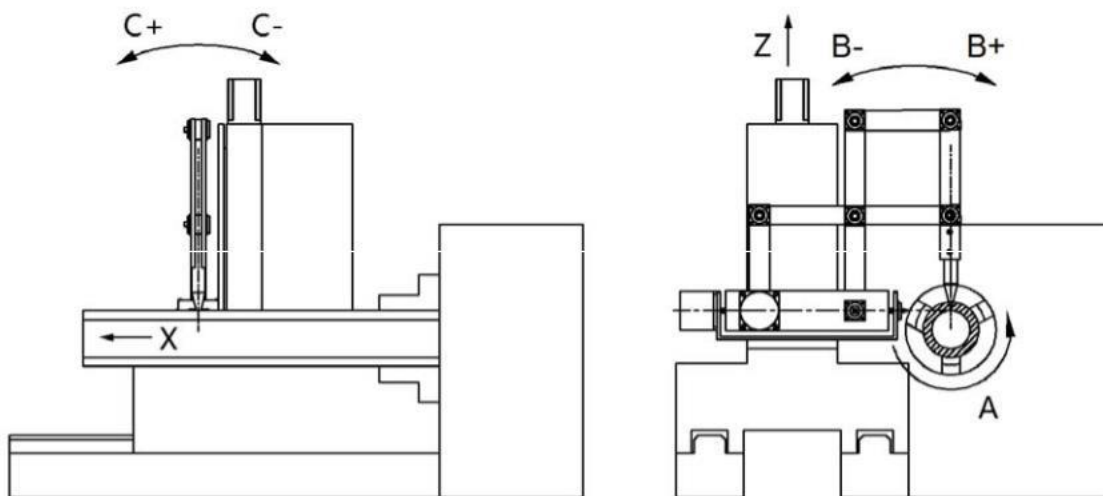
1. Bi-axial machining mode (groove machining is not supported)

In this mode, the structure has two machining axes – X axis (translation) and A axis (axis of rotation, steel pipe). The cutting torch for machining is vertical, so the groove cannot be machined (the item for setting groove is unavailable). It only guarantees the size of inner profile of pipe fitting during cutting, so excessive cutting of outer profile may occur.



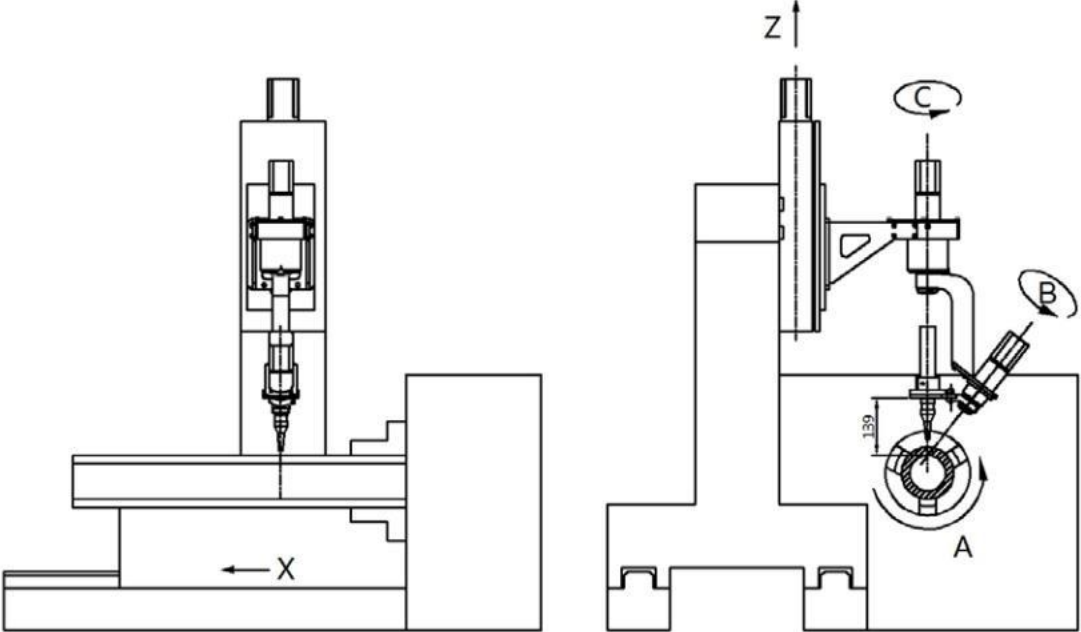
2. The swinging gun has a parallelogram structure (parallelogram mode)

The structure consists of four axes – X axis (translation), A axis (axis of rotation, steel pipe), B axis (for swinging gun) and C axis (for swinging gun). The gun moves along with workpiece during machining. Both groove and intersecting angle can be machined.

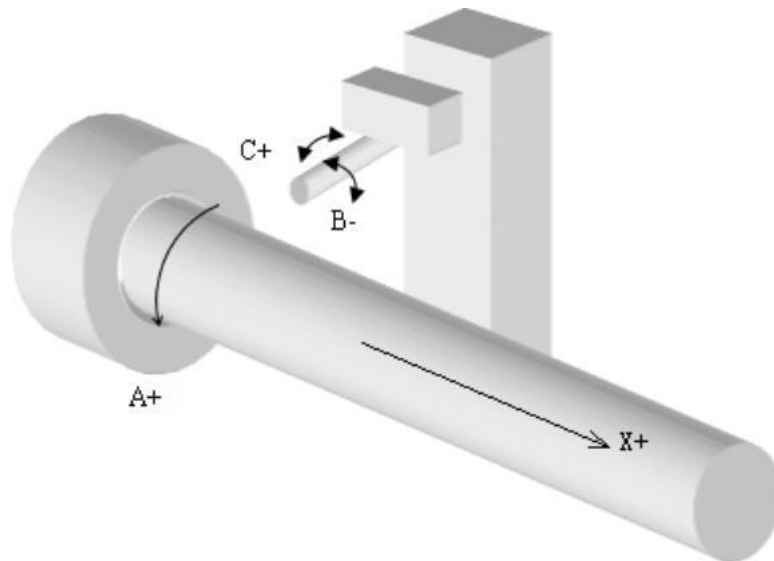


3. The swinging gun has a structure with infinite swiveling head (fitting angle mode)

The structure consists of four axes – X axis (translation), A axis (axis of rotation, steel pipe), B axis (for swinging gun) and C axis (for swinging gun). The gun moves along with workpiece during machining. Both groove and intersecting angle can be machined.



Appendix II: Reference for coordinate system

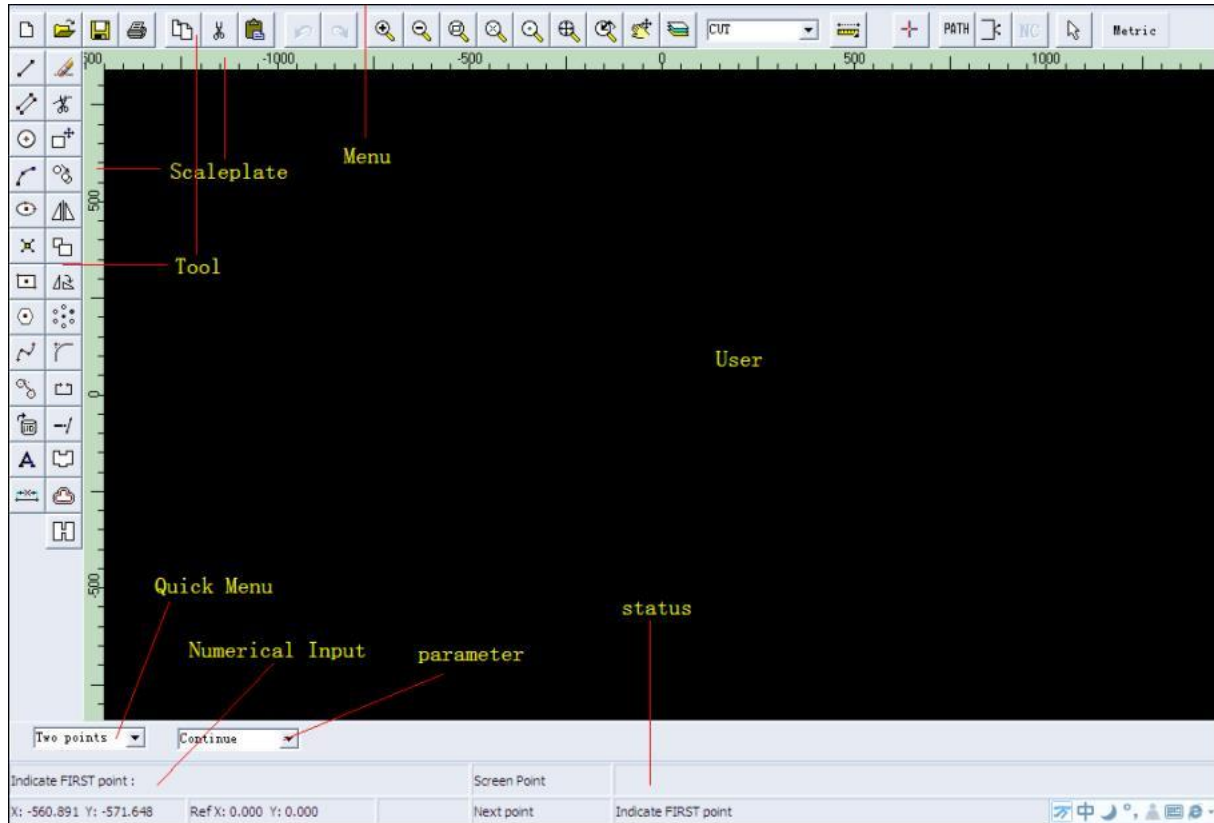


Instructions of direction:


1. X axis is a translation axis. The direction far away from chuck is the positive direction of X axis, while the direction close to chuck is the negative direction.
2. A axis is the rotation axis around X axis. When you view it from the positive direction of X axis to its negative direction, the counterclockwise direction is the positive direction of A axis, while the clockwise direction is the negative direction.
3. B axis is the axis swinging left and right. When you view it from the direction facing cutting gun, clockwise swing is defined as positive direction (B+), while counterclockwise swing is defined as negative direction.
4. C axis is the axis swinging forth and back. When you view it from the direction facing cutting gun, counterclockwise swing is defined as positive direction (C+), while clockwise swing is defined as negative direction.

Appendix III: Introduction to drawing module

The module is a simplified version of StarCAD in our StarCAM series. With this module, you can add text and graphic cutting features to pipe fitting. Only the operation method related to pipe cutting is described briefly in this appendix. The operation interface is as follows.

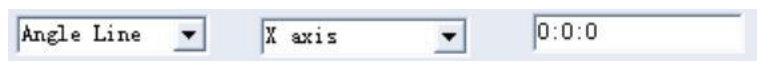


1. Basic operation

- 1) Operation of tool buttons: you can click any tool button to select the corresponding operation as required. If you want to draw a straight line, you can choose the tool button of straight line ().
- 2) Select type of operation: you can select the type of the desired operation in Quick Menu Area. For example, the options for straight line include two-point line, angle line, perpendicular line, etc.



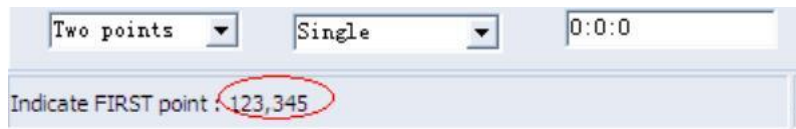
- 3) Provide parameters for drawing solids: specifically, provide geometric parameters of solids during drawing, such as coordinate point, direction and value.



At last, select the reference point for graph drawing in drawing area to complete the operation.

- 4) Input mode of coordinate point:

Direct input by keyboard: when it is required to input coordinate value in prompt area, you can input coordinate value directly by using keyboard. The coordinate value will be displayed on the left side of numerical area (as shown in the picture below).

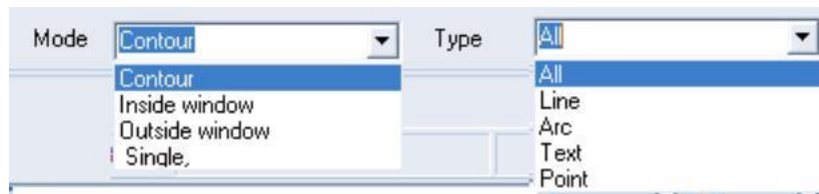


Input in parameter interaction box: click the textbox for inputting parameters, and the numerical input box appears. Input parameter values in the box. Click “√” to confirm your input, (as shown in the picture below).



5) Select and pick solids

Prior to all editing and modification operations, you must pick solids or set of solids first. The standard picking method is as follows:



To cancel the picking operation, you can simply click the right mouse button or select other operations directly.

2. Introduction to drawing tool



Straight Line Tool: the straight line can be drawn in multiple forms, such as two-point line, parallel line, angle line, angle bisector and angle tangent. You can select the forms in Quick Toolbar.



Angle line: it is used to draw a line at an angle with respect to X axis and Y axis, or a line that forms an angle with respect to the other line.

Angle bisector: it is used to draw a bisector of angle of intersection formed by two intersecting lines. The direction of lines is determined by the position of the lines you select.

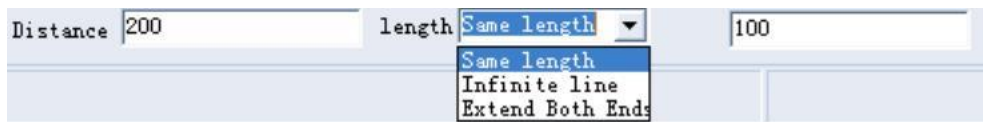
Angle tangent: it is used to draw a line that is tangent to a circle and forms an angle with respect to the other line.

Perpendicular line: it is used to draw a line perpendicular to the other solid.



Parallel Line Tool: it is used to draw parallel line with equal length, extended parallel line and

parallel construction line.

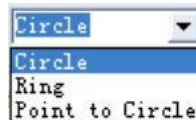


The same as original solid: it is used to draw parallel line whose length is equal to reference solid.

Construction line with infinite length: it is used to draw construction line with infinite length which is parallel to reference solid.



Circle Tool: it is used to draw a circle by given center and radius and a concentric ring.

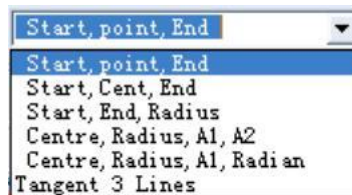


Circle: it is used to draw a circle by given center and radius.

Ring: it is used to draw a concentric ring with given inside radius and outside radius.



Arc Tool: it provides multiple methods to draw arc.



Origin, point, end point (three-point arc): it is used to draw an arc with the known origin, point and end point.

Origin, center of circle, end point: it is used to draw an arc with the known origin, center of circle and end point.

Origin, end point, radius: it is used to draw an arc with the known origin, end point and radius.

Center of circle, radius, start-stop angle: it is used to draw an arc with the known center of circle, radius, start angle and stop angle.

Center of circle, radius, start angle, radian: it is used to draw an arc with the known center of circle, radius, start angle and radian.

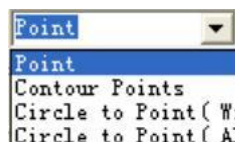
Tangent to 3 lines: it is used to draw an arc which is tangent to three known lines.



Ellipse Tool: it is used to draw an ellipse with the known major and minor axes.



Point Tool: solid point can be used as the reference point for positioning during solids drawing.

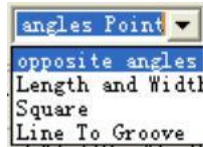


Arbitrary point: it is used to draw a point in the position of the coordinate point given by users or the screen capture point.

Bisecting point of profile: it is used to draw equally spaced points with equal number according to the parameters given by users.

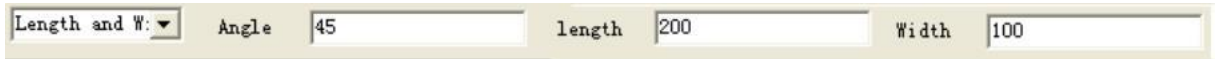


Rectangle Tool: it is used to draw the rectangle and square selected by users.



Diagonal point: it is used to draw a rectangle with the known coordinates of any two diagonal points.

Length and width: it is used to draw a rectangle with the known length, width and rotation angle.



Square: it is used to draw a square with the known length of side and rotation angle.

Line to slot: it is used to change the drawn line to the slot with designated width.



Polygon Tool: it is used to draw a solid polygon with the known number of side, length of side and rotation angle.



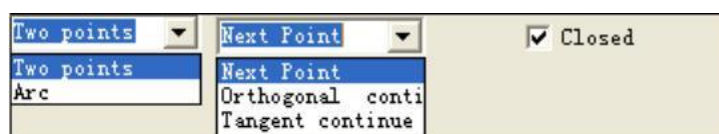
Spline Tool: it is used to draw the fitting spline with the known coordinate points according to the fitting precision given by users.



Transition Arc Tool: it is used to draw the transition arc (arc and arc, line and arc, line and line) between two solids with the given radius.



Outline: it is used to draw the outline composed of line and arc rapidly, similar to polyline in popular graphics software.

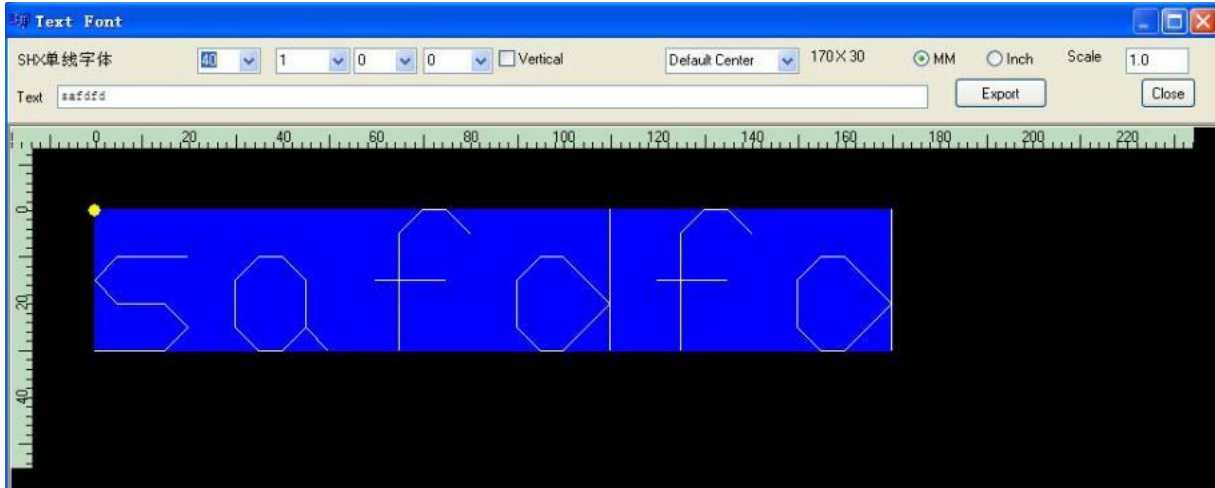


Lines: it is used to draw lines continuously, where:

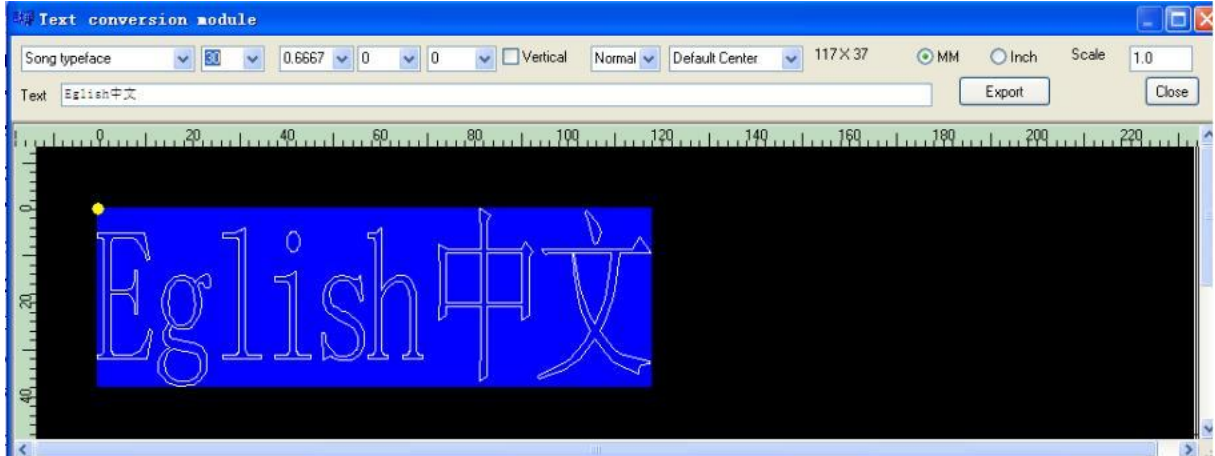
Arcs: it is used to draw arcs between any two points continuously.



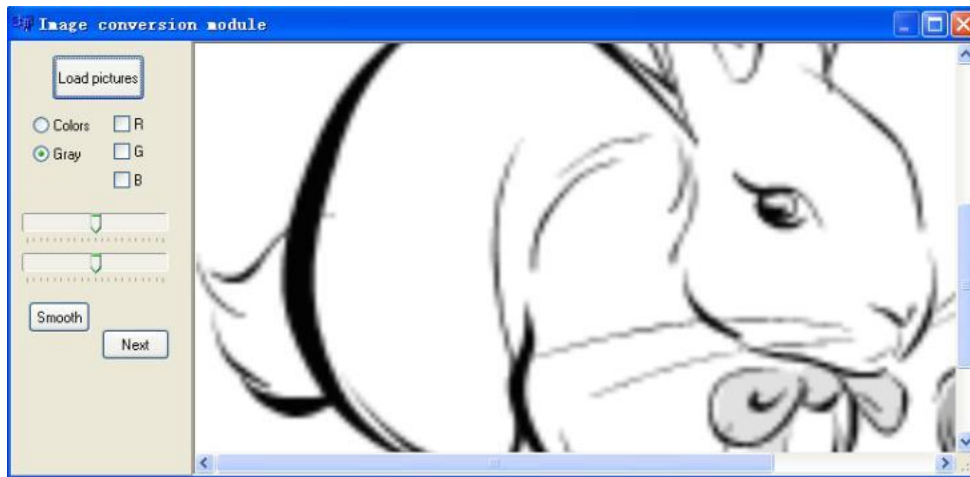
Single-line style marking tool: it is used to mark single-line style words of graph, especially for dusting marking and machining on marking layer. Chinese and English text is both available during marking.



Text cutting tool: it is used for graphic transformation and machining of Chinese and English characters supported by the operation system.



Graph cutting tool: it is used to generate graphs by using common graphics files (BMP, JPEG, etc.) according to the parameters selected by users.



Click “Import graph” to import the picture file supported by system. The picture will be displayed in gray by default. Click “Smooth” several times to adjust the definition of edge transition of graph. Slider can be used to adjust brightness and contrast.

Click “Next” to enter the screen below. You can drag the number bar of binarization threshold to change the range for reserving and deleting in the graph.



Click “Next” to enter the interface of pattern formation (as below). The graphical tools on its top are used for local trimming of graph (supplement lines, fill foreground color and fill background color). “Extract Profile” is used to generate the outline of black profile in the graph. The “Display Picture” selection box can close graphics display to only show the generated outline. The dropdown list is used to select the fineness of generated graph. The dragging box is used to determine the filtering value in minimal area. “Proportion” determines the amplification factor of generated graph. “Output” is used to insert the generated graph into the position of insertion point in the current drawing area. You can choose to output in metric system (mm) or in British system (inch). To return to the first picture for restarting, click “New”. To return to the previous page and adjust binarization threshold, click “Back”.



3. Edit and modify

As a fundamental function of interactive drawing software, it is intended to edit and modify the current graph.



Delete Tool: it is used to delete single or multiple solid objects rapidly.

Note: if some solid is deleted incorrectly, you can “Undo” it to restore the deleted solid.



Smart Trimming Tool: it is used to trim and delete excessive lines of single solid or intersecting solids rapidly. The system will automatically calculate the cross point and trim excessive lines at the cross point in the position selected by users. It is the most useful editing tool during editing and modification.



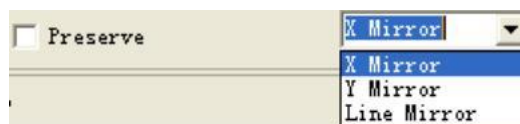
Move Tool: it is used to move the solid selected by users to the coordinate position specified by users. It is a basic method for graphic positioning.



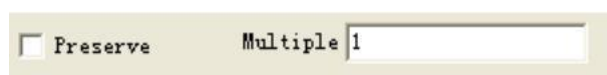
Copy Tool: it is used to copy the solid selected by users to the position specified by users, without deleting the original graph.



Mirror Tool: it is used to mirror or copy symmetrically the selected solid by regarding a line as the axis of symmetry.



Scale Tool: it is used to scale the selected solid in accordance with the given factor of proportionality and reference point.

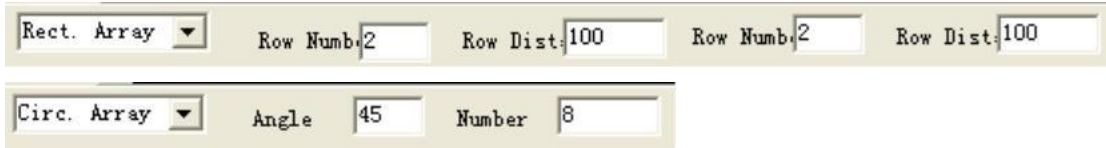




Rotation Tool: it is used to rotate the selected solid in accordance with the given angle and reference point.



Array Tool: it is used to create circular array or rectangular array of the selected solid in accordance with the selected array type and given array parameters. It helps generate the parts graph with multiple similar solid profiles rapidly.



Angle Transition: it is used for the transition of angle formed by lines from right angle, fillet, corner cut, fillet and round to sharp corner.



Breaking Tool: it is used to break the specified solid.

Steps: click “Break” on the “Modification Tool” toolbar, or choose the “Break” menu under the “Modify” menu. Select the solid to be broken as required by the system, and then provide the coordinate position to be broken.



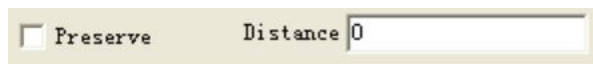
Edging Tool: it is used to extend or shorten the specified solid based on the reference solid.



Convex-concave Cut Tool: it is used to change the specified line to a semicircle or rectangle with convex or concave structure.



Profile Scaling Tool: it is used to enlarge or shrink the specified outline.



Bridging Tool: it is used to bridge the two part profiles in accordance with the specified width. It mainly applies to integrating multiple part profiles, reducing the number of cutting and punching, and lowering the loss of cutting torch during parts cutting.

Solid bridge: in this mode, you can click the adjacent solid edges for bridging.

Bridge by drawing lines: in this mode, you can pull a line that passes multiple solids by using the mouse, so as to achieve multiple zeroes.

Sales and Technical Support

Beijing Starfire Control Technology Co.,Ltd

Address: Shijingshan district of Beijing haite garden 44 floor. Room 204

ZIP:100041

TEL: 86-10-88909779

FAX: 86-10-88909779

WeChat: 18901200180

Web site: <http://www.starfenc.com>