BEND-TECH
DRAGON MACHINES

Dragon CAM Operator's Manual
Part 2: Dragon CAM Overview
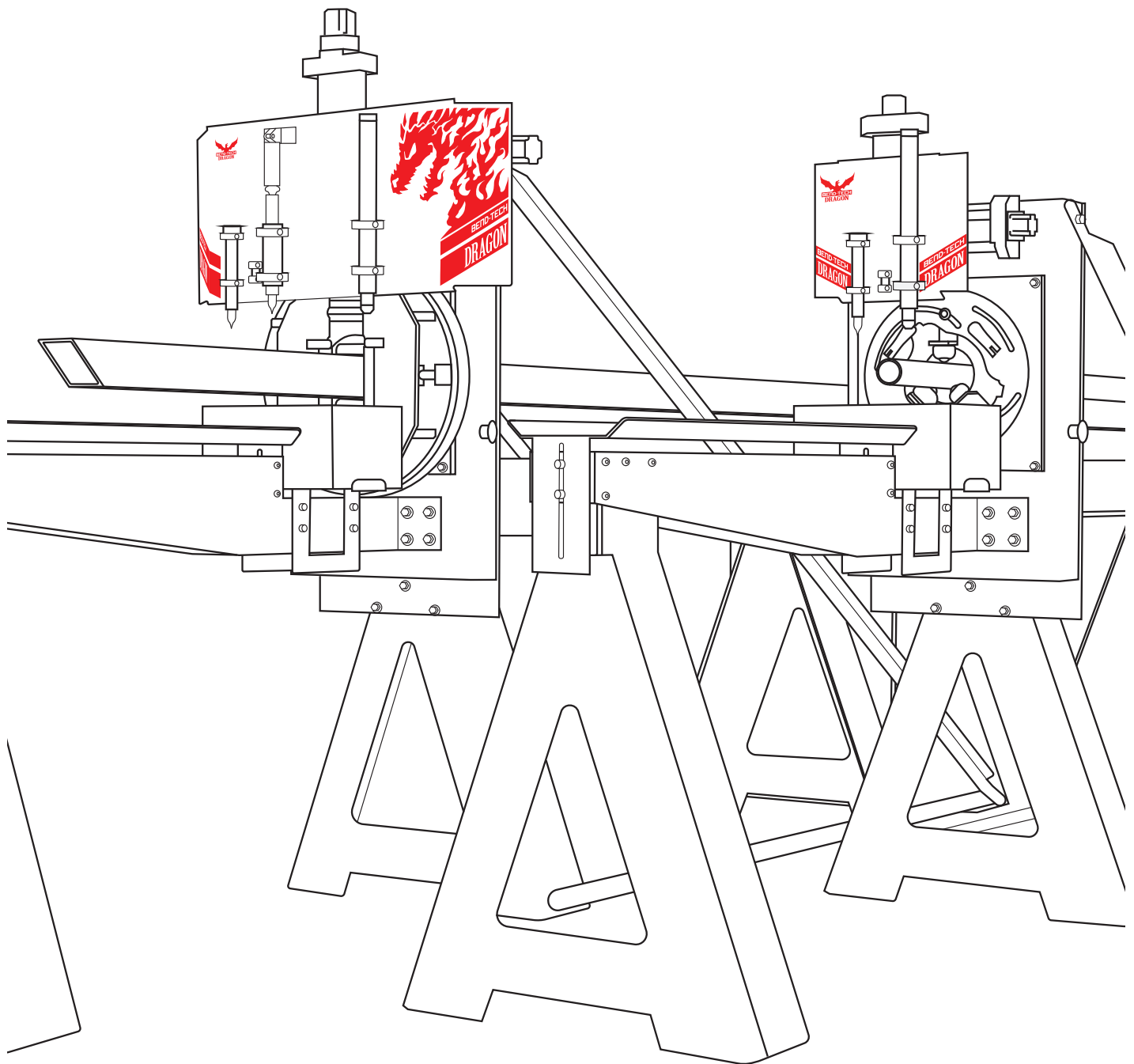
Dragon Machines

Operator's Manual
Revision 004

English
Original Instructions

October 2021

Contents

Contents

01

Create Parts

1.1 Create Straight Part

To create a new straight part, choose STRAIGHT from the Dragon CAM Task Menu. This will open the Create New Part interface. This section will discuss the options and settings available when creating a part. For a quick tutorial for creating a part see Section 2.1 of the Operator's Manual Part 03.

1.1.1 Die & Material Tab

The material is selected from the Die & Material tab. Any material used to create parts must be added to the Tube / Pipe Library before making new parts. Enter a PART NAME and set the TUBE LENGTH. This tube length value sets the length of the part not the stock length. Once the length is entered the part will appear in the 3D Design Window.
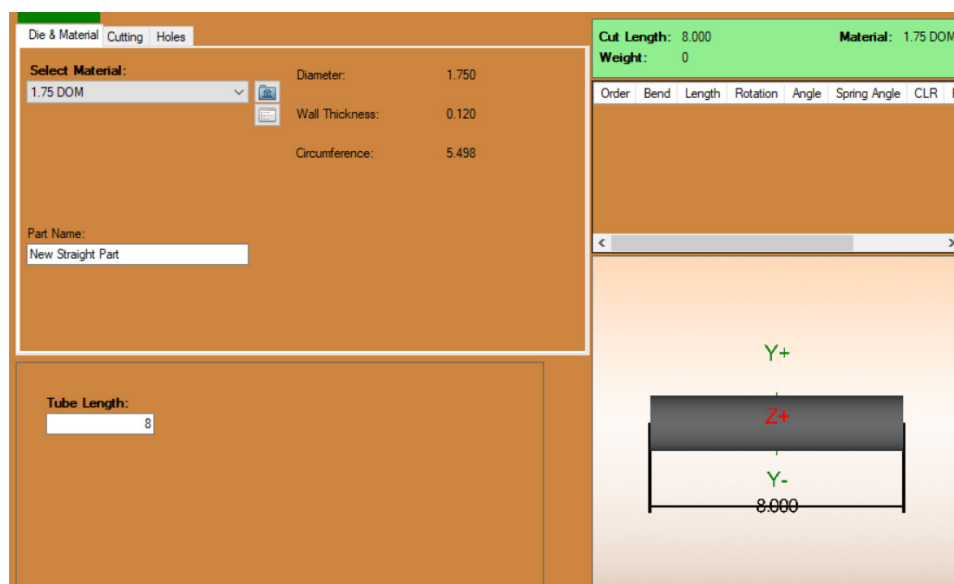
1.1.2 3D Interface Control

Once the part is visible in the 3D design window, the operator can use the mouse to rotate the part, zoom in / zoom out, or drag the entire part by holding down the scroll wheel down.

3D Interface Control	
Zoom	Scroll in/out
Drag	Hold down scroll wheel, or click and hold both mouse buttons
Rotate	Hold down right button
Reposition	Click the Home button

1.1.3 Cutting Tab

The Cutting interface is used to add start and end cuts to the material. Clicking the ADD button will add a start cut to the part. This cut defaults to a cope. To change this cut to a miter, click the TYPE button. To change the location of this cut, click the LOCATION button. Click the ADD button a second time to add a cope cut to the opposite end of the part.
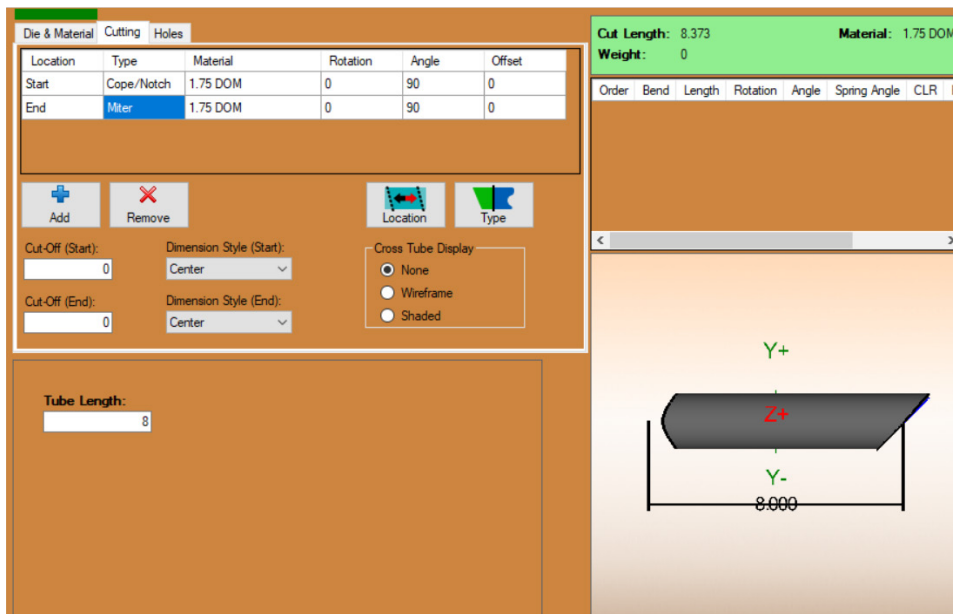
Tip

Double-clicking the Type or Location cell from the Cutting List will change the location or type of the cut.

To change the size of material the part copes to, click the Material cell from the Cutting List, and select the material from the Tube / Pipe Library. The material being coped to will need to be added to the Tube / Pipe Library before it will show up in the list. This option does not apply to miter cuts.

The Rotation property moves the cut around the circumference of the material. The Angle property changes the angle of the cut. The Offset property changes the offset of the cuts in relation to the center line of the part. These can be adjusted as needed from the Cutting List.

<div style="background-color:red; color:white; text-align:center;">

Important

The Offset Value must not be larger than the overall diameter of the material.

</div>

1.1.4 Cross Tube Display Options

There are two Cross Tube display options for the 3D Display Window. Choose between Wire Frame display or Shaded display to visualize how the part works with cross tubes. Choose None for part display only, with no cross tube visualization.

1.1.5 Hole Tab

The Holes interface is used to add holes of various sizes and shapes to the part. To add a new hole to the part, click the ADD button. This will open a popup window with a drop down menu. Select the shape of the hole from the drop down menu and click OK.

Triangle Holes

Triangle Holes will open an additional interface. The required dimensions for Triangle Holes are any combination of three values as seen in the diagram. The base point will also need to be selected. Once the information is added, click OK. This will add the hole to the part, and the other dimensions can be edited as usual.

1.1.6 Hole Dimensions

There are several dimension options available to customize the holes.

<div style="background: red; color: white; text-align: center;">
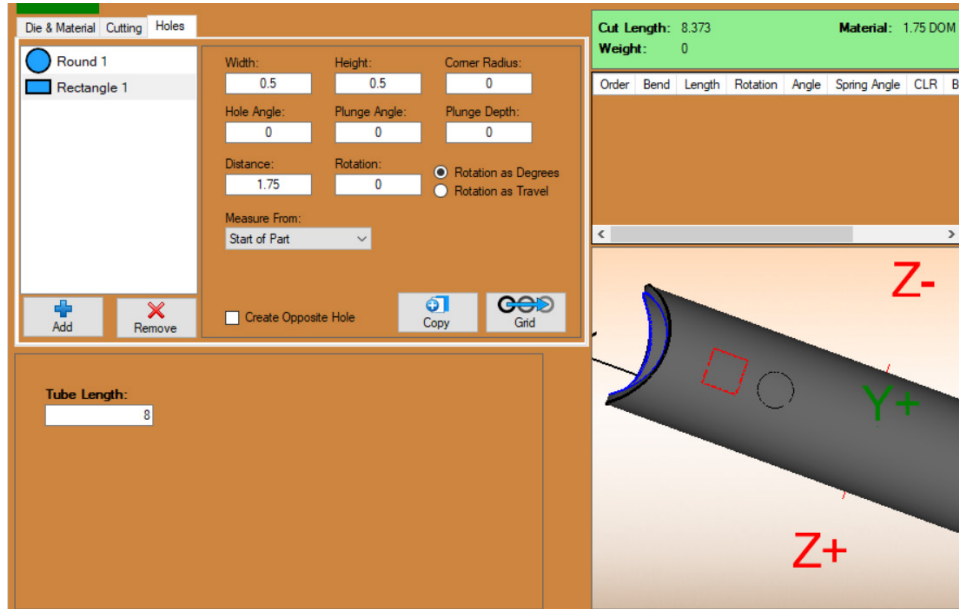
Important

</div>

If the hole is too big for the material, the software will shade all text boxes red until the hole size is corrected.

Width

WIDTH adjusts the width of the hole. This is a required dimension for all hole shapes.

Height

HEIGHT adjusts the height of the hole. This is required for Rectangle, Oval, and Ellipse holes.

Corner Radius

The CORNER RADIUS adjusts how round the corners of the hole are. This is as option for Square and Rectangle holes only.

Hole Angle

The HOLE ANGLE is the angle of the hole in relation to the length of the part. This option is not used for Round holes.

Plunge Angle

The PLUNGE ANGLE is the angle of the hole relative to the wall thickness of the part. This can be thought of as the angle of the object that would create the hole by passing through the part.

Plunge Depth

The PLUNGE ANGLE is an optional dimension. Enter a positive value to have the hole plunge the specified distance into the material surface. Leave at zero to ignore this feature.

Distance

DISTANCE adjusts the location of the hole. By default, This is measured from the start of the part/leg, to the center of the hole.

Measure From

There are four options that can be used to measure the distance of the hole from. By default, holes are measured from the start of the part. From the MEASURED FROM drop down menu, other options can be selected. These options include Start of Part, End of Part, Center of Start Cut, and Center of End Cut.

Rotation

ROTATION setting is the rotation around the part to the center of the hole. Depending on whether, Rotation as Degrees or Rotation as Travel is selected, this value will be either in degrees or distance.

1.1.7 Create Opposite Hole

By checking the CREATE OPPOSITE HOLE box, a second hole will be created on the opposite side of the material from the selected hole.

1.1.8 Copy and Grid

To copy a selected hole, click the COPY button. This will add a duplicate hole that can be adjusted as needed. To add a series of holes or a grid of holes to the part, click the GRID button. This will open a popup window where the grid settings can be added.

Number of Columns

The NUMBER OF COLUMNS value determines the number of holes needed along the length of the part.

Number of Rows

The NUMBER OF ROWS value determines the number of holes needed around the circumference or perimeter of the part.

Spacing Between Columns

The SPACE BETWEEN COLUMN and ROWS is the distance between each hole. This is measured from the center of one hole, to the center of the next. This can be measured in distance or degrees, as selected in the ROW SPACING options.

> ## Note
>
> *When deleting holes created by the Grid or Copy function, each individual hole will need to be deleted one at a time.*

1.1.9 Finishing the Part

Once the part is designed, it can be sent to EDIT FLAT for adding custom geometry. If no other edits are needed and it is ready to be cut, it can be sent to NEST PART.

1.2 Creating a Bent Part

To create a new bent part, choose BENT from the Dragon CAM Task Menu. This will open the Create New Part interface. This section will discuss the options and settings available when creating a bent part. For a quick tutorial for creating a part see Section 2.2 of the Operator's Manual Part 03.

1.2.1 Die & Material Tab

Select the Material and Die needed for the new Bent Part. Both the Material and the Die must be added to their respective libraries before they can be selected and used. Change the PART NAME as needed. Select the NUMBER OF BENDS from the drop down menu and set the START ANGLE dimension.

The section below these settings will populate with a series of text boxes where the dimensions of the legs and bends of the part can be entered. These settings include the LENGTH of the leg, ROTATION of the bend, ANGLE of the bend, DIMENSION TYPE, and DIE.

Once these dimensions are filled out, the part will appear in the 3D Display Interface.

Dimension Type

Apex vs Tangent Dimension Type

In the design values grid, choose between two different Dimension Types as defined here:

Apex measures the outside intersection of the bend's two sides. Apex is essentially the measurement of where the material would intersect if there were not a bend.

Tangent measures the material minus the bend, or the straight runs of the material. Use Tangent when the straight lengths of the part are known.
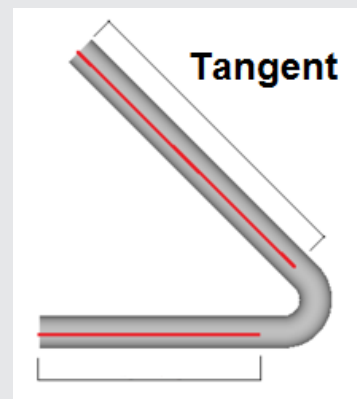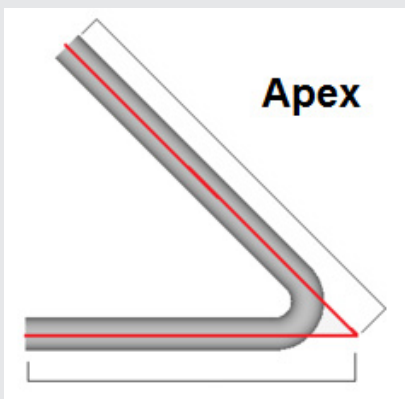
1.2.2 3D Interface Control

Once the part is visible in the 3D design window, the operator can use the mouse to rotate the part, zoom in / zoom out, or drag the part by holding the scroll wheel down.

3D Interface Control	
Zoom	Scroll in/out
Drag	Hold down scroll wheel, or click and hold both mouse buttons
Rotate	Hold down right button
Reposition	Click the Home button

1.2.3 Cutting Tab

The Cutting Tab is the same for both Bent Parts and Straight Parts. See Sections 1.1.3 & 1.1.4 for detailed information.

1.2.4 Holes Tab

The Holes tab is the same for both Bent Parts and Straight Parts. See Sections 1.1.5 - 1.1.8 for detailed information.

1.2.5 Manufacturing Warning

The Manufacturing Tab will display any errors with the Bent Part and possible actions to take to fix the issues.

1.3 Saving Parts

Parts, Edit Flat Designs, and Nesting Projects can be saved and re-opened later. Click FILE and select SAVE. Name the file and click SAVE. When closing out of the program with a Part Design, Edit Flat Design, or Nesting Project open, the software will ask if the open design needs to be saved.

1.4 Opening Saved Parts

Click BROWSE from the Task Menu to open previously saved Parts, Edit Flat Designs, and Nesting Projects. Navigate to the location of the saved file. Select the file and click OPEN. This will open the saved projects in Bend-Tech CAM.

02

Importing

2.1 Importing Files Overview

The Dragon software has a variety of options for importing design files created in other design or 3D modeling programs.

The software offers Auto Import which converts imported projects and parts into files that are ready for the Dragon software. Manual Import files typically require editing using Bend-Tech import tools.

It is possible to import a variety of different CAD files into Bend-Tech software including STEP, IGES, and DXF files. There are many different formats and idiosyncrasies to these types of files, and Bend-Tech cannot account for all scenarios. The software is designed to work with as many formats as possible, but Bend-Tech cannot guarantee 100% compatibility.

2.2 Importing Files in Dragon CAM

When opening a non-native file in Dragon CAM, an Import Options popup will appear, prompting to select MANUAL IMPORT or AUTO IMPORT. Bend-Tech recommends selecting AUTO IMPORT whenever possible. On a successful import, the file will open in the Dragon CAM Import Interface.

On a successful import, the Import Interface will display a list of all the parts in the file and a 3D rendering of the parts/assembly. The name of each part and the quantity of each part can be modified as needed for the specifications of the project. This information will carry forward to the NEST PART process.

Name	Length	Bend Count	Quantity	Material
OG MAV CAGE5	3.161	0	1	1.560" - 0.095" Round
OG MAV CAGE5	18.547	1	1	1.750" - 0.095" Round
OG MAV CAGE5	10.791	0	1	1.750" - 0.095" Round
OG MAV CAGE5	10.791	0	1	1.750" - 0.095" Round
OG MAV CAGE5	52.500	0	1	1.750" - 0.095" Round
OG MAV CAGE5	52.500	0	1	1.750" - 0.095" Round
OG MAV CAGE5	52.500	0	1	1.750" - 0.095" Round

The part(s) can be viewed and edited in Edit Flat, which can be used to further define elements of the project that may not have imported accurately. Select the parts that need editing and click EDIT FLAT from the menu.

The software will automatically attempt to match the material specified in the imported files to a material in the Bend-Tech Dragon Tube / Pipe Library. If it cannot match the material to one that's already been created in the Library, the software will create a Temporary Material. Before the project can be run, the Temporary Material will need to be converted to a permanent entry in the Tube / Pipe Library.

Temporary Material

If an imported project contains material that is not on the machine's Material List, the software will automatically enter it as a Temporary Material. Before the Nest Part process can be performed, the material will need to be updated in the Tube / Pipe Library.

When clicking Nest Part, an "Incomplete Material Error" popup warning will appear. Click OK.

To add the Temporary Material as a permanent addition to the Material List, navigate to the Tube / Pipe Library by clicking TOOLS from the Top Navigation Bar and selecting Tube / Pipe Library from the list.

Locate the Temporary Material in the Material List. Verify the settings were imported, and adjust them if necessary. Set the Lead In / Out settings appropriately.

Click SAVE to add the material to the Tube / Pipe Library. The material can now be used with the imported parts.

After importing the file, and after adding any material to the Material Library, select the part(s) that need to be produced from the list. Click NEST PART to open a Nest Part Project for the imported design. If the imported design contains multiple materials, a Nest Part Project will open for each material type.

Tip

To select multiple parts to send to either Edit Flat or Nest Part, select the first part in the list, hold down SHIFT, and click the last part in the list to select all the parts. To select only a few, hold down CTRL and click on each part to select it.

2.2.1 Undefined Import Elements

The imported files may contain elements such as different or incomplete material, or a design in a format that the Bend-Tech software does not recognize. A poor import will show up as red lines or a series of edges. Parts with undefined elements may need to be defined manually, imported separately, or recreated using the Bend-Tech software.

2.2.2 Tekla & SDS2 Plugins

Bend-Tech Tekla & SDS2 Plugins work directly with Tekla and SDS2 to import files that can be viewed and edited in Bend-Tech Dragon software. The Tekla & SDS2 Plugins default to the Auto Import feature; they do not give the choice for Manual Import. When importing with the Tekla & SDS2 Plugins, the plugin creates an .enc file that Dragon CAM uses to create the Parts List and visualize the part(s) or assembly in the 3D interface.

The file name and the number of parts can be changed depending on the project from the Parts List.

2.3 Manual Import

Manual Import can be used for STEP or IGES file imports. If an Auto Import file or series of files fail, choose Manual Import to pull the files into the Bend-Tech software. Manual Import will translate the file, find any edges within the file, and display it as simple line information according to those edges.

2.3.1 Manual Import Part Display

When Manual Import is performed, the Bend-Tech software will display the part in a Manual Import interface. This interface contains tools that are used to define the part and make it complete.
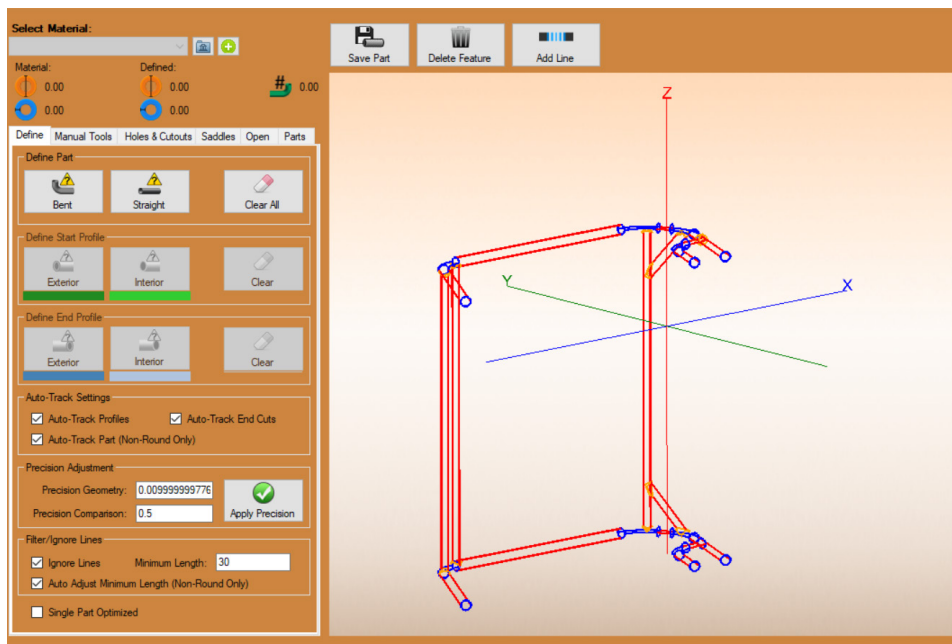
2.3.2 Part Definition

The first section of the Manual Import interface contains information about the imported parts. On import, this section will contain no information. As parts are identified, the dimensions and number of bends will populate with new information. Before identifying any parts, verify that the material is added to the Tube / Pipe Library. Click the BROWSE icon to access the Library. If the material is not listed, add the material needed for the imported parts to the Library before continuing.

Important

When importing files, the software requires a full material profile. Elements of a design that do not use a full profile, such as a longitudinal cut, will not be recognized by the importing feature.
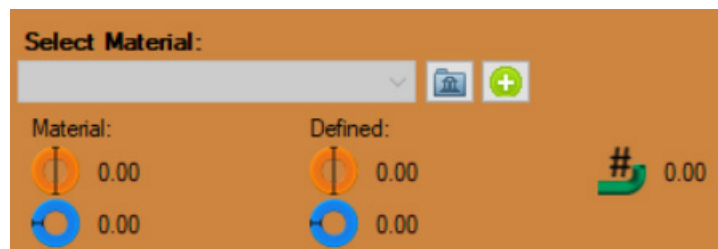
2.3.3 Bend Count Display

Below Select Material, the software will display a bend count. It will appear as # with a green bent part icon. The number next to it is the number of bends that have been recognized in the imported file.

2.3.4 Part Actions

Above the Display Window are three Part Actions. The SAVE PART button is used to save a part once it has been defined. The DELETE FEATURE button is used to delete individual features or lines. ADD LINE is used to add in missing or deleted line features.
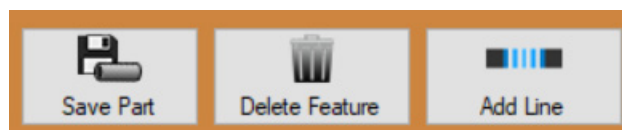
2.4 Define Tab

The Define tab tools are used to automatically identify parts and includes some global definition settings such as Precision Adjustment.
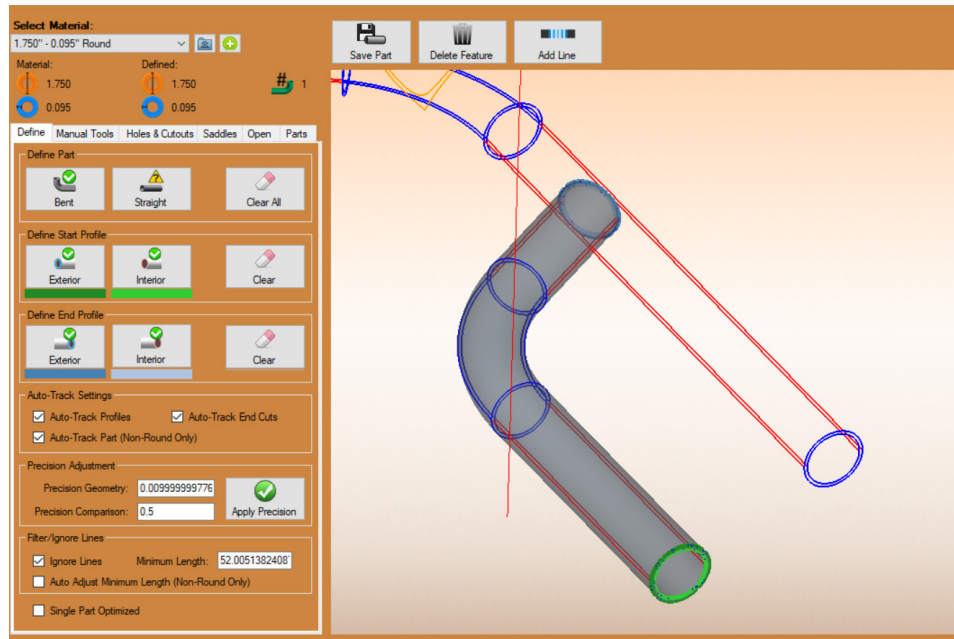
2.4.1 Define Part

Bent

The Bent Define Part option is used to identify a bent part. Select the outermost line at an end cut to identify the size of material and define the bent part. If the software defines the part correctly, it will change to gray. Click SAVE PART before defining the next part.

Straight

The Straight Define Part option is used to identify a straight part. Select the outermost line at an end cut to identify the size of material and define the straight part. If the software defines the part correctly, it will change to gray. Click SAVE PART before defining the next part.

Clear All

Clear All is used to clear any unsaved defined parts. Useful when a part does not define correctly using the Define Part tools.

2.4.2 Define Start Profile

If the software is unable to define the Start Profile of a part defined using the Define Bent or Define Straight tools, use Define Start Profile to select the Exterior and Interior lines of the Start Cut.

Exterior

The Exterior tool is used to select the exterior line for a Start Cut.

Interior

The Interior tool is used to select the interior line for a Start Cut.

Clear

Clear will reset the selections in the event they were not defined correctly.

2.4.3 Define End Profile

If the software is unable to define the End Profile of a part defined using the Define Bent or Define Straight tools, use Define End Profile to select the Exterior and Interior lines of the End Cut.

Exterior

The Exterior tool is used to select the exterior line for a End Cut.

Interior

The Interior tool is used to select the interior line for a End Cut.

Clear

Clear will reset the selections in the event they were not defined correctly.

2.4.4 Auto Track Settings

The Auto Track options are used to automatically track part profiles, end cuts, and non-round parts when defining parts. Unchecking these options will disable them.

2.4.5 Precision Adjustment

Values for Precision Geometry and Precision Comparison are changed in the Precision Adjustment box. Clicking Apply Precision will define the elements of a part to a greater degree. The effectiveness of this tool depends on how well the Bend-Tech importer is able to define the part during the import process. Use caution when using Precision Adjustment as it can affect design elements in a way that compromises the overall part.

2.4.6 Filter/Ignore Lines

The Filter/Ignore Lines options are used to ignore lines that are longer than the MINIMUM LENGTH specified. These lines are ignored in some of the tracking functions when the option is enabled. The value is always metric.

When AUTO-ADJUST is set, a reasonable value is set based on the user's first selection. Usually the longest line of the feature.

2.4.7 Single Part Optimized

When SINGLE PART OPTIMIZED is enabled, tracking functionality generally works better. Some STEP files are automatically detected to be a single part and set this option as enabled automatically.

2.5 Manual Tools Tab

If the Parts are unable to be automatically defined with the Define tool, the Manual Tools can be used to identify parts based on their component features, recreate elements, and define elements. This is a time intensive process.

Tip

When using Manual Tools it is best to retain at least one element, such as a bent section, that has been imported correctly. Base the rest of the edits off that correct segment.

2.5.1 Define Straight Section

The Define Straight Section options are used to define straight parts or straight legs of bent parts.

Two Points

Use the TWO POINTS option to define a straight section between two selected points. When using the Two Points feature, the software will display a red point at the center of an element. To define a straight section of material, click on a red point and connect it with another red point.

Circle → Distance

Use the CIRCLE→ DISTANCE option to define a straight section by selecting the circle of an end cut, and a point that indicates the end of the straight section.
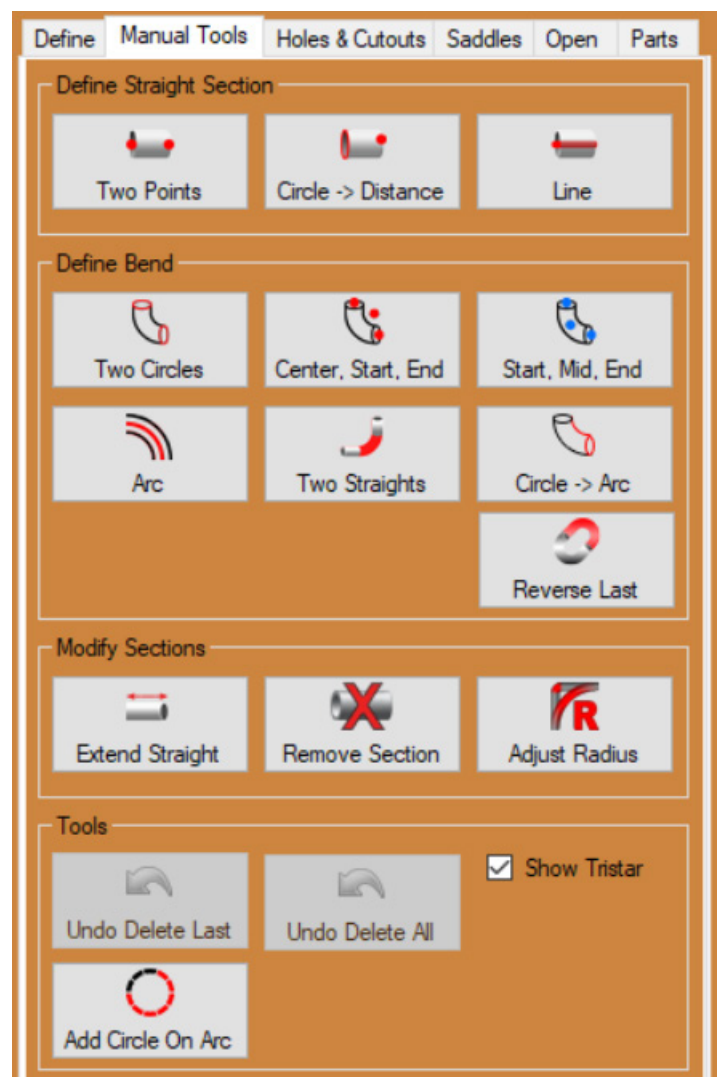
Line

In some cases, an imported element may not be represented as solid, and will simply have a line showing its length. Use the LINE option to define a straight section from this type of line.

2.5.2 Define Bend

Bends are a critical part of any design, and creating them accurately is important to importing the design. When using Manual Import, the software may not recreate the bends as designed. In this case, manually define the bends to match the original design using the Define Bend tool.

Two Circles

The TWO CIRCLES option is used to define a bend between two distinct circles at each end of the bend. Click the first circle in the bend, then it will prompt to click the second circle. The software will then connect the two circles with the appropriate bend.

Center, Start, End & Start, Mid, End

If a bend has been imported and there are missing circles, or there are no circles, choose the CENTER, START, END tool, or the START, MID, END tool, to connect points and create a bend segment. While both features operate similarly the same, the feature used depends on the given import and the elements it contains.

Arc

ARC is used to define a bend based on an arc. The Import may display a bend by displaying an arc line. To define the bend, select ARC and use the cursor to create the bend segment based on the arc line.

Two Straights

The TWO STRAIGHTS option defines a bend by calculating the curve of the bend from two previously defined straight sections of the part.

Circle → Arc

The CIRCLE → ARC method defines a bend based on a selected circle and an arc. The Import may display a circle on one end of the bend and no circle on the other end of the bend. If there is an arc present, use the CIRCLE → ARC tool. Click the circle, then use the arc as a guide to place the missing circle. This will define the bend.

Reverse Last

The REVERSE LAST option is used to change the direction the element travels along its circular arc. For example, picture a bend as a section removed from a donut with two straight cuts splitting it apart, it swaps which piece of donut is effected by the tools.

2.5.3 Modify Sections

Modify Sections is used to redefine elements of the imported design. Use these options to extend, remove, or adjust elements.

Extend Straight

The EXTEND STRAIGHT tool is used to extend a piece of material that may have only partially imported, leaving a missing section. Click on the Extend Straight icon, then click on the starting end of the missing segment. Click the location where the segment should extend to, or click the segment where it should connect.

Remove Section

REMOVE SECTION is used to break a part or assembly down to its basic elements that may include circles and line sections. Remove Section is helpful if the part was not defined correctly. Remove Section is used to remove sections of parts or entire parts one at a time.

Adjust Radius

If an imported part displays an incorrect bend radius, use the ADJUST RADIUS tool to edit the radius. The Adjust Bend popup displays the initial CLR as reference. In the New CLR text box enter the adjusted bend radius. The adjusted bend radius can be used to adjust all matching bends in the design. Click OK to apply the changes to the import.

2.5.4 Tools

The Tools section includes UNDO DELETE LAST and UNDO DELETE ALL buttons. The ADD CIRCLE ON ARC tool is used to convert an arc to a circle. The SHOW TRISTAR option will enable or disable the Tristar in the Display Window.

2.6 Holes & Cutouts

If there are holes or cutouts that need to be defined, click the Holes & Cutouts tab. In the Define Holes/Cutouts box, click Exterior, then use the mouse to click the outside edge of the cut. Click Interior, then click the interior edge of the cut to define the inside edge. If any lines are created incorrectly, click Clear All and start over.

2.7 Saddles

The Saddles tab uses line elements from the imported information to create Saddle definitions. Locate a circle in a position where two pieces of material intersect. Then, in the Define Saddles box, click the Exterior or Interior icon. Next, click on the corresponding circle in the import.
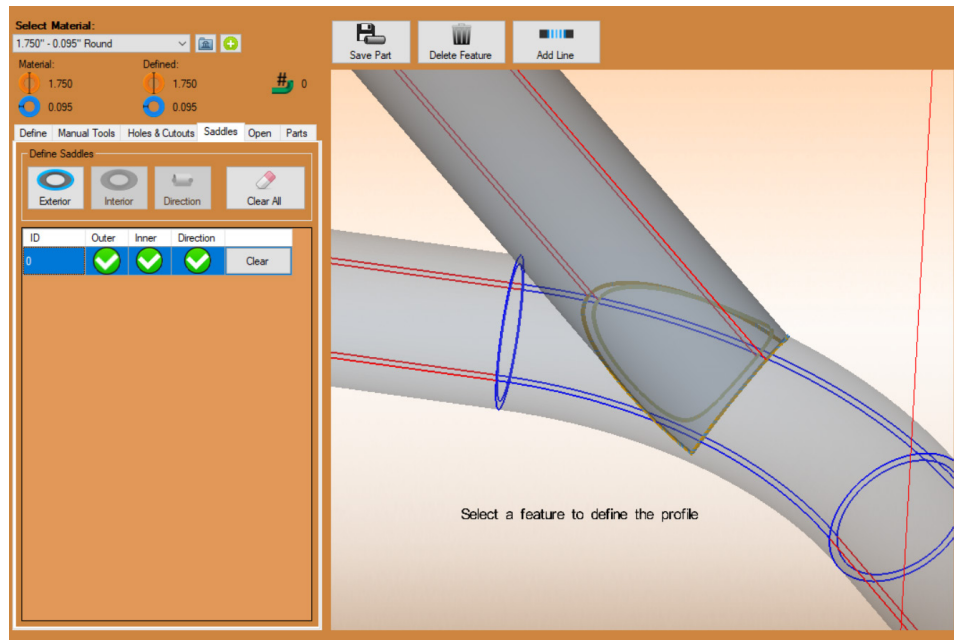
Tip

Bend-Tech recommends using Auto Saddles before attempting to manually define saddles. The Auto Saddles tools is located under the Parts tab.

2.8 Open

To define Open Features such as specialized design cuts, specialized material shapes, or other custom features, navigate to the Open tab. In the Define Open Features box, click Auto-Track or Manual Definition to define non-typical features of an import.

2.9 Parts

The Parts tab will list all parts that have been defined. From this tab, Saddles can also be defined automatically with the AUTO SADDLES tool. If there are missing saddles, go to the Saddles tab and define them manually.

The Select tools are used to select parts and DELETE SELECTION will deleted any selected parts.

2.10 Manually Importing Square or Rectangular Material
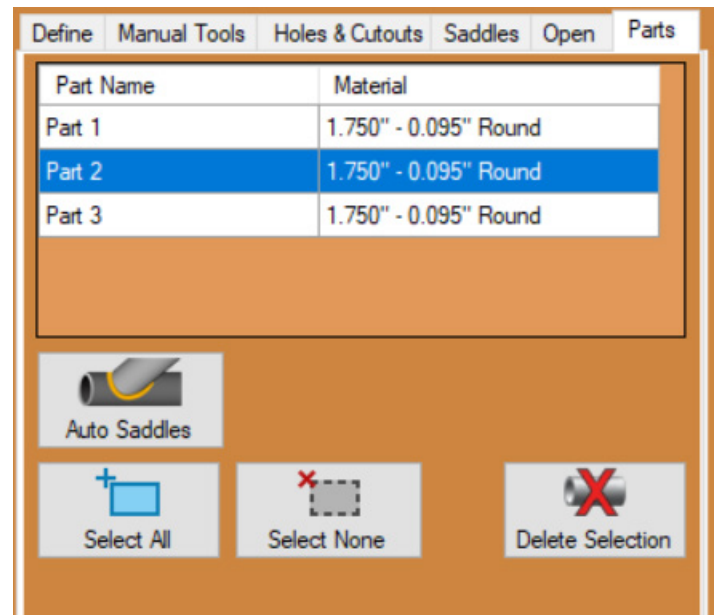
In order to import square or rectangle material, Bend-Tech requires a radius at the corners of the material. With square or rectangle material, each side of the radius on the outside of the material will feature a line that needs to be defined. Square and rectangle material will import with many more edge elements than round, and typically requires more time to define.

2.11 Importing NC1 Files

It is possible to import NC1 files into Bend-Tech CAM, however, significant loss of information and part details can become an issue. Some NC1 importing issues include minimal data points regarding cut profiles as well as difficulty synchronizing materials.

Under CREATE NEW in the Bend-Tech Dragon Task Menu, click the Import icon. Locate the NC1 file for import and click OPEN.

An "Import Options" pop-up will appear. The part can be opened in Designer or it can be imported to Nesting. Bend-Tech recommends opening the part in Designer to ensure all the elements of the import are correct. Click OK.

2.11.1 NC1 Layout Viewer

When opened in Designer, the part will open in the NC1 Layout Viewer Interface. This interface will display all the part information the Bend-Tech software was able to gather on import.

Important

NC1 files will always be formatted in metric measurments.

2.11.2 Select Material

The import feature will attempt to match a material in the Material Library with the material contained in the NC1 file. If it cannot find an exact match, it will choose the closest material. Consult the Select Material drop down to ensure the material is appropriate for the imported design.

Tip

Verify the proper material is selected when importing NC1 files for nesting. If the software cannot find a matching material, the import will fail. The NC1 import software does not check if the material is suitable for importing.

Important

Always check imported NC1 designs in Edit Flat to endure the OD matches the cutting geometry.

2.12 Importing DXF Files

It is possible to import DXF files into Edit Flat. DXF files can be applied to a part as cutting information.

2.12.1 Importing a DXF File

With a part open in the Straight Part design interface, click Edit Flat. In Edit Flat, click the Import DXF icon at the top of the interface. An Import DXF File popup will appear. Locate the DXF file for import, click on the file, and click OPEN.

When OPEN is clicked, an Import/Export DXF popup will appear. Choose Standard for the DXF Import/Export method.

A "Set DXF Base Point" interface will appear. Click OK and the DXF information will appear in the Edit Flat part display. If the information looks correct, drag the DXF information to the required location on the part.

Tip

Be familiar with Edit Flat in order to properly place the DXF files in Edit Flat.

2.12.2 DXF Actions

Once the DXF information is placed on the part, click the Actions tab and assign the appropriate action to the information.

2.13 Faceted Model in CAD

If the part appears in the import interface but is not recognized as a proper part by the import software, it will typically be displayed as a series of lines rather than a single line with a shaded representation of the material. The Faceted Model tool will convert the series of lines representing a part into a defined part.

2.13.1 Opening a File in Tekla Import

In Dragon CAD, click CAD Import. In the Import CAD File popup click Faceted Model. An Import Tekla Model popup will appear. Locate the desired file, click on it and click OPEN. The file will open in Tekla Import - HR1 (mixed facet and clean) interface.

2.13.2 Creating Tube

Locate the part or parts that are defined by lines. Under the Define tab, in the Define box, click the "Create Tube by Dragging" icon. Use this feature to drag a line across each element to create a section of tube. Perform this action for all sections of tube.

2.13.3 Join with Bend

Once the straight sections of tube are defined, join the tubing with bends to complete the part. Under the Define tab, in the Define box, click the "Join with Bend" icon. Click one element of the tube, then click the next element in the design and it will create the appropriate bend to join the two sections. For a part with multiple bends, click the sections in series to continue creating bends until the part is complete.

Quick Nest

3.1 Quick Nest Overview

Quick nesting is used to nest multiple lengths of parts that require straight end cuts, and no internal cuts. It will automatically nest the parts, on the required number of sticks, for the specified material and stock length.

3.1.1 Building the Nest

To get started, select the MACHINE and MATERIAL from the drop down menus. Enter the STOCK LENGTH of the material into the provided text field. Then click ADD to add a new part. Enter the LENGTH of the part and the QUANTITY into the spreadsheet. Continue to add as many parts as needed.

To delete any of the parts from the spreadsheet, select the part to delete and click REMOVE. CANCEL will close out of the interface without sending the project to Nest Part.

Select Machine: Dragon — Library

Select Material: 1.75 DOM — Library

Diameter: 1.750
Wall Thickness: 0.120
Weight: 0.000

Stock Length: 240

Needed Lengths:

Length	Quantity
12	3
24	1
36	2
48	4

Add   Remove   Cancel   OK

Click OK once all the parts have been added, to send the parts to a Nesting Project. The software will automatically nest the parts onto the required number of sticks. The project can now be run on the Dragon Machine as normal.
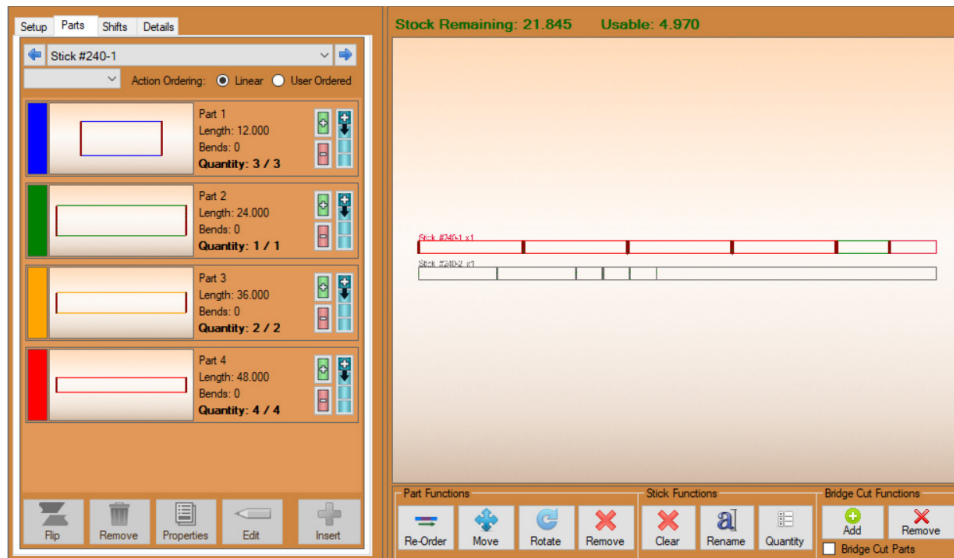
04

Edit Flat

4.1 Edit Flat Overview

After creating a Straight or Bent part, the part can be sent to Edit Flat where parts can be fine tuned before fabrication: create holes and text, create custom geometry, calculate dimensions, edit actions, etc.

4.2 Actions Menu

Cancel

The CANCEL button is used to end a function that is currently in process or tied to your cursor.

Example

If your text is tied to your cursor and you realize you made a typo, you can click the Cancel button and fix the typo before you place the text in the design.

Back

The BACK button will allow you to go back and re-do a step while operating a multiple step function.

Delete

Select the DELETE button and then a feature in the design to erase it completely from the design.

Free Select

Use the FREE SELECT button to enable the option to place an entity anywhere in the design. When Free Select is disabled, placement of entities in the design will be constrained to a point.

Display

Use the DISPLAY button to view and change settings for display.

Paste

The PASTE button is used after using the Copy tool to paste the desired part back into the design.

Import DXF

The IMPORT DXF button is used to import .dxf files into Edit Flat so that they can be added to the part design.

4.3 Part Data

The Part Data tab shows basic part settings such as the selected machine, selected material, cut length, and any bending instructions the part has. Use this tab to add or adjust the Part ID, name the part, and part notes.

4.3.1 Actions
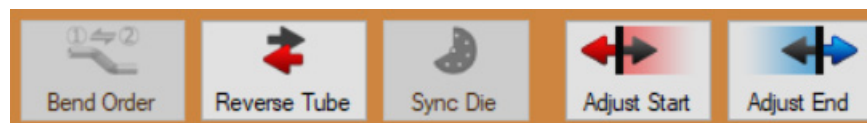
Bend Order

The BEND ORDER button will open the bending simulation, and will allow the user to manipulate the part for bending only.

Reverse Tube

The REVERSE TUBE button will reverse all of the geometry of the part including copes, bend locations, text, holes, etc. This will NOT mirror the part.

Important

Reverse Tube will NOT mirror the part.

Sync Die

SYNC DIE is used to update the part with the current die settings, or change the die for any bend.

Adjust Start

The ADJUST START button is used to adjust the start location of the part. This function lets you quickly add or subtract length off the beginning of the part.

Adjust End

The ADJUST END button is used to adjust the end location of the part. This function lets you quickly add or subtract length off the end of the part.

4.3.2 Part ID

LOCATION X and LOCATION Y adjusts the position of the Part ID text. The origin for these coordinates is the lower left hand corner of the part in the Flat Display. LOCATION X will move the Part ID along the length of the part, while LOCATION Y will move the Part ID along the circumference of the part.

Note

LOCATION Y can be measured in distance or degrees. Select the appropriate option from the Y-Value Format options.

ANGLE adjusts the angle of the Part ID. SIZE adjusts the size of the text. The ACTION drop down menu is used to select how the Part ID will be applied to the part. The options include engraving, marking, and cutting.

To change the Part ID text, change the TEXT field. This will default to the Part Name that was chosen when the part was first created. Several font choices are available to choose from the FONT drop down menu. For polyline fonts (text drawn with a single line vs text drawn with an outline) choose either __Arial or __Times New Roman.

4.3.3 Other Options

Cut-Off Start and Cut-Off End

The CUT OFF START and CUT OFF END options are used to add additional material to the beginning or end of the part that will be cut-off after bending.

Part Name

The PART NAME in this section is different than the Part ID. This will not show up on the part. It is used to identify parts for saving files or for a more detailed name than what is used for a Part ID.
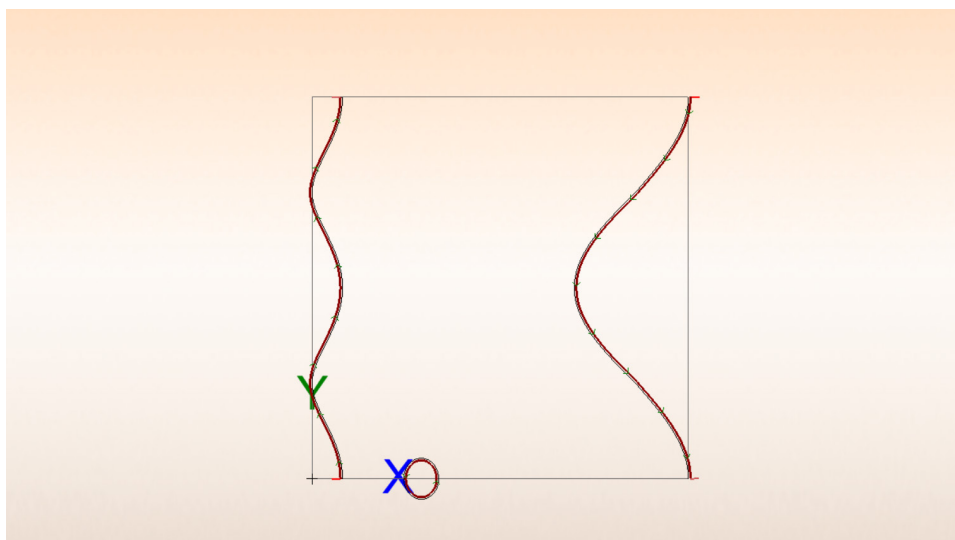
Date

The DATE text field can be used to record a design date, a production date, or another date that is important to the part or project.

Notes

Use the NOTES section to record any specific instructions or notes regarding the part.

4.3.3 Display Window

The right portion of the interface is the Display Window. The part will display as a flat, unrolled 2D drawing.

4.4 Create Geometry

The Create Geometry tab is where custom or new geometry can be added to the part. This is useful for creating non-standard holes, end cuts, or other unique geometry that may be needed for a part. Multiple methods for creating geometry are provided.

4.4.1 Point

Cursor

Select the CURSOR option and click anywhere on the screen to place a point. The point will be placed at the exact location of your cursor and will not reference any other entities while being placed. The point will not be highlighted blue before placement.

Incremental

Select the INCREMENTAL function to place a point an incremental distance away from a selected reference point. When selecting this option provide X and Y coordinates, and then select a reference point in the display area. The point will be highlighted in blue while the reference point is being chosen. X is the horizontal distance, and Y is the vertical distance from the reference point.

Note

A negative (-) X value will place the point to the left of the reference point. A negative (-) Y value will place the point below the reference point.

Angle

Use the ANGLE option to place a point at the entered ANGLE and DISTANCE from a selected reference point. If the ANGLE field is set to 0-degrees, the new point will be created directly to the right of the reference point. To place the new point, click on a point to use as a reference point. The point will be highlighted in blue while the reference point is being chosen.

Absolute

The ABSOLUTE option is used to set a specific X and Y coordinate for the point. The reference point will always be the lower left corner of the part, or the origin that shows X and Y in the display area. Once the X and Y Coordinates are entered, click APPLY to add the new point to the part.

Intersection

The INTERSECTION option is used to place a point at the intersection of two lines.

Entity End

Use the ENTITY END option to place a point at the end point of an object or line in the display area. The point will be placed at the end that you click closest to and will be highlighted in blue before the object is chosen.

Entity Center

Use the ENTITY CENTER option to place a point in the center of an object or line. The point will be placed directly in the center of the object highlighted. The point will be highlighted in blue and will be visible before the object is chosen.

Note

This option is not meant to find the center of self-created objects like flanges, but rather lines, arcs, text, tabs, slots and holes.

Entity Mid-Point

The ENTITY MID-POINT option is used to place a point at the mid-point of an object. The point will be placed directly in the center of the section of the object highlighted. The point will be highlighted in blue and will be visible before the object is chosen. Unlike Entity Center, the ENTITY MID-POINT option will select the mid-point of the line that makes up an object (like a hole), instead of the center of the actual object.

Opposite Side

Use the OPPOSITE SIDE option to place a new point on the opposite side of the selected point.

4.4.2 Line

Two Points

The TWO POINTS option is used to draw lines between two points. Click on the first point in the display area and it will turn into a blue circle. Click on a second point to create a line between the two points. Click the CONTINUOUS checkbox to enable continuous selection. This will automatically start new lines with the first point set as the end of the last line. To stop drawing lines while in Continuous Mode click CANCEL from the Actions Menu.


Horizontal

Use the HORIZONTAL option to create a horizontal line the length specified in the LINE LENGTH text box. Place the line by clicking a point in the display area. The mid-point of the line will be centered on the point you select.

Vertical

Use the VERTICAL option to create a vertical line the length specified in the LINE LENGTH text box. Place the line by clicking a point in the display area. The mid-point of the line will be centered on the point you select.

Horizontal and Vertical

Use the HORIZONTAL AND VERTICAL option to create two intersecting lines of equal length. Set the line lengths using the LINE LENGTH text box. Click on a point to place the lines. Both lines will be placed with the intersection directly over the point you select.
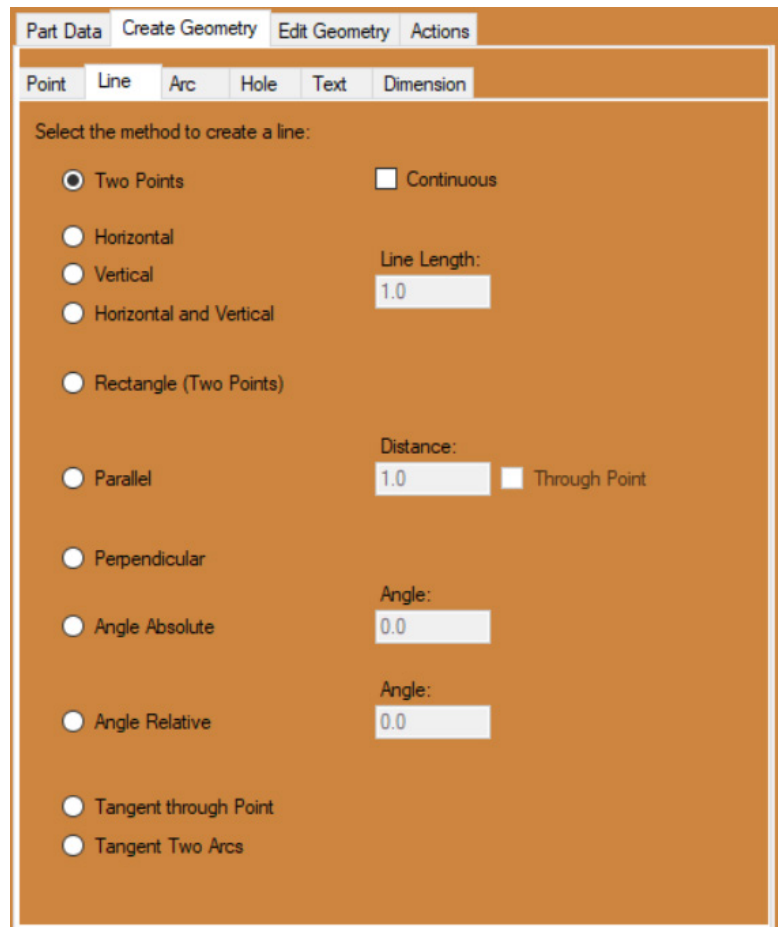
Rectangle

The RECTANGLE (TWO POINTS) option is used to create a rectangle from a primary point and a secondary point. Click a point in the display area, this will highlight it with a blue circle. A blue rectangle will be drawn between the selected point and the point nearest the cursor. Move the cursor near the intended second point, and make sure the blue rectangle is correct. Click to create the rectangle.

Parallel

Use the PARALLEL option to place a parallel line a specific distance from a reference line, or through a point in relation to it. If a distance is entered into the DISTANCE text box and the THROUGH POINT checkbox is not selected, click a line to set it as the reference line. After picking the reference line, select which side the parallel line will be placed on by clicking on that side of the reference line. If the THROUGH POINT checkbox is selected you must first select a reference line, followed by a point that the parallel line will pass through.

Perpendicular

The PERPENDICULAR option is used to create a line perpendicular to a previously established line. Set the length of the perpendicular line using the LINE LENGTH text box. Then, click a line in the display area to set it as the reference line. The side clicked on determines the direction of the line. Click on any point to place the perpendicular line there.

Angle Absolute

Select the ANGLE ABSOLUTE option to create a line in the display area with a specific angle. First, enter the desired length in the LINE LENGTH text box, and the desired angle in the ANGLE text box. Click on a point to place the Line there.

Angle Relative

The ANGLE RELATIVE option is used to place a line at an angle in reference to another line. Enter the length of the desired line in the LINE LENGTH text box and the relative angle in the ANGLE text box. Click on a line to use as a reference line, and click on a point to use as a starting point for the new line. The angle of the line will be the sum of the angle of the reference line and the value entered in the ANGLE text box.

Tangent through Point

Select the TANGENT THROUGH POINT option to create a line connected to an arc. Click near the side of the arc that the line will be drawn to, then click on a point away from the arc to create the line.

Tangent Two Arcs

Select the TANGENT TWO ARCS option to add a line between two arcs by clicking near the side of the first arc that the line will be drawn to. Select a second arc to create a line that attaches to the tangent of that arc.

4.4.3 Arc

Fillet

The FILLET tool is used to connect two points with an arc at the entered RADIUS. Click on the first line, which will turn blue. A blue arc will come off the first line and connect with whichever line is nearest to the cursor. Change the direction of the fillet by moving the cursor to the other side of the first line. Click to place the fillet. If the AUTO-TRIM checkbox is checked, any part of either line that extends past the fillet will be cut off.

Values

Use the VALUES options to create an arc based on three dimensions. Enter the RADIUS, START ANGLE, and END ANGLE of the arc into their respective text fields. Click on a point to place the arc with its center directly over that point. Check the FULL CIRCLE checkbox to automatically set the start angle at "0" and the end angle at "360", creating a full circle.

Tangent - Three Lines

The TANGENT - THREE LINES option creates an arc by using three lines as tangents. Set the start and end angle of the arc using the START ANGLE and END ANGLE text boxes. Click on three lines within the part, in any order, to create the arc.

Three Points

Select the THREE POINTS option to create an arc based off of three points. The first and third points will be end-points. The second point will be on the arc and will define the radius of the arc. If the first and third points are the same point, a circle will be drawn between the first and second point.

Arc - Center and End Points

Use ARC - CENTER AND END POINTS to create an arc based on a point that lies in the center of the arc. Click on the point that will lie in the center of the intended arc. Click on another point to create the first end point. A blue arc will be drawn from the first end point to the point on the arc closest to the cursor. Move the cursor until the second end point is in the desired location, and click to place the arc.

Circle - Center and Edge Points

The CIRCLE - CENTER AND EDGE POINTS option will create a circle based on a central point and an edge point. Click on a point to set it as the center point. A blue circle will be drawn with a radius set to the point nearest to the cursor. Move the cursor to the desired point and click to place the circle.
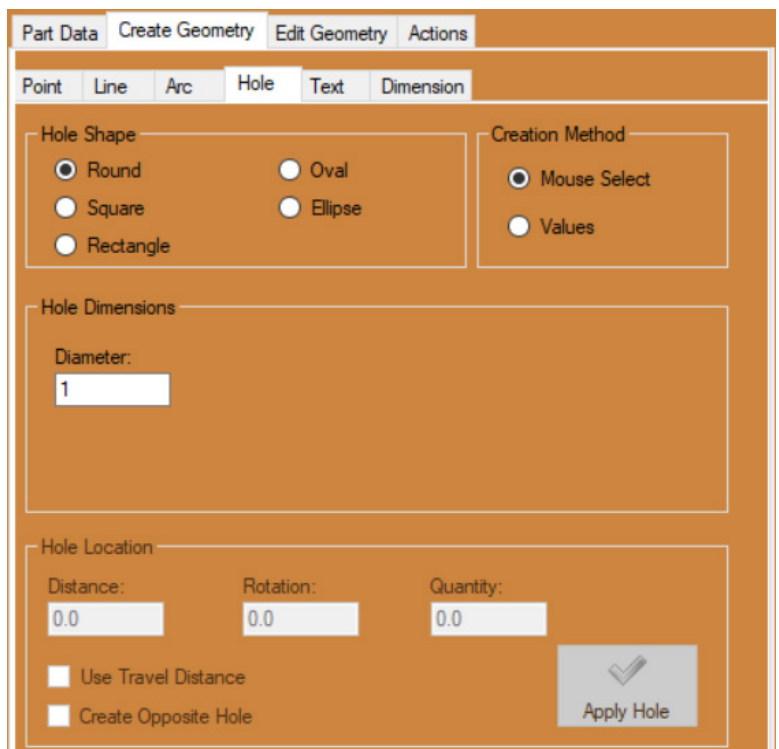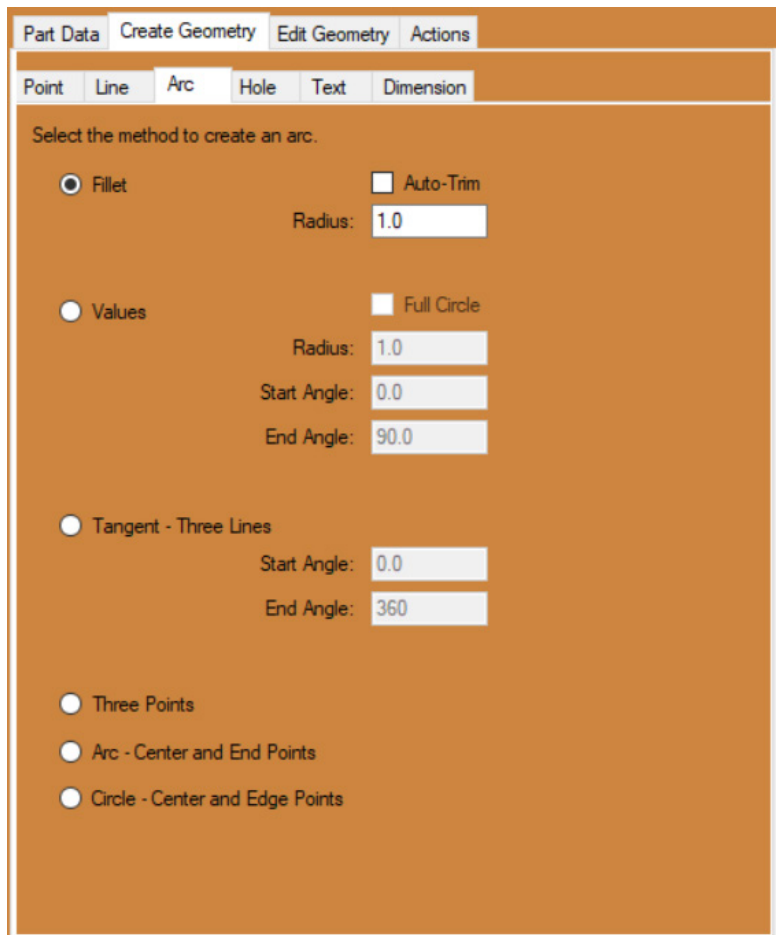
4.4.4 Hole

Hole Shape

Available hole shapes include round, oval, square, ellipse, and rectangle. The holes can be placed freely, using the MOUSE SELECT Creation Method or with set VALUES, such as distance and rotation. Hole Dimensions include width, height, angle, and corner radius. Not every hole type will use every dimension.

Hole Location

DISTANCE is the distance from the origin that the hole will be placed. ROTATION is the distance in around the circumference of the tube. Use the USE TRAVEL DISTANCE option to set the rotation value as a length around the circumference of the tube (inches or millimeters) rather than an angle of rotation (degrees). The CREATE OPPOSITE HOLE option is used to create an opposite hole.

4.4.5 Text

Use the NEW button to create new text on the part. The MOVE button is used to move the text to a new location on the part. EDIT is used to edit any existing text. When using the EDIT button, click APPLY after edits are finished.

Enter the desired text into the TEXT field. Choose the FONT FAMILY from the drop down menu. If the text needs to be bold or italic, check the appropriate option. To set the text at an angle, enter an angle in degrees into the TEXT ANGLE text box. The font size is changed by entering the desired height of the text into the FONT SIZE text box. The LETTER PADDING text field is used to adjust the space between the letter.

The text will display in the Display area below the Text Properties.
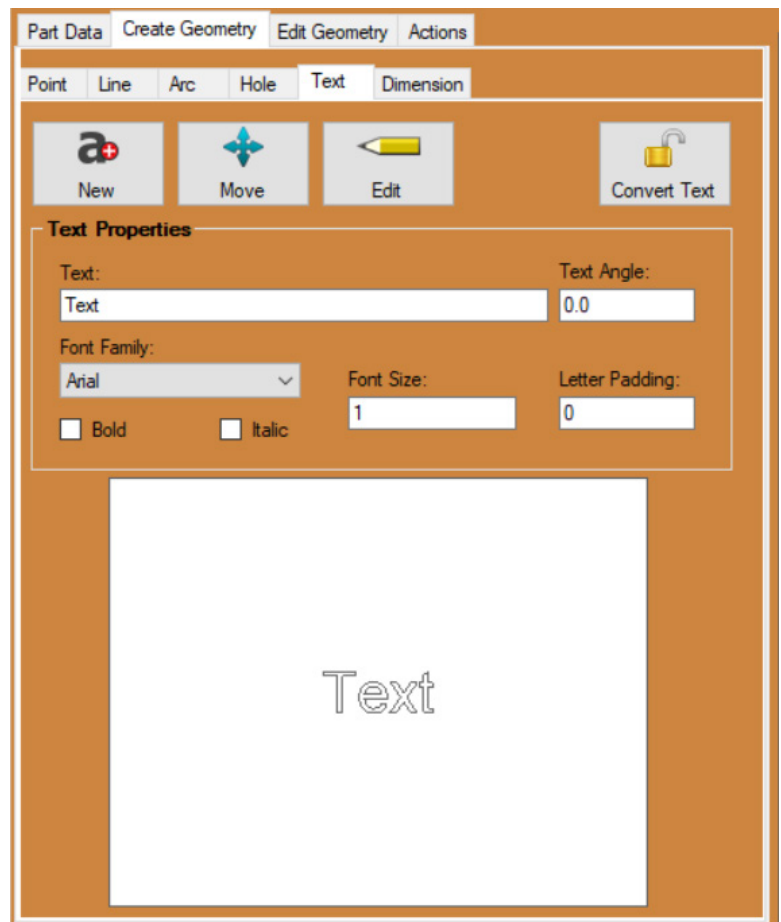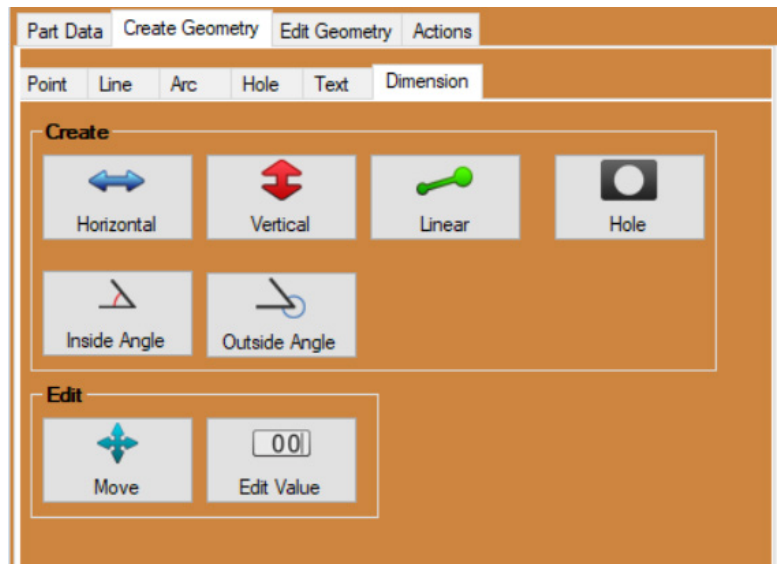
4.4.6 Dimension

Horizontal

To create a horizontal dimension, click the HORIZONTAL button and click two points in the display window. A dimension with dimension lines will appear and move with the cursor. Move the dimension to the desired location using the cursor and click to place it.

Vertical

To create a vertical dimension, click the VERTICAL button and click two points in the display window. A dimension with dimension lines will appear and move with the cursor. Move the dimension to the desired location using the cursor and click to place it.

Linear

The LINEAR options will create a dimension between two points. When two points in the display window are clicked, a dimension with dimension lines will appear and move with the cursor. Move the Dimension to the desired location using the cursor and click to place it.

Angle

Angle dimensions can be assigned to inside or outside angles. Click the appropriate ANGLE button. Click the outside point of the first line. The points can be endpoints of a line or individual points. Click the inside point of the first line. Now click the outside and inside points of the second line. The dimension with dimension lines and arc will appear and move with the cursor. Move the Dimension to the desired location using the cursor and click to place it.

Hole

To set dimensions for a hole, click the HOLE button and click on a hole. The radius of the hole, along with an abbreviated version of the name of its shape, will be drawn and bound to the cursor. Move the cursor until the dimension is in the desired location and click to place it.

Move

To move a dimension, click on the MOVE button and then click on a dimension. The dimension will now move with the cursor and can be repositioned. Click to place the dimension.

Edit Value

To edit a dimension, click on the EDIT VALUE button and click on a dimension. A window will pop up and ask for a new display value for the dimension. Enter a value and click on the "OK" button.

4.5 Edit Geometry

Features to Include

The Features to Include section is where various features can be selected or unselected depending on what needs to be edited.

Selection

There are three selection methods for selecting geometry. CURSOR SELECT allows for objects to be deleted, moved, or copied by simply selecting them with the cursor. The WINDOW selection method is used to draw a box around all the objects that need to be selected. The POLYGON selection method is used to select everything within a polygon that is drawn around the objects.
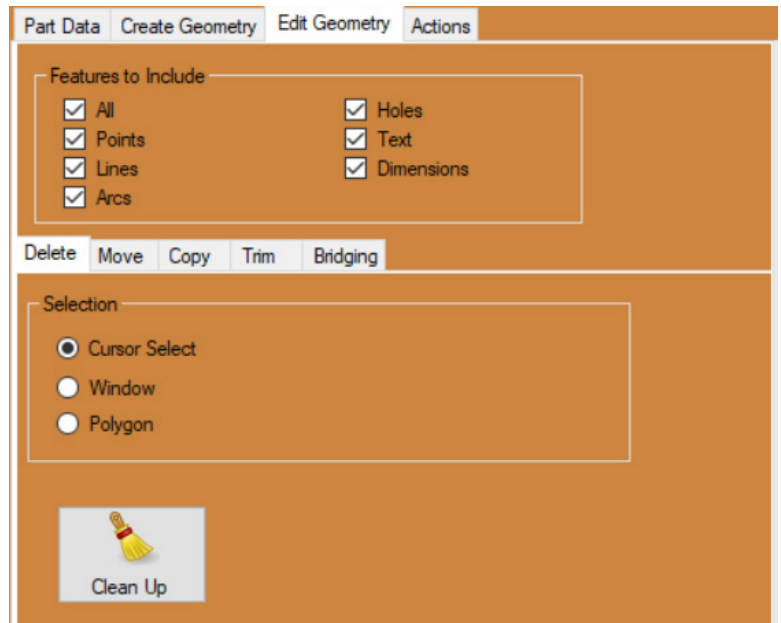
4.5.1 Delete

In addition to the selection methods described above, the Delete tab also includes the CLEAN UP button. This will remove any double lines or features that are irrelevant.

4.5.2 Move

Before using any of the Move and Copy methods, ensure the Selection method is chosen. Select any element in the display window. Choose any of the Move and Copy methods, and the action will be immediately applied.

Incremental

The INCREMENTAL method will move or copy selected elements by specified amounts. Enter positive numbers in the HORIZONTAL and VERTICAL text fields to move elements to the left or down, and positive numbers to move elements to the right or up.

Angle

The ANGLE method will move or copy elements based on a specific angle and distance. 0-degrees is directly to the right. Enter the DISTANCE from the original location that the selection should travel.

Old - New

The OLD - NEW method is used to move elements from a specific user designated point, to a new point. Select elements and click on a point to set that point as the original starting point. When a second point is clicked the selection will be moved to that new point.

Old - New with Direction

The OLD - NEW WITH DIRECTION method works similar to the Old - New method, but it requires a direction point to be identified. Select the elements using the selection method, and choose an original starting point and direction point. Now choose two new points to move the elements to. The original starting point will move to the first new point, and the original direction point will align with the new direction point.

Mirror - Line

The MIRROR - LINE is used to mirror the selected elements across a line. Select elements and click on a line. The elements will be mirrored across that line.

Mirror - Two Points

The MIRROR - TWO POINTS option will mirror elements across a line drawn between two user designated points. Select the elements and choose two points. The elements will be mirrored across the line between the points, regardless of if a line exists there or not. The line between the points will not be drawn.

Rotation

The ROTATION method will rotate the selected elements around a user designated point, a specified number of degrees. Enter an angle in the ROTATION ANGLE text field. Select the elements and choose a point that the selected elements will rotate around.

Scale

The SCALE tool moves selected objects a scaled amount from a base point. Enter the scale amount into the SCALE AMOUNT field. Select the object(s) that need to be moved, then select a base point to scale the object from. This will attach the object to the cursor based off the selected base point. Click again to set the object in a new position.

Tip

It is recommended to set the SCALE AMOUNT to a small number, because even minor movements with the cursor will move the object far off the part.

4.5.3 Copy

To COPY objects, select the object or objects with the appropriate Selection Method. Click a second time to set the base point. A copy of the object will now be highlighted in blue and attached to the cursor at the base point. Move the object into position and click to set it into place.

4.5.4 Trim

Trim Single

TRIM SINGLE will trim/extend an object that intersects with another element. Select TRIM SINGLE and click on the line/object that needs to be kept, rather than the side that is to be trimmed. The side that is clicked will turn blue. Any part that is not blue will be trimmed off.

Move the cursor close to the intersection of the two objects and make sure the correct section is blue, then click on object that needs to be trimmed.

<div style="background:red;color:white;">

Important

Remember that lines continue to infinity even if they are drawn with a finite length, so even if two objects don't touch they could still intersect.

</div>

Trim Both

TRIM BOTH will trim/extend both objects that intersect each other. Click on the side of the first object that should be kept, rather than the side that will be trimmed. The first entity will turn blue. Anything in blue will be kept, while black sections of either entity will be trimmed off. Click the second entity on the side of the intersection that should be kept to trim both objects at the intersection.

Trim Continuous

TRIM CONTINUOUS will trim/extend intersecting objects without needing to select each starting object. This tool only works under very specific circumstances and may not produce expected results. Click on an object, making sure to click the side that should be kept, rather than the side that will be trimmed off.
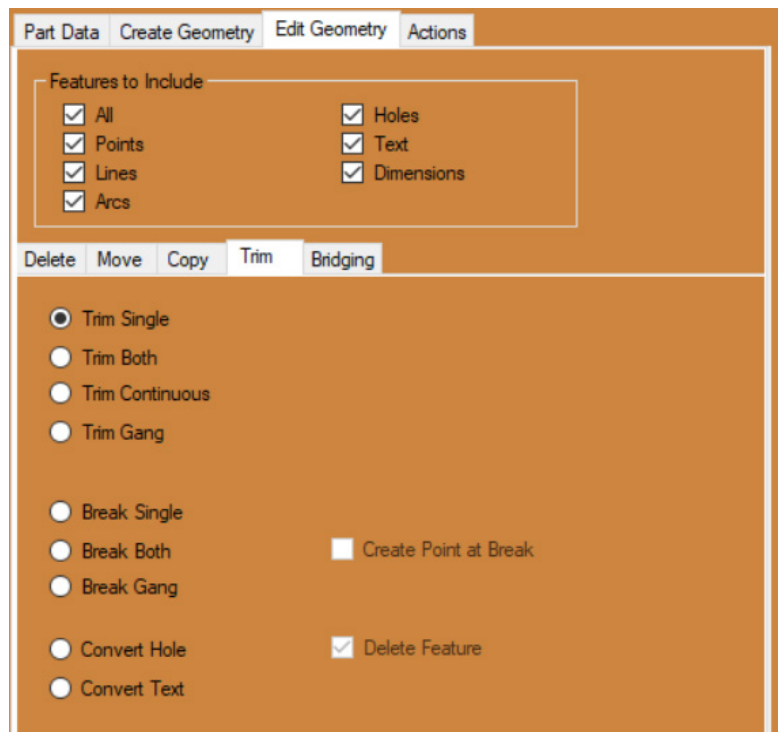
It will turn blue. Move the cursor near the next intersecting object, verify that every part of the first entity that should be kept is blue, then click the second object to perform the first trim. Continue this process to make trims will all intersecting objects until finished. Once finished, click CANCEL to stop trimming.

Trim Gang

TRIM GANG is used to trim multiple elements that intersect a single object. Select the object that all elements will be trimmed to. It will highlight in blue. Then click on each element that intersects the first object that needs to be trimmed. The side of the element that is clicked on, is the side that will be kept.

Break Single

Use the BREAK SINGLE option to break a selected object at an intersection. Click on the object to select it. Then click on the intersecting element to break it at the intersection point. Check the CREATE POINT AT BREAK box to create a point at the intersection of the two objects.

Break Both

BREAK BOTH will break both objects at the intersection. If the CREATE POINT AT BREAK checkbox is selected, a point will be created at the intersection of the objects.

Break Gang

Use BREAK GANG to break multiple elements that intersect with a single object. Click on the primary object that will be used to break intersecting elements. Click on the intersecting elements one at a time to break them at the point of intersection with the first object. If the CREATE POINT AT BREAK box is checked, a point will be created at each break.

Convert Hole

The CONVERT HOLE option is used to convert holes into arcs and lines.

Convert Text

The CONVERT TEXT option is used to convert text into arcs and lines.

4.5.5 Bridging

Auto-Trim

Using AUTO-TRIM will automatically trim the lines that the Bridge crosses, and trim the Bridge to the correct length.

Width
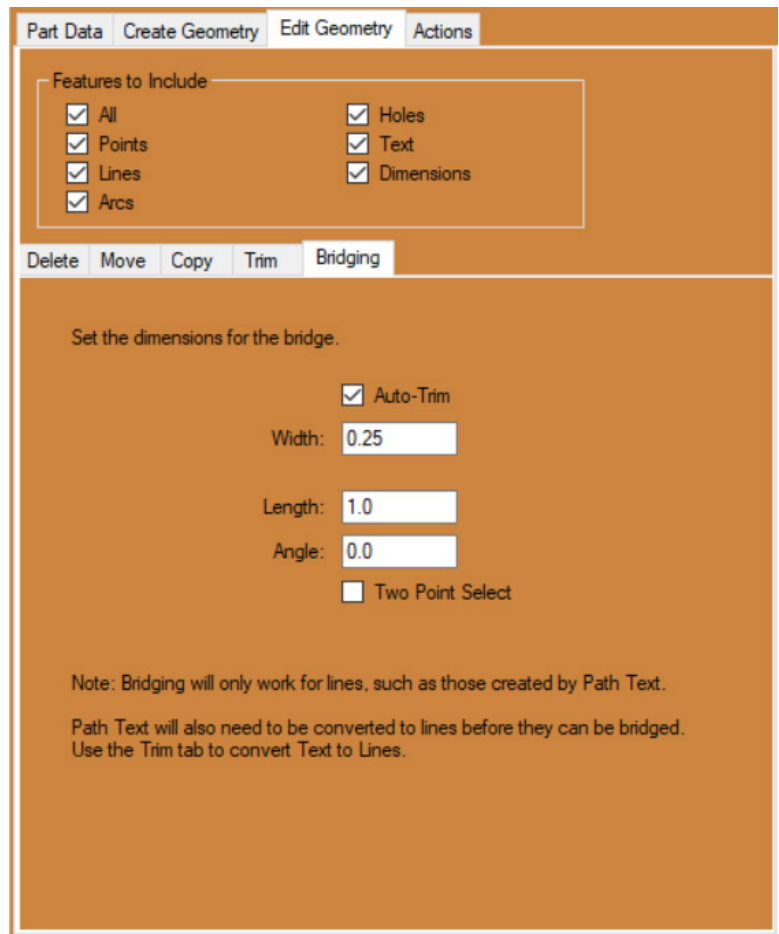
The WIDTH field is used to set the width of the Bridge.

Length

The LENGTH field sets the length of the Bridge.

Angle

The ANGLE field is used to set the Bridge at an angle in degrees.

Two Point Select

When the TWO POINT SELECT is checked, the Length and Angle is set by selecting two points.

4.6 Actions

The Actions tab is used to assign, re-assign, or convert actions, fine-tune Toolpaths, and adjust Lead-In locations.

4.6.1 Create

Use the Create options to create Actions for new or edited geometry. These Actions include Engrave, Marking, Start Cuts, Cut Out (like holes), End Cuts, Pierce Holes, Polyline Cuts, and NRC holes.
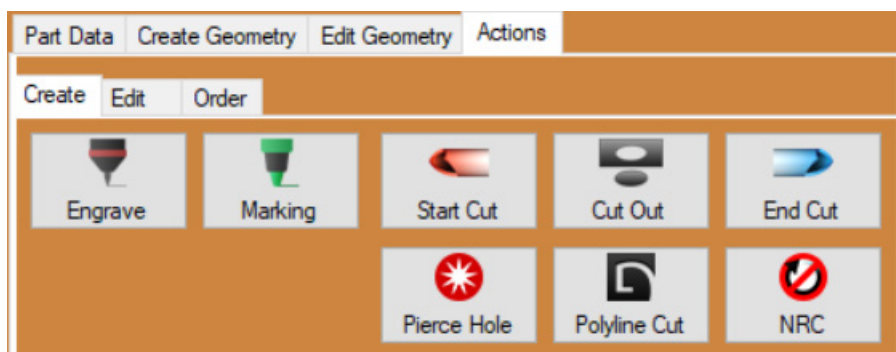
4.6.2 Edit

The Edit tab contains options for converting, removing, editing, and reversing Toolpaths. Options here also include the ability to add a Diamond Tab to notches created in DragonCAD.
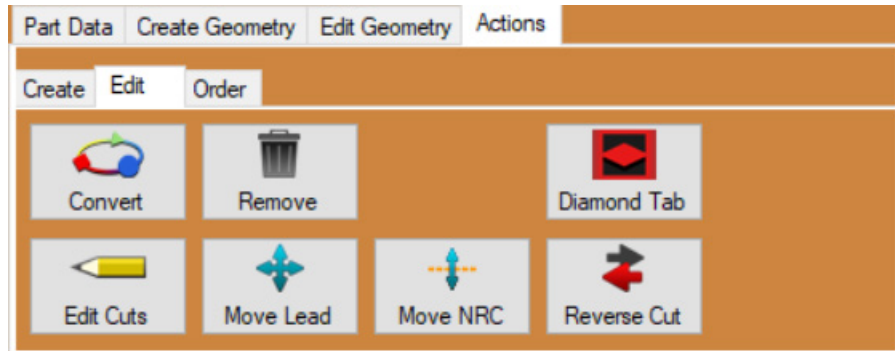
4.6.3 Order

The Order tab contains options for re-ordering the Action. The CLICK option is used to select the order of the Actions by clicking on the Toolpaths in the Display Window. The FLOW option is used to set the order by dragging the mouse across the Toolpaths in the Display Window.
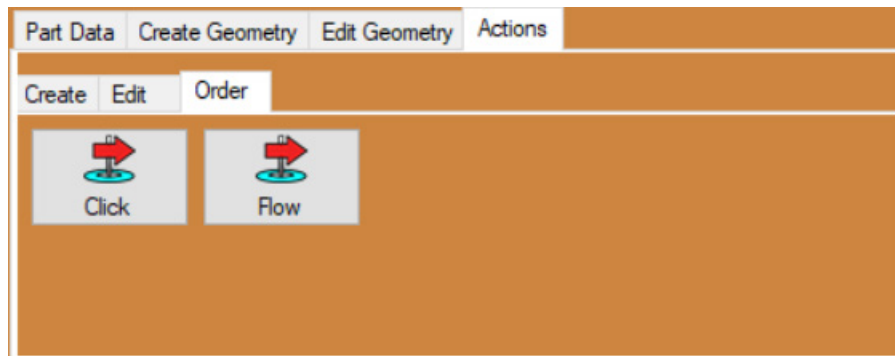
Nest Part

5.1 Nest Part Overview

Multiple parts that use the same material can be added to a single Nesting Project. With a part open in the Part Designer or Edit Flat, click NEST PART from the menu. This opens a new Nesting Project.

5.2 Settings

Ensure the correct Machine is selected from the drop down menu, and the material listed is correct for the parts being run. In the Stock Lengths portion of the interface, set the length and number of sticks being used for the project.
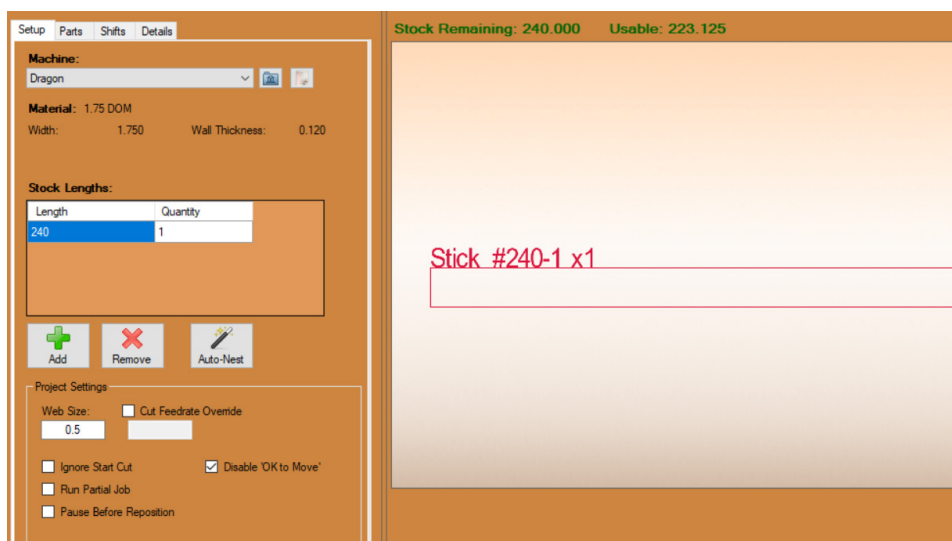
5.2.1 Actions

To add new stock lengths, click the ADD button and enter the length and quantity of the new stock length. To delete a stock length, select the stock length and click REMOVE. When the quantity of the stock lengths are set to 0, click the AUTO NEST button. This will automatically nest the parts on the appropriate number of sticks needs for the quantity of parts being cut.

5.2.2 Project Settings

The Project Settings options override defaults on a per project basis. The Web Spacing between parts and the Cutting Feed Rate can be adjusted. The other options are detailed below.

Ignore Start Cut

When checked, the IGNORE START CUT option will cause the software to ignore the cut profile at the beginning of the stock tube.

Run Partial Job

When checked, the RUN PARTIAL JOB option allows for selecting individual stock pieces to process when transferring a project to the machine.

### Quick Repeat (Production Mode)

When checked, the QUICK REPEAT option will cause the machine to repeat a project immediately upon completion.

Disable 'OK to Move'

When checked, the DISABLE 'OK TO MOVE' option will cause the Torch OK to Move reporting to be turned off. This is useful for dry running a project, or other instances where the reporting is not needed.

5.3 Parts

The Parts tab will show all of the parts that have been loaded into the Nesting Project. They will be organized according to order of fabrication. This tab will also show the name, length, and quantity of each part.
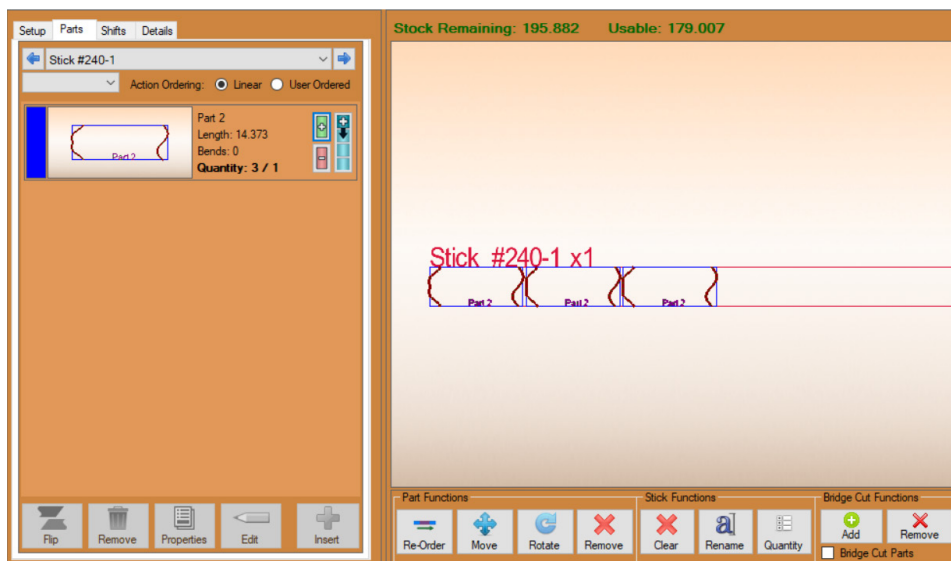
Active Stick & Stick Selection

At the top of the Parts List is a drop down menu where the individual sticks for the Nesting Project can be selected. The Left and Right arrow buttons also cycle through the sticks.

Part Sorting

The PART SORTING drop down menu is the drop down menu located next to the Active Stick menu. It offers a variety of options for sorting the part in the Parts List.

5.3.1 Parts Actions

Auto Fill

The AUTO FILL button is used to fill the selected stick with copies of the selected part.

Add

The ADD button will add one part at a time to the selected stick.

Minus

The MINUS button will remove one part at a time from the selected stick.

Flip

The FLIP button will reverse all of the geometry of the highlighted part from the list above, and create a new listed part. These will be marked 'Flipped' in the part name.

Remove

The REMOVE button will remove the selected part from the part list.

Quantity

The QUANTITY BUTTON is used to input the quantity you need for each part.

Insert

The INSERT button is used to insert the highlighted part into the design at a location designated by the user. Double clicking the part in the list is also a quick way to add the part to the stock tube.

5.3.2 Part Functions

Re-order

The RE-ORDER button is used to select a length of tubing in the display screen and move it to a new location in the order.

Move

The MOVE button is used to move a part and any parts behind it along the length of the stick. When using MOVE, click the part to move and move it to the new position. After moving the part, click the left mouse button to finish placing the part in the new location.

Rotate

The ROTATE button is used to rotate the part around the diameter of the stick. When using ROTATE, click the part to rotate and move the part into the desired position. Clicking a second time will place the part.

Tip

The Rotate tool is helpful in reducing waste between parts by allowing the user to line up end cuts for more optimal spacing.

Remove

Use the REMOVE tool to remove unwanted parts from the Nest by clicking on the REMOVE button, and then clicking the parts in the display window. This will not remove the parts from the Parts List, this only removes them from the sticks.

5.3.3 Stick Functions

Remove All

The REMOVE ALL button will remove all of the nested parts from the selected stick.

Rename

The RENAME tool is used to rename the selected stick.

5.4 Shifts

The Shifts tab will allow the user to assign the actions of the machine to each specific contour, profile, and mark within the nesting project.

5.4.1 Shifts Actions

The Shifts List will list every action for each Nested Stick.
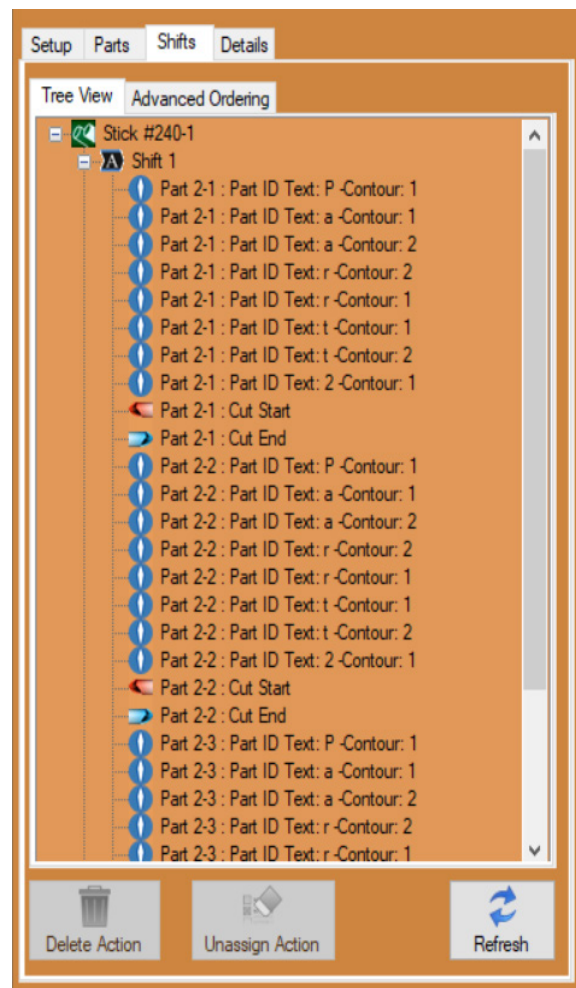
Delete Action

Use the DELETE ACTION button to delete a selected Shift. The Shift will be removed from the nesting project.
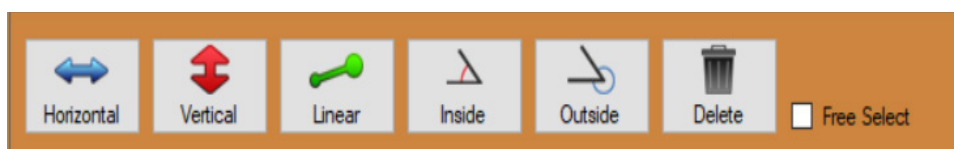
Unassign Action

The UNASSIGN ACTION button is used to unassign the selected action. This will exclude them from the shift.

Refresh

The REFRESH button will undo all alterations made in the shifts order.

5.4.2 Dimensions

Horizontal

Use the HORIZONTAL tool to place a horizontal dimension line between two points.

Vertical

Use the VERTICAL tool to place a vertical dimension line between two points.

Linear

Use the LINEAR tool to place a dimension line between two points.

Inside

Use the INSIDE tool to place an inside angle measurement dimension.

Outside

Use the OUTSIDE tool to place an outside angle measurement dimension.

Delete

Use the DELETE button to delete dimensions. Click the DELETE button and then click on the unwanted dimensions.

Free Select

The FREE SELECT option is use to place a dimension anywhere in the design. When FREE SELECT is disabled the placement of dimensions will be constrained to selected points.
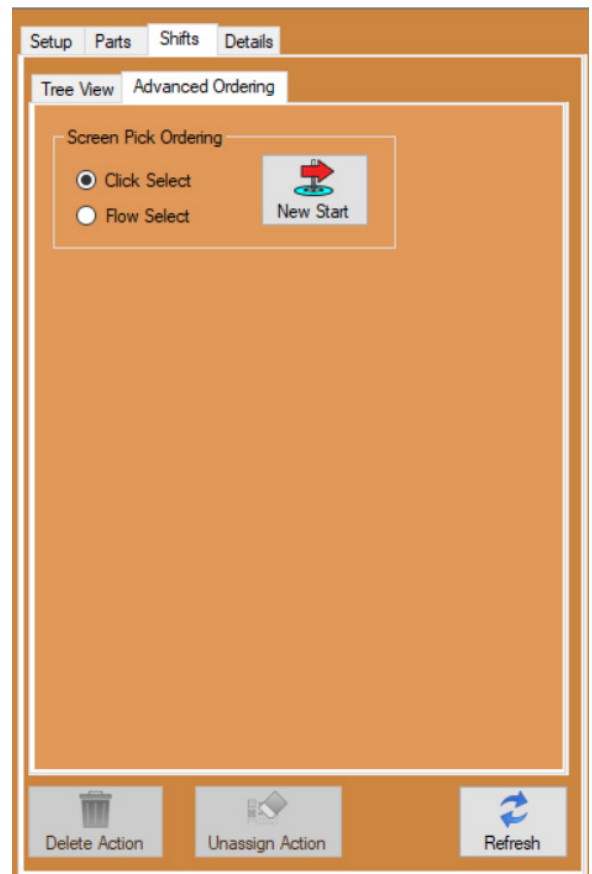
5.4.3 Advanced Ordering

The Advanced Ordering options are used to change the order of the shifts and actions for the Nesting Project.

5.5 Details

The Details tab includes an ESTIMATED TIME for cutting and a MATERIAL COST, if pricing information was included for the material in the Tube / Pipe Library.

5.6 Run Project

When the nesting project is finished and ready for fabrication, send the project to the Dragon machine by clicking RUN PROJECT at the top of the screen. This will open machine control, which is covered in the following chapter.

Machine Control

6.1 Machine Control Overview

When RUN PROJECT or MACHINE CONTROL is clicked two interfaces will open. The first is Mach3, which is the machine driving software. Mach3 is rarely used, and can be minimized. The second interface is the Bend-Tech Machine Control, and it is used to operate the Dragon Machines.

6.2 Machine Control Features

6.2.1 Action Indicator Lights
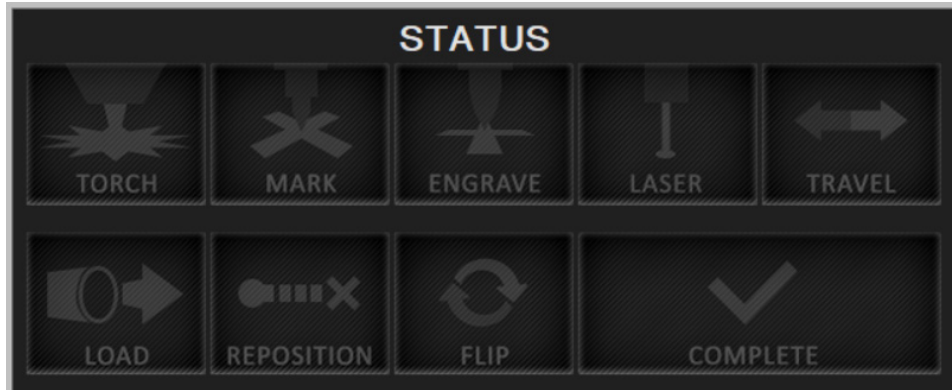
The Action Indicator Lights show which action(s) are currently being preformed by the Dragon machine. These include when the Torch is active, when the Trolley is traveling, when the laser, engraver, or marker are in use, when the machine is in Load Position, when the material needs to be repositioned or flipped, and when the project is complete.

6.2.2 Axes Panel

The Axes Panel show the current position of all machine axes. The house icon listed next to each Axis is the HOME button. Click this to send the Axis to its Home Position. HOME ALL AXES will send all of the axes to their Home Positions one at a time.

6.2.3 Start, Pause, Emergency Stop

The START button is used to start machine operations. Click START once to move the machine into Load Position. Load the material into the machine, and secure it in the Gate and Chuck. Click START a second time to start part production.
Clicking the PAUSE button during operations will pause part production. To resume operations after clicking PAUSE, click the START button.

Use EMERGENCY STOP if the machine does anything unusual.

Important

There are four additional Emergency Stops on the Dragon. All of them will immediately stop all operations when pressed.

6.2.4 Jog Controls
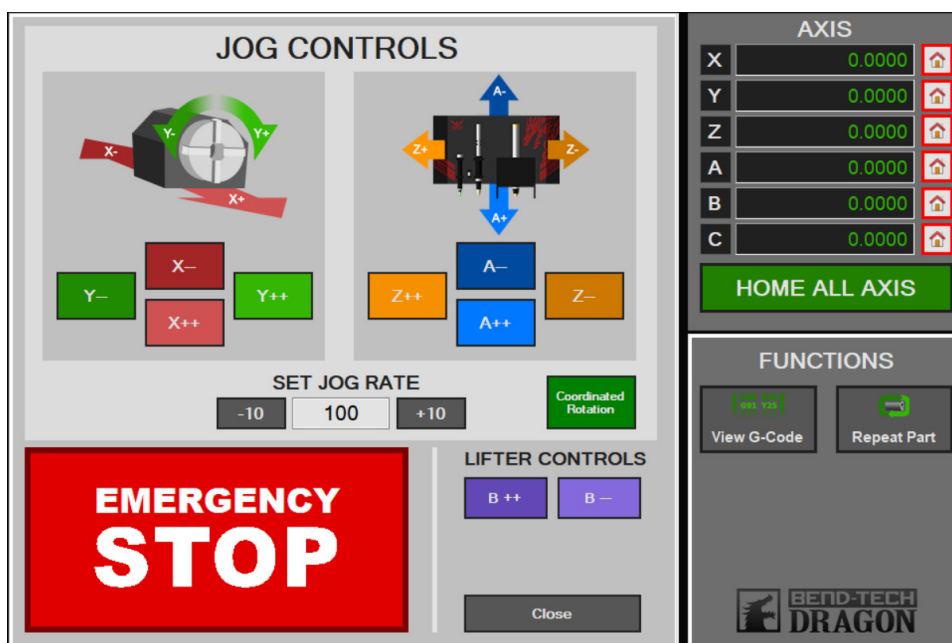
The Jog Controls are accessed by clicking the JOG CONTROLS button. This controls the Trolley movement (X-Axis), the rotation of the Chuck (Y-Axis), the vertical movement of the Toolhead (A-Axis), and the horizontal movement of the Toolhead (Z-Axis). The JOG RATE is adjustable, and controls how fast the trolley moves along the axes.

6.2.5 Coordinated Rotation
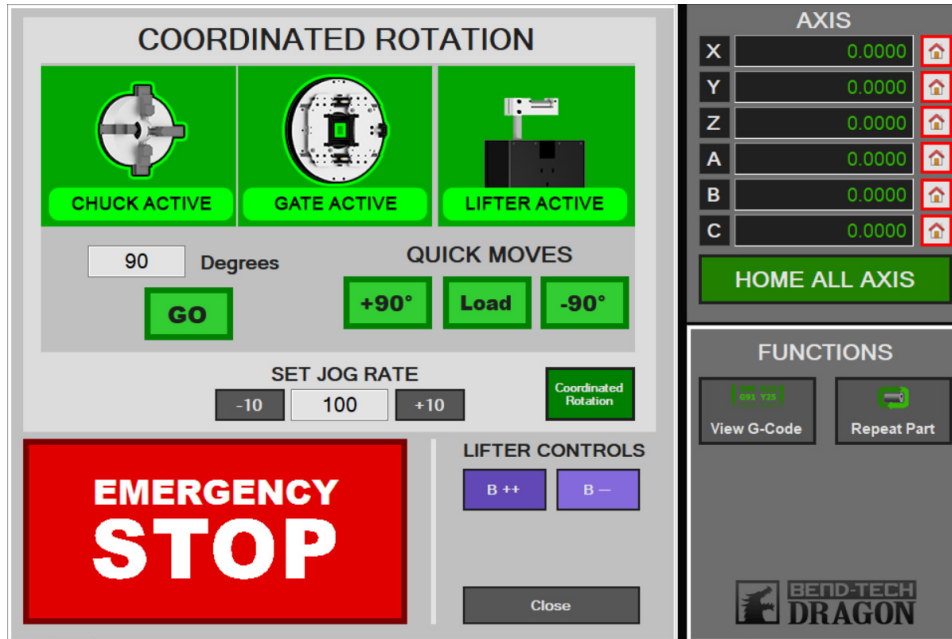
Coordinated Rotation is an option available to Dragon machines equipped with a Powered Gate. To access the Coordinated Rotation controls, select Jog Controls and click COORDINATED ROTATION.

The interface will display a Chuck, Gate, and Lifter diagram. They will be grayed out if INACTIVE and green if one or both are ACTIVE. When both the Chuck and the Gate are active, the Chuck and the Gate will move in tandem when rotated.

When the Lifter is active, the Material Support Lift will automatically adjust for the material loaded into the machine.

! Caution !

If material is already loaded into the Dragon machine equipped with a Powered Gate, both the Chuck and the Gate must be ACTIVE and rotated through Coordinated Rotation. Otherwise, the material will twist, which could damage the machine or injure bystanders.

Degree Rotation

To rotate the Gate and/or Chuck a specific number of degrees, enter the degrees into the DEGREES field.

Quick Moves

QUICK MOVES are plus and minus 90-degrees quick moves. Useful for rotating the gate quickly when accessing the Gate Lead Screws.

Jog Rate

The JOG RATE adjusts the rate that the machine axes move when using the Jog Controls, Lifter Controls, and Coordinated Rotation.

6.2.6 Lifter Controls

The LIFTER CONTROLS control the Material Support Lift. B++ controls the downward movement of the Material Support Lift. B-- controls the upward movement of the Material Support Lift.
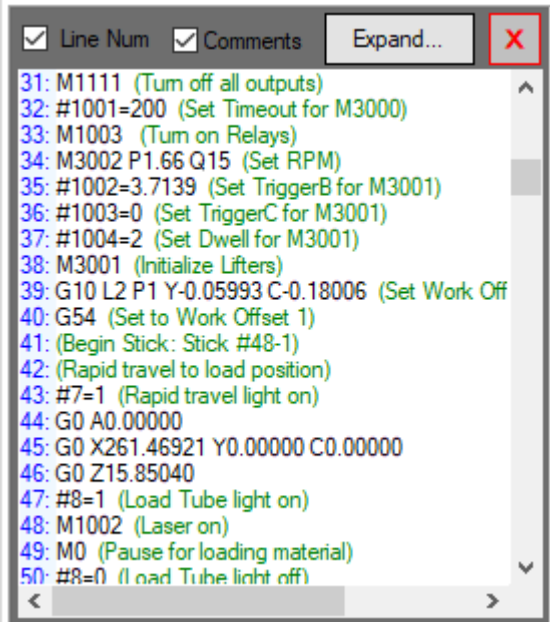
6.2.7 Functions

View G-Code

The VIEW G-CODE functions is used to view the G-Code (with or without line numbers and comments) that was sent to the Dragon Machine. This is not viewable unless G-Code has been sent to the machine.

Repeat Part

Once a project has finished being cut on the Dragon, pressing REPEAT PART will subtract the used material automatically from the stock length and reload the previous part(s) G-Code. This allows the operator to rerun a project without the need to adjust the nest. This will only work if the stock has not been adjusted or removed, and the material length allows for the project to be run again.

# *Support Session*

7.1 How To Send A Support Session

Properly setting up a Support Session with a Bend-Tech Service Technician can be key to keeping the Dragon up and running.

1. Open the Part Design or Nesting Project

If there is a problem with a specific part design or nesting project, open the project before sending a Support Session, then keep the project open after submitting.

2. Open the Support Session

On the Bend-Tech Dragon Home interface, click the Help tab, then click Support Request
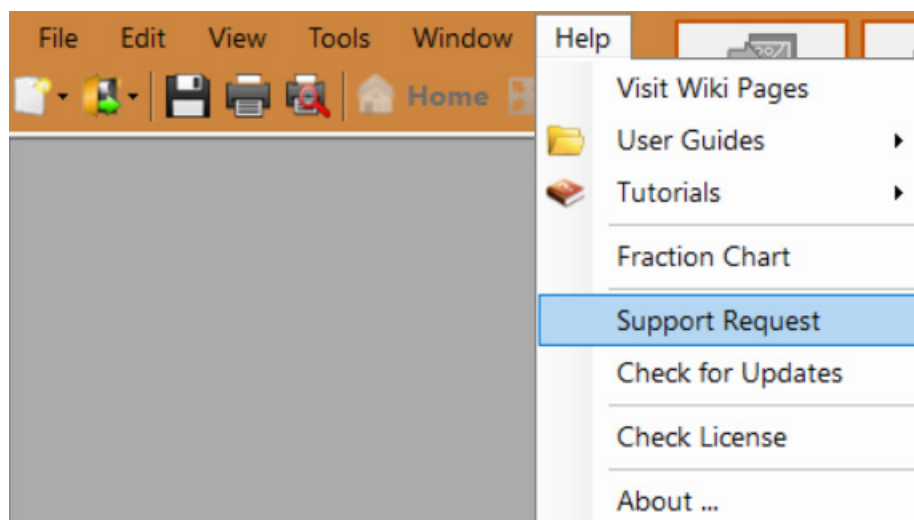
3. Fill in the requested information

Fill in the customer contact form. If a Bend-Tech service technician is familiar with this issue, enter their name in the field provided (this is optional).

4. Provide Details about the Problem

Give a detailed description of the issue in the description of Problem text box. The more information given, the better the Service Technicians can help.

5. Attach the Files

If there is a file associated with the issue, click the folder icon on the right-hand side and upload the design file.

6. Submit the Session

Click Submit.

Bend-Tech LLC

729 Prospect Ave. Osceola, WI 54020
1-651-257-8715

sales@bend-tech.com
www.bend-tech.com