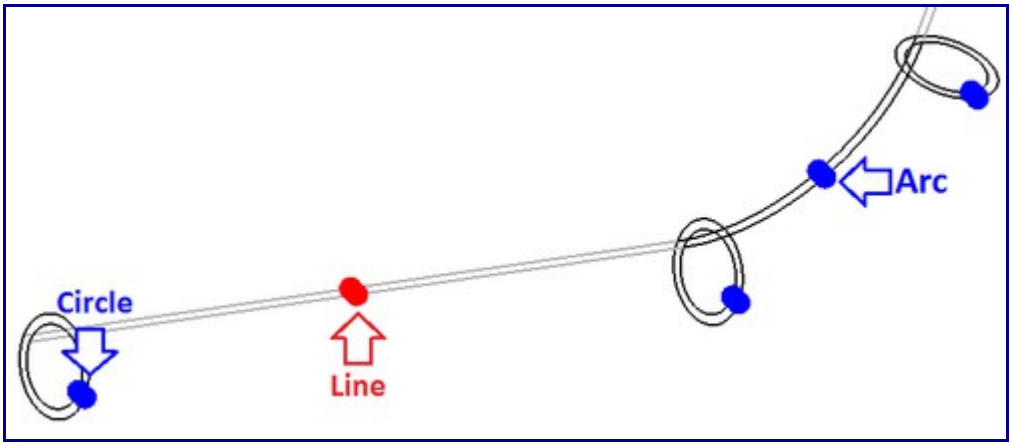
STEP and IGES Import - Part Definition

Introduction



STEP, IGES, BREP, and STL parts can be imported by manually selecting certain features of the part. Parts are represented by lines, circles, and arcs that can be selected using the blue or red pickpoints on each - see image to the right. Depending on the part design and what kind of features are used to represent the part, some methods may be more appropriate than others.

Parts are defined by selecting...

1. Only circles *(recommended)*

2. Circles and Lines *(recommended)*

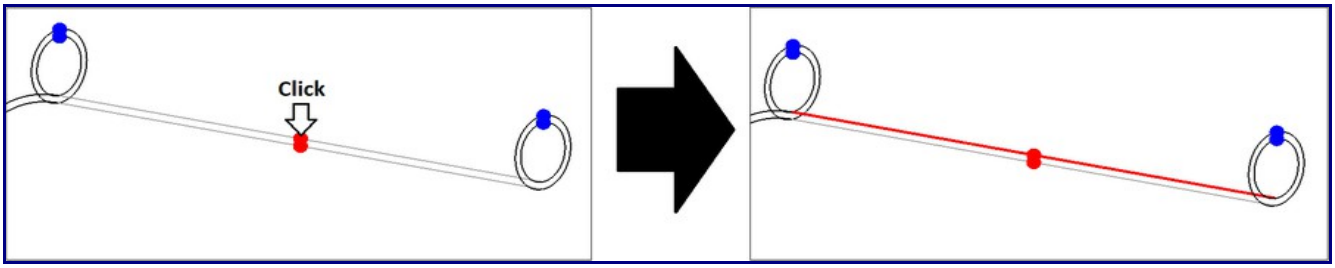
3. Only Lines

4. Arcs and Lines ...*or*

5. Arcs, Lines, and Circles

Selecting Features

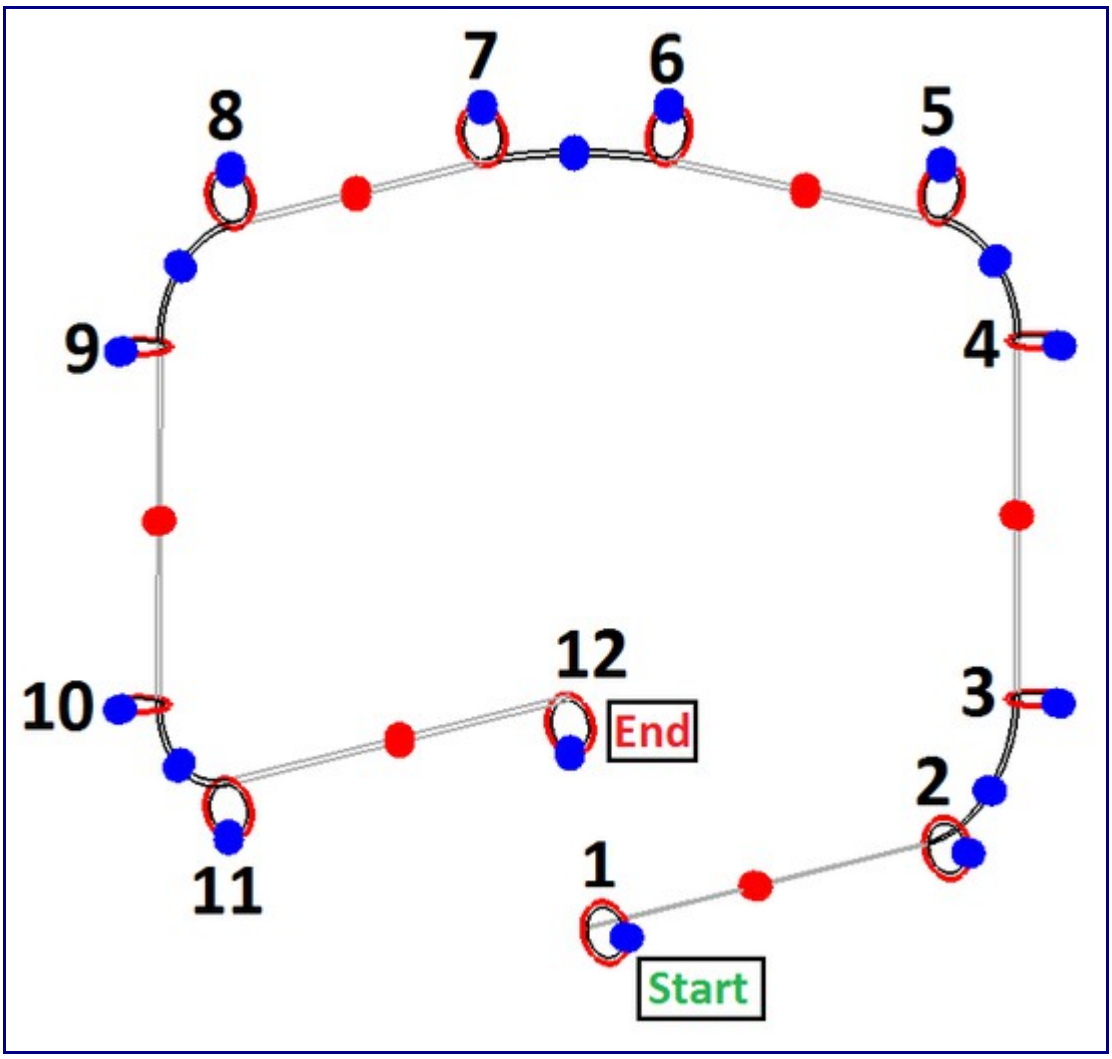
Features must be selected to define a part. **To select a feature**, first click the **start** button. Then click directly on the Pickpoint on the feature to select it. Once selected, the feature will be highlighted in red. See image below.



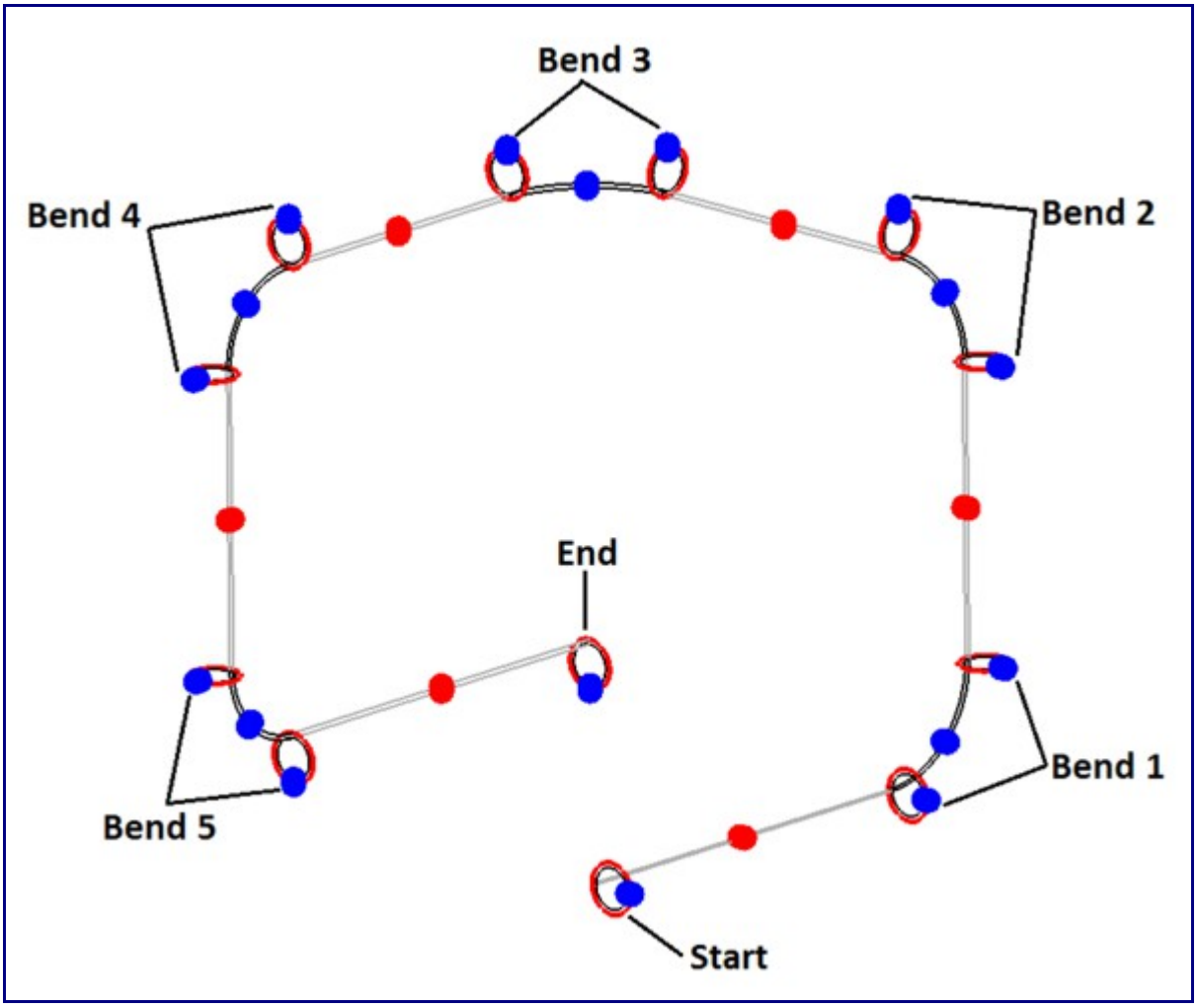
Circles Only

While using this defining method, **only circle features** will be selected. When defining a part using only circles, these rules below must be followed:

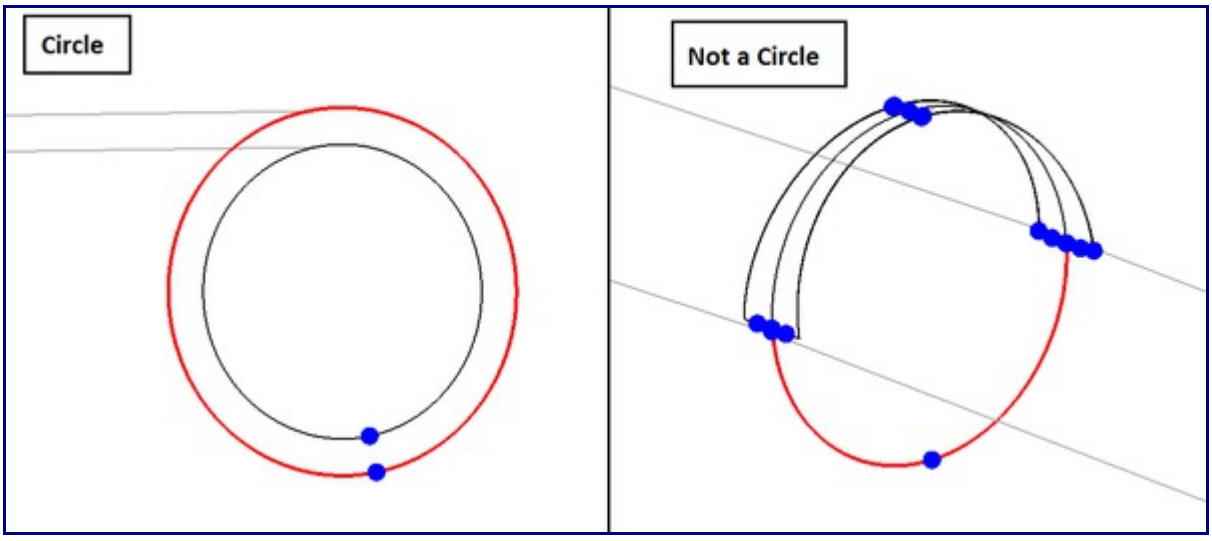
1.) Circles must be selected ***in order***. See image below.



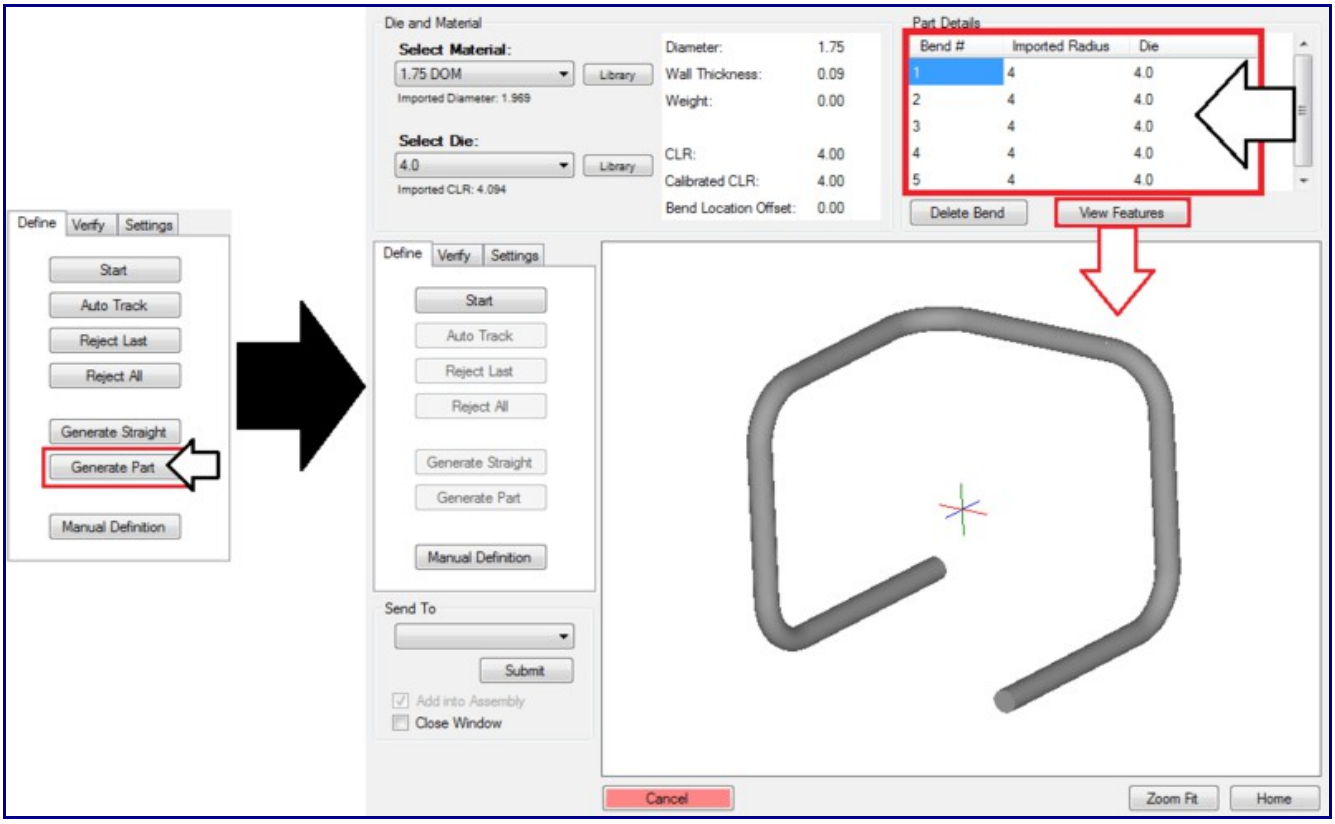
2.) A circle is needed at the **start and end of the part** and at the **start and end of each bend**. See image below.



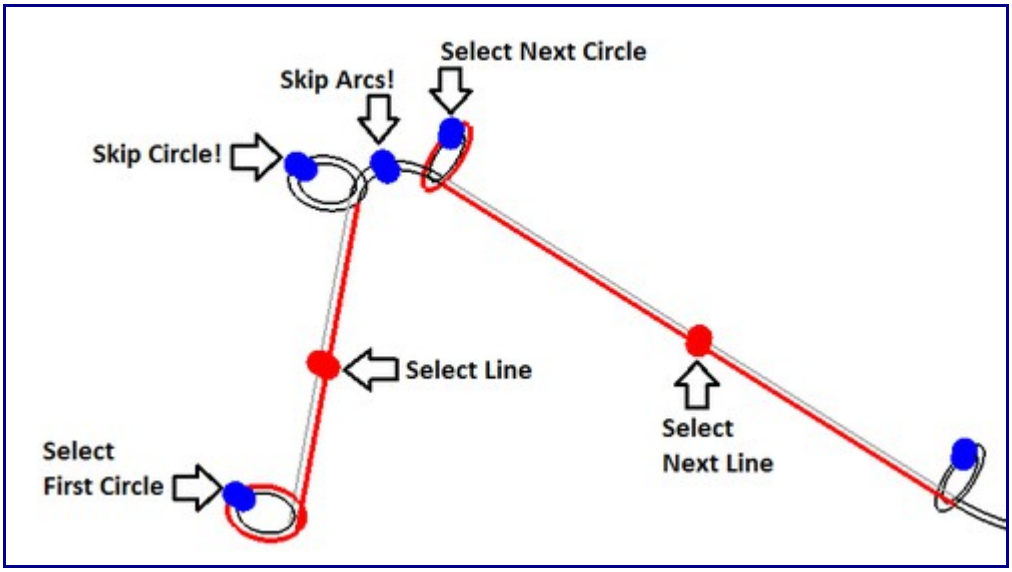
3.) Circles **must be complete** to be considered actual circles.



To define a part using circles only, click the **start** button and select all necessary circles on the part. Once the circles have been selected properly, click the **Generate Part** button to generate the part. The bend information will be shown in the Part Details chart at the top of the window. This chart will list each bend with the imported radius value and the die chosen for each bend. To toggle between viewing the part in the selected material and the part features, use the **Preview Part/View Features** button.



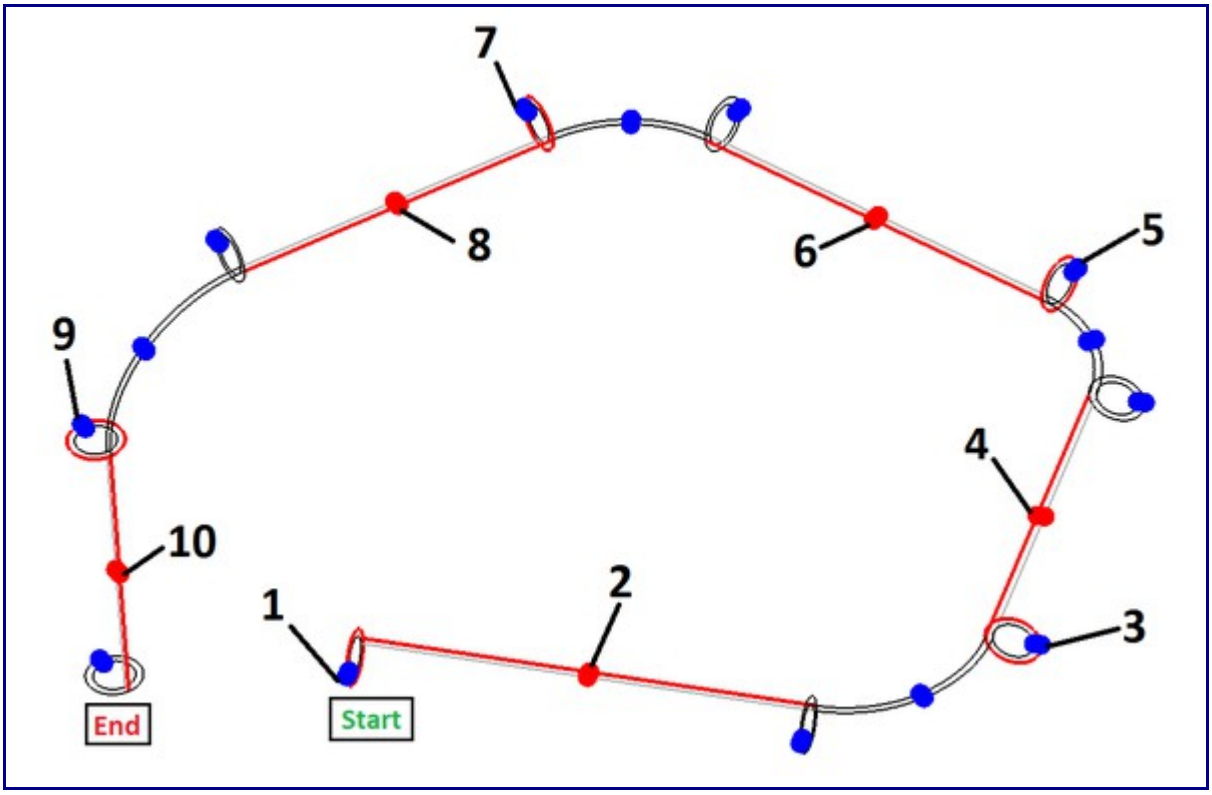
Circles and Lines



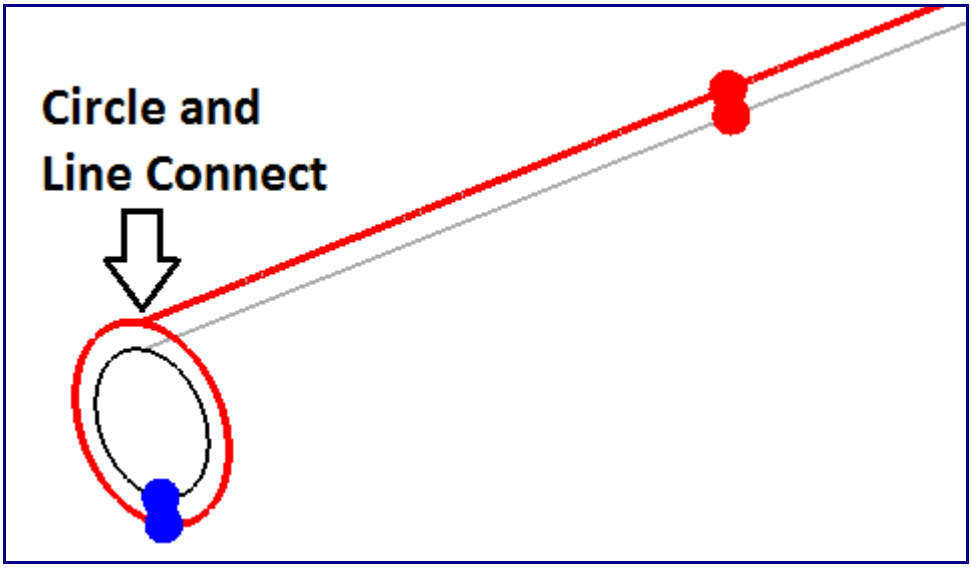
While using this defining method, **circles and lines** will be selected. The lines selected ***do not*** represent the center line of the tube. The first circle will be the circle that defines the starting edge of the tube. Each ***single straight line*** will be paired with ***one circle*** that connects to the beginning of the line. For example, select the first circle then the line that extends from that circle. Skip the circle at the end of the selected line and anything between then continue with the following circle. See the image to the right for an example of how to select the circles and lines.

When defining a part this way, these rules below must be followed:

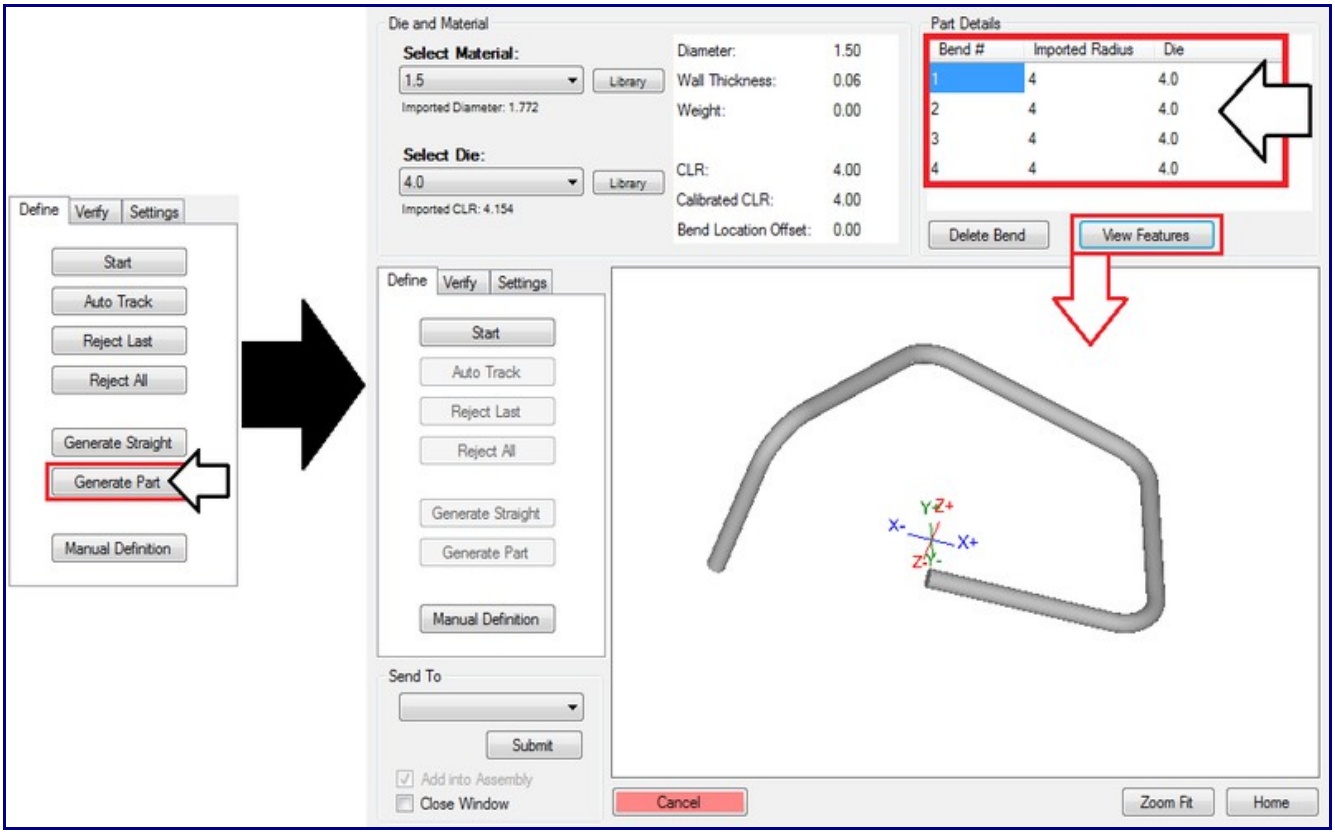
1.) Circles and lines must be selected ***in order starting with the first circle of the part***. See image below.



2.) Each circle and line pair ***must connect***. There cannot be large gaps between the circle and line.



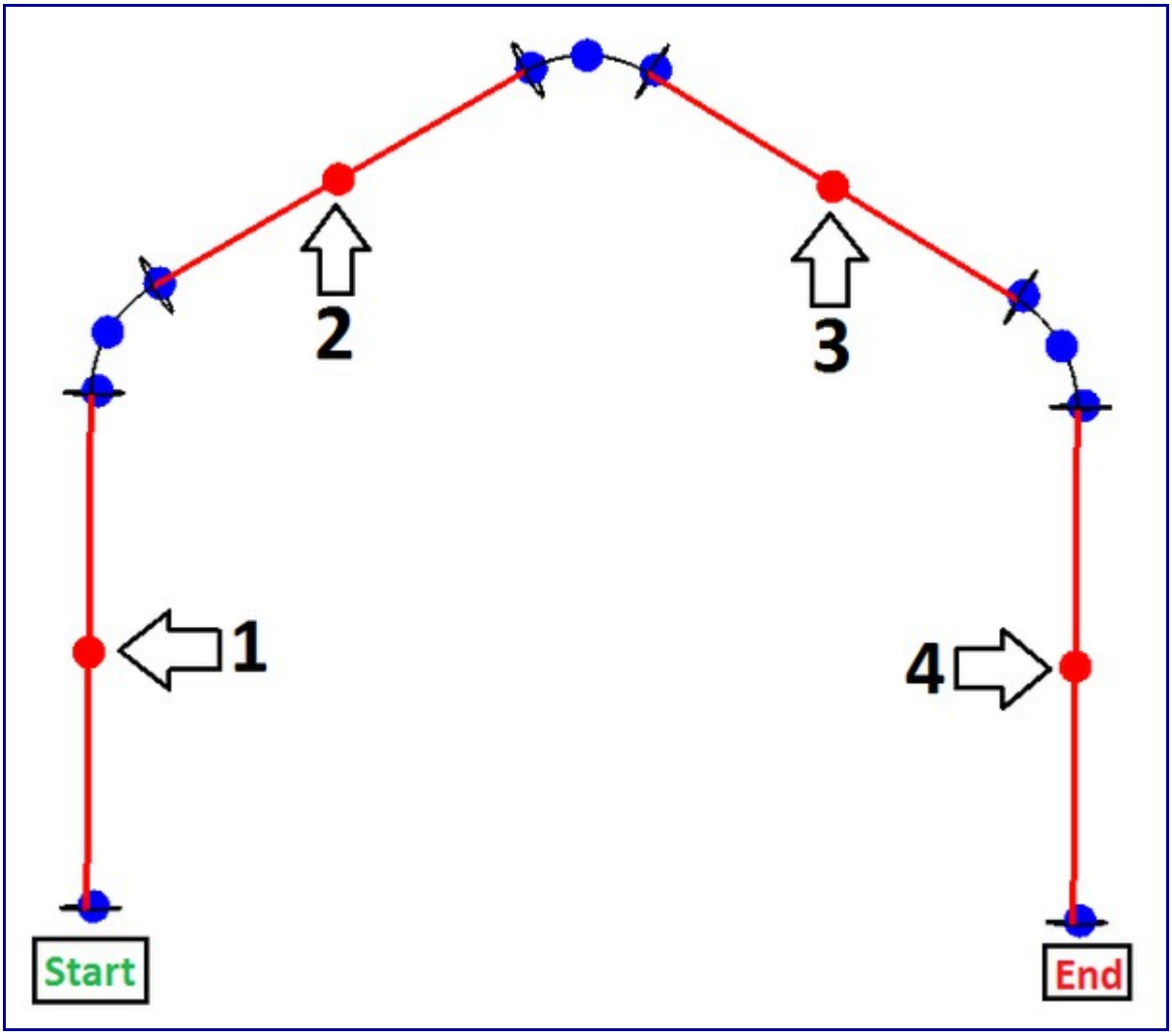
To define a part using circles and lines, click the **start** button and select all necessary circles and lines on the part. Once these features have been selected, click the **Generate Part** button to generate the part. The bend information will be shown in the Part Details chart at the top of the window. This chart will list each bend with the imported radius value and the die chosen for each bend. To toggle between viewing the part in the selected material and the part features, use the **Preview Part/View Features** button.



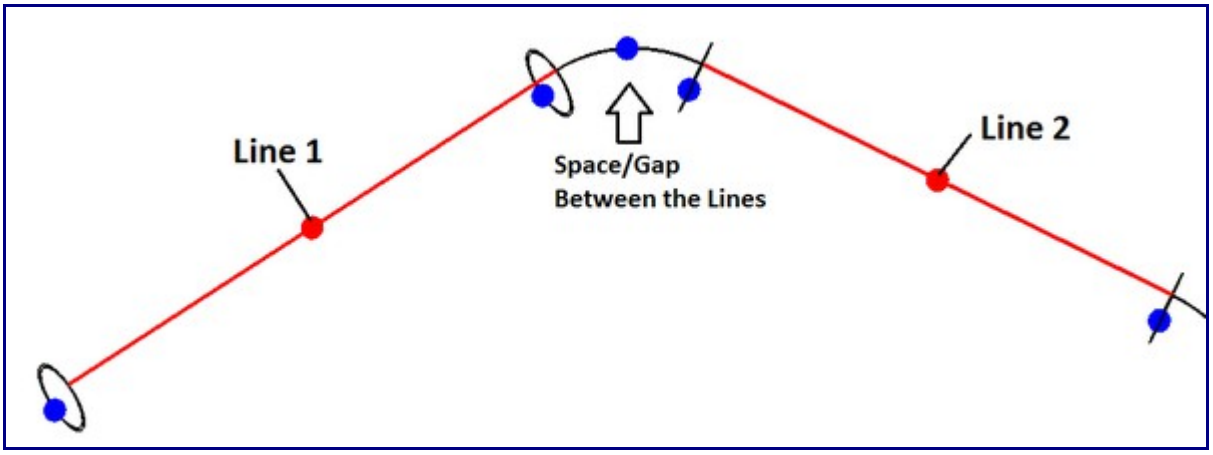
Lines Only

While using this defining method, **only straight line features** will be selected. The lines selected will represent the ***center line*** of the part. To successfully define a part using this method, these rules must be followed:

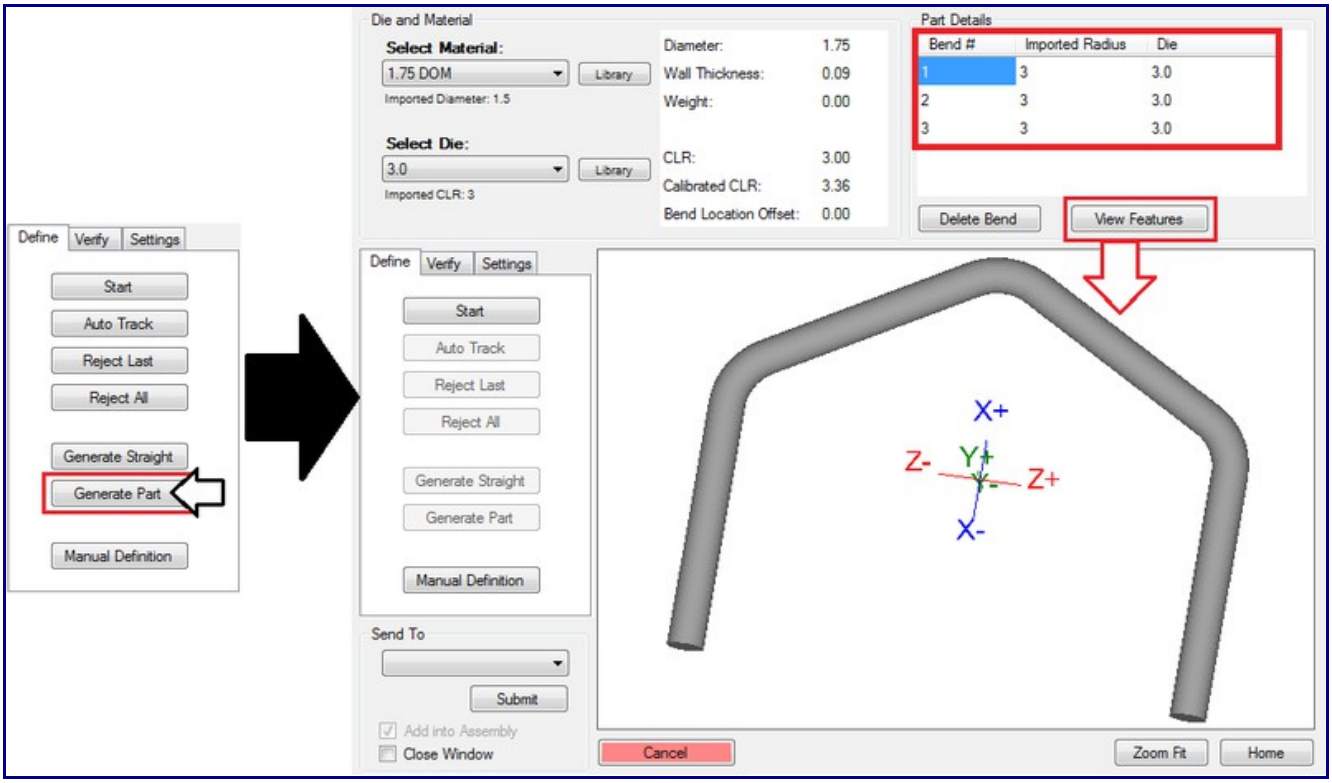
1.) The lines must be selected ***in order***.



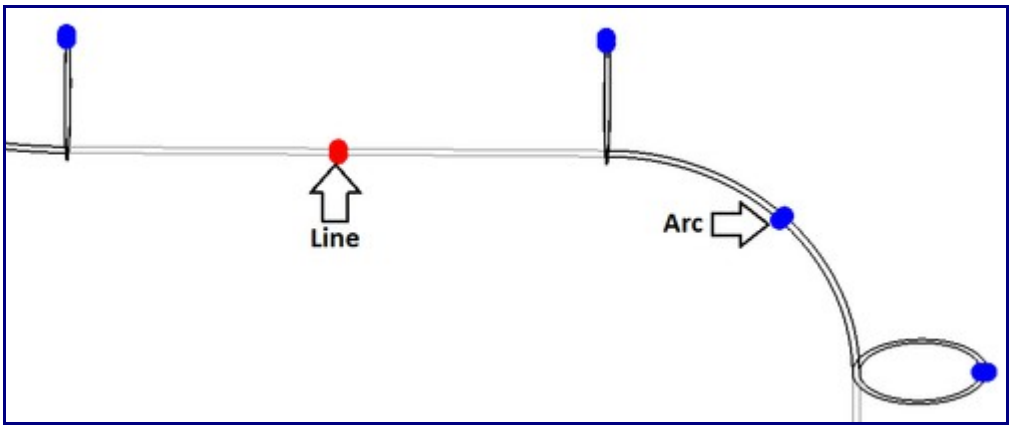
2.) The lines ***do not have to connect***. Spaces are allowed between the selected lines.



To define a part using lines, click the **start** button and select the lines on the part. Once they have been selected, click the **Generate Part** button to generate the part. The bend information will be shown in the Part Details chart at the top of the window. This chart will list each bend with the imported radius value and the die chosen for each bend. To toggle between viewing the part in the selected material and the part features, use the **Preview Part/View Features** button.

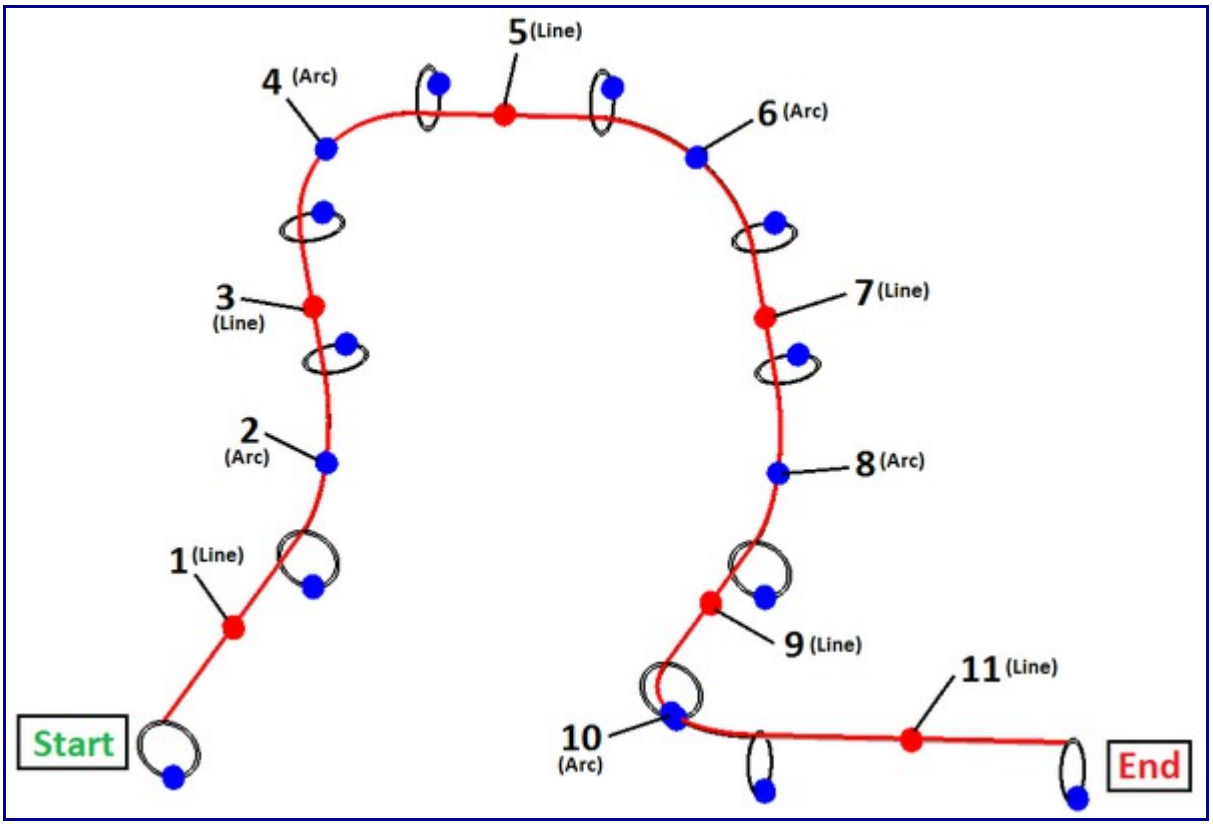


Arcs and Lines

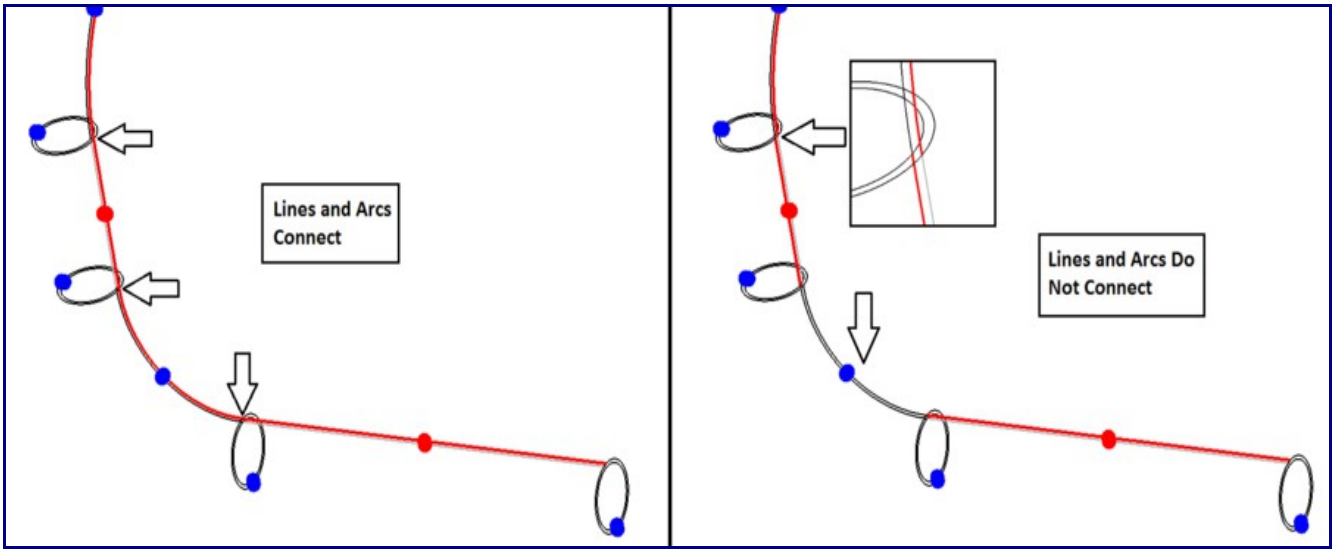


While using this defining method, **straight lines and arc features** will be selected. The selected lines and arcs will represent the ***center line*** of the part. To successfully define a part using arcs and lines, these rules must be followed:

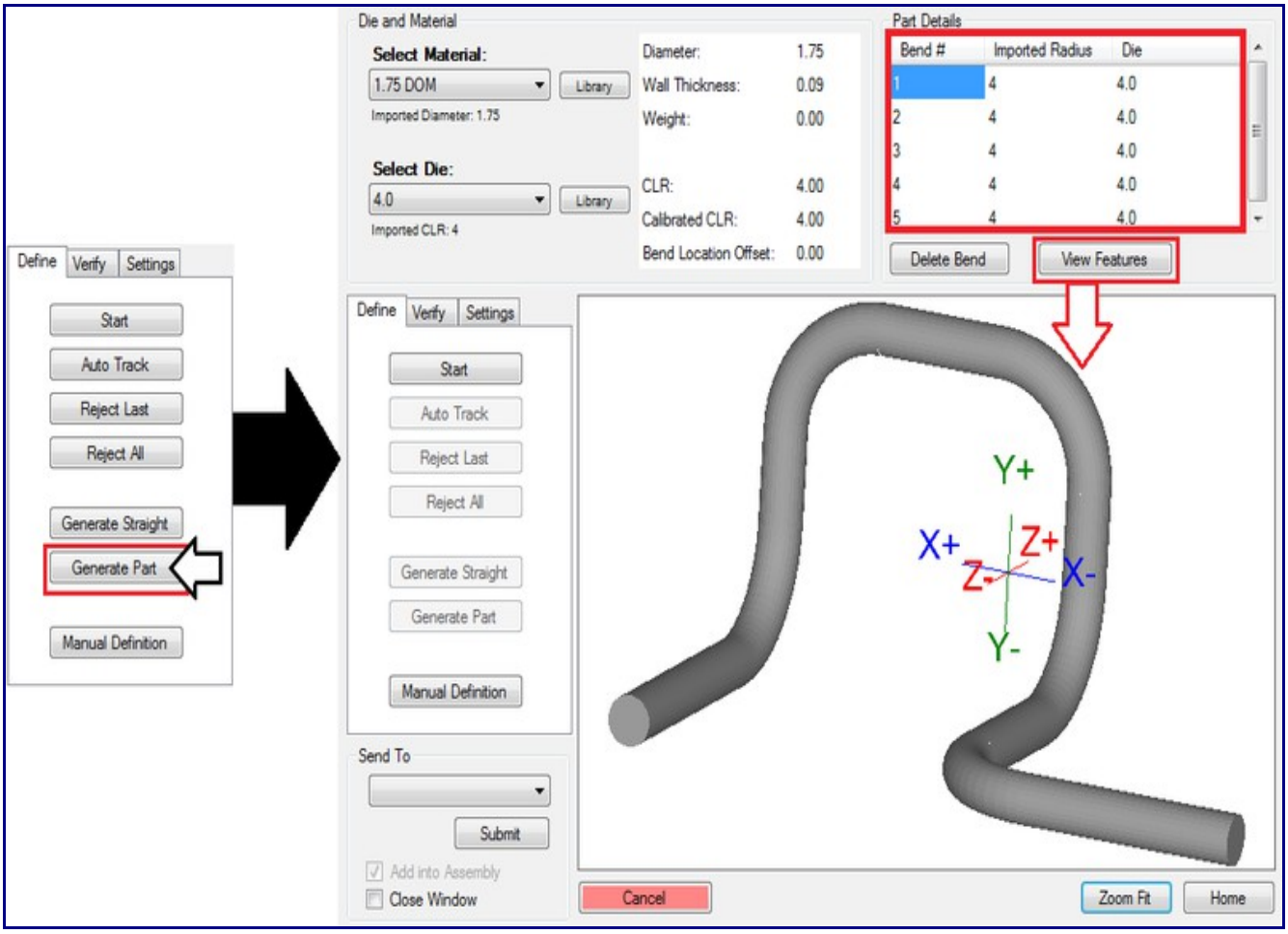
1.) The lines an arcs must be selected ***in order***.



2.) The selected line and arc features **must connect**. There cannot be large gaps between the lines and arcs.



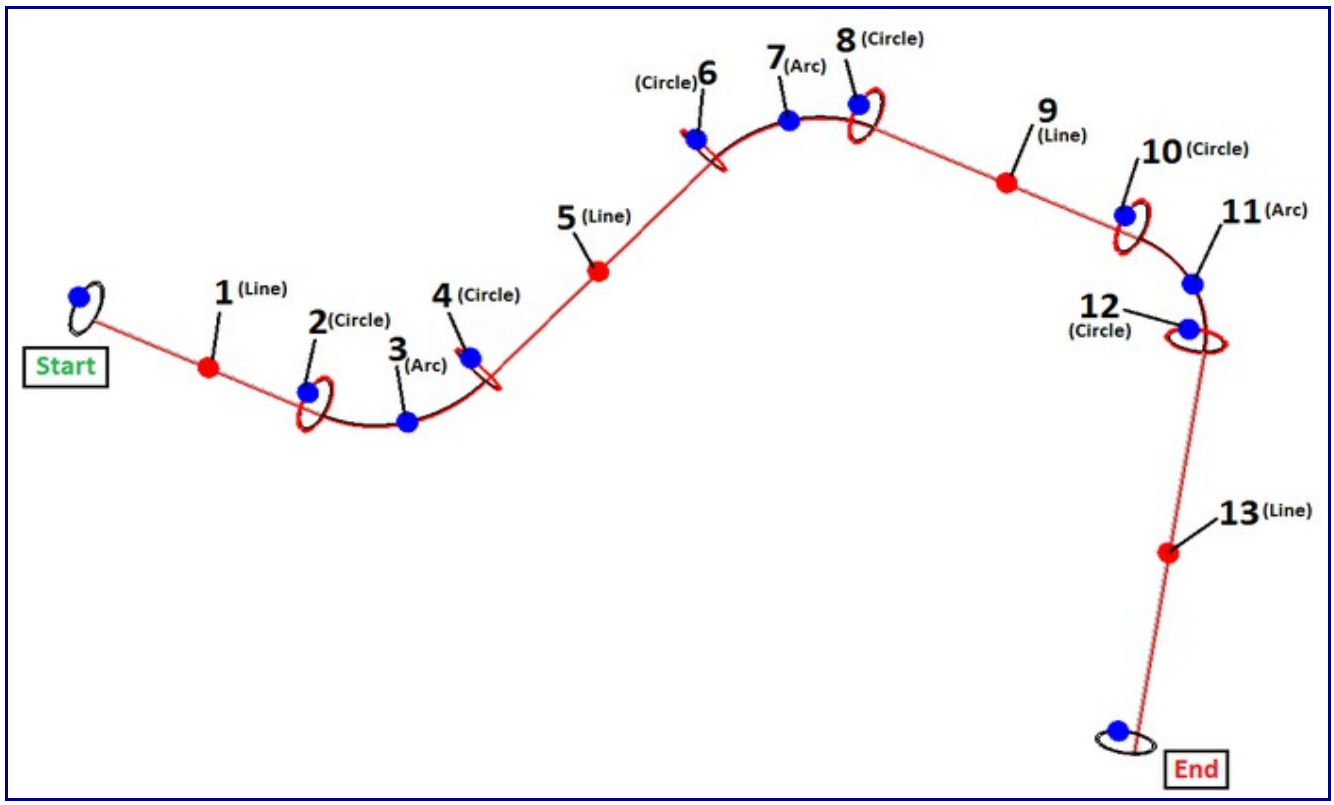
To define a part using lines, click the **start** button and select the lines and arcs. Once they have been selected, click the **Generate Part** button to generate the part. The bend information will be shown in the Part Details chart at the top of the window. This chart will list each bend with the imported radius value (based on the radius of the selected arc for that particular bend) and the die chosen for each bend. To toggle between viewing the part in the selected material and the part features, use the **Preview Part/View Features** button.



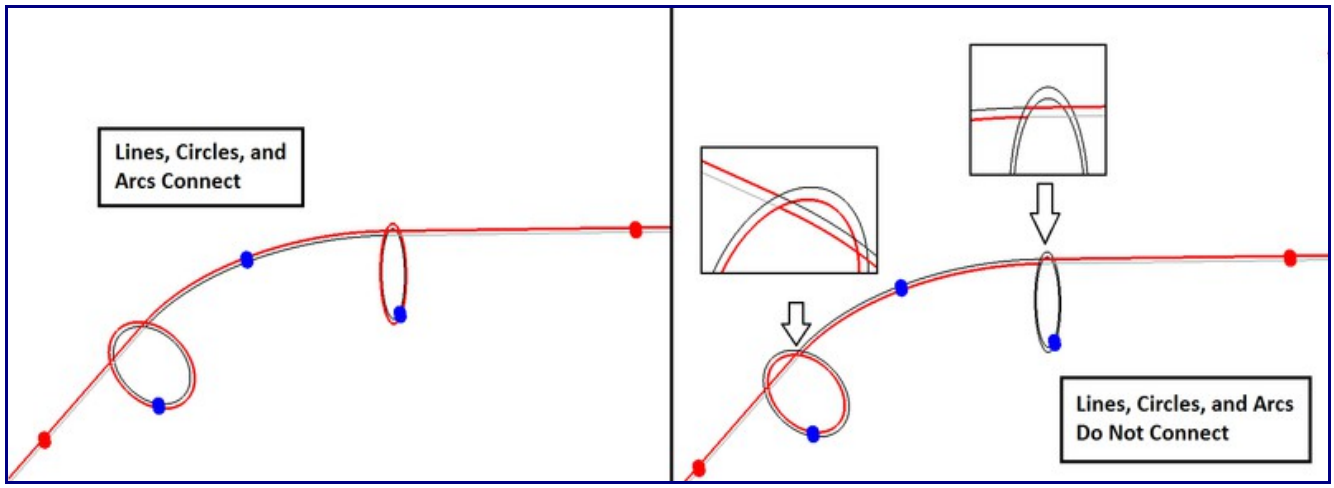
Circles, Arcs, and Lines

While defining parts using this method, **line, circle, and arc features** will be selected. The selected lines and arcs will not represent the center line of the part. To successfully define a part using circles, arcs, and lines, these rules must be followed:

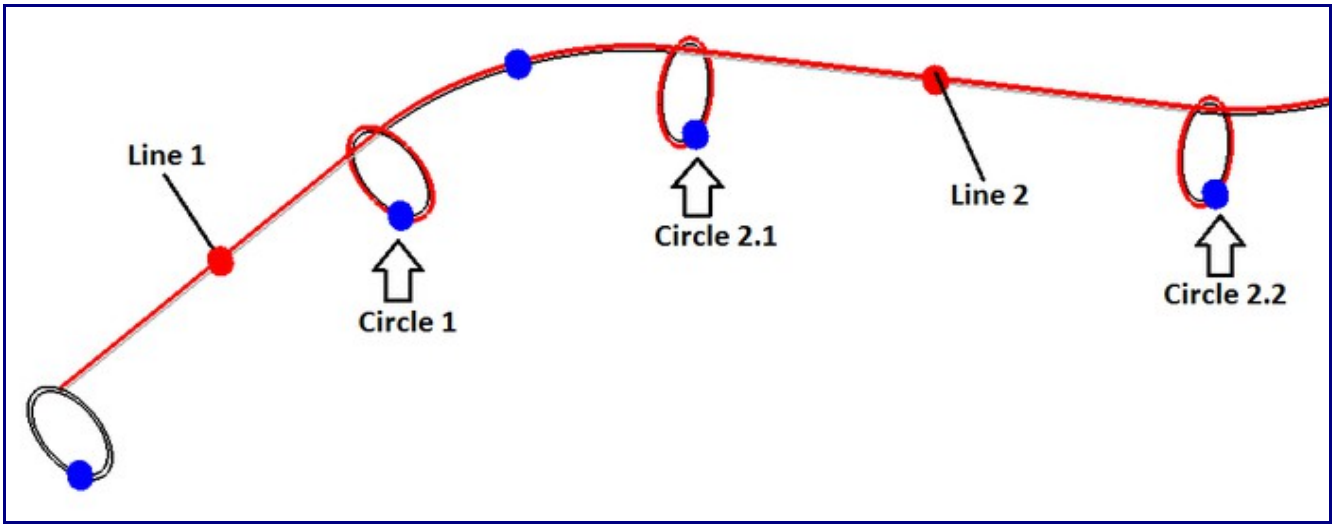
1.) The circles, lines, and arcs must be selected ***in order***.



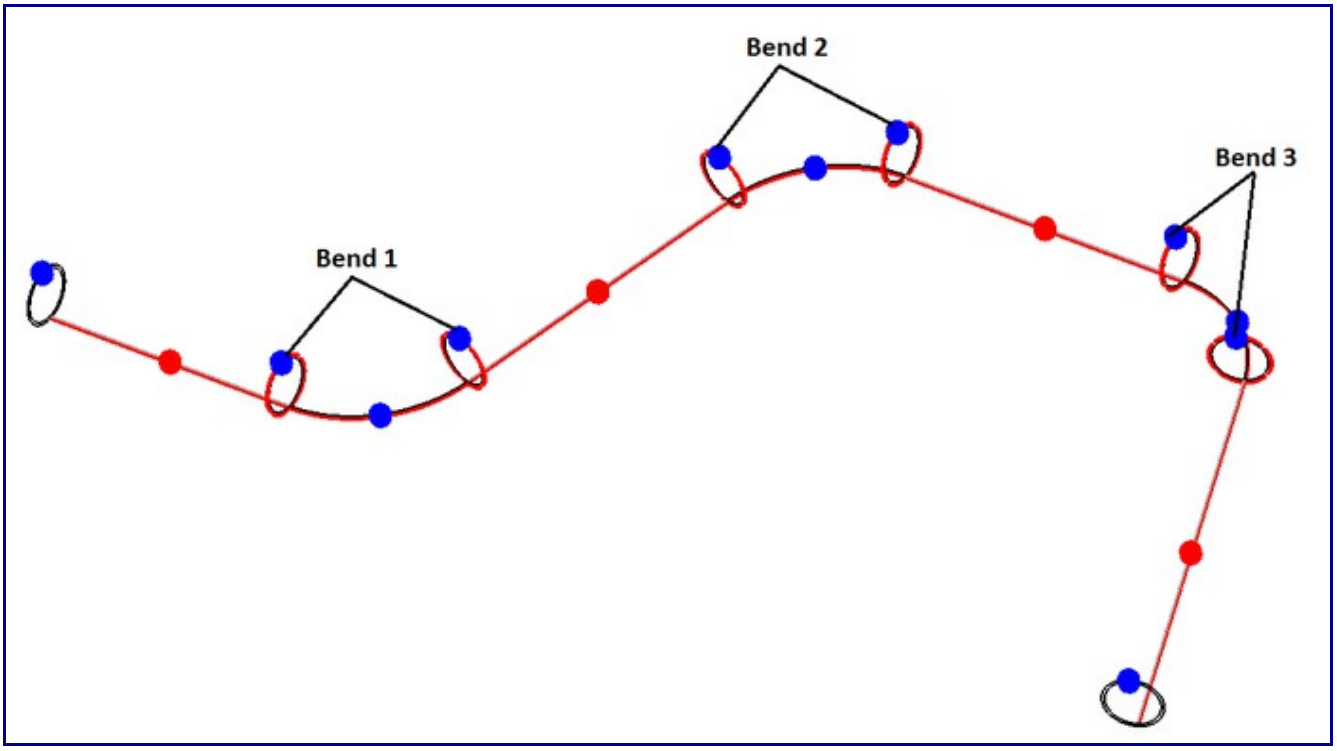
2.) Each circle, line, and arc ***must connect***.



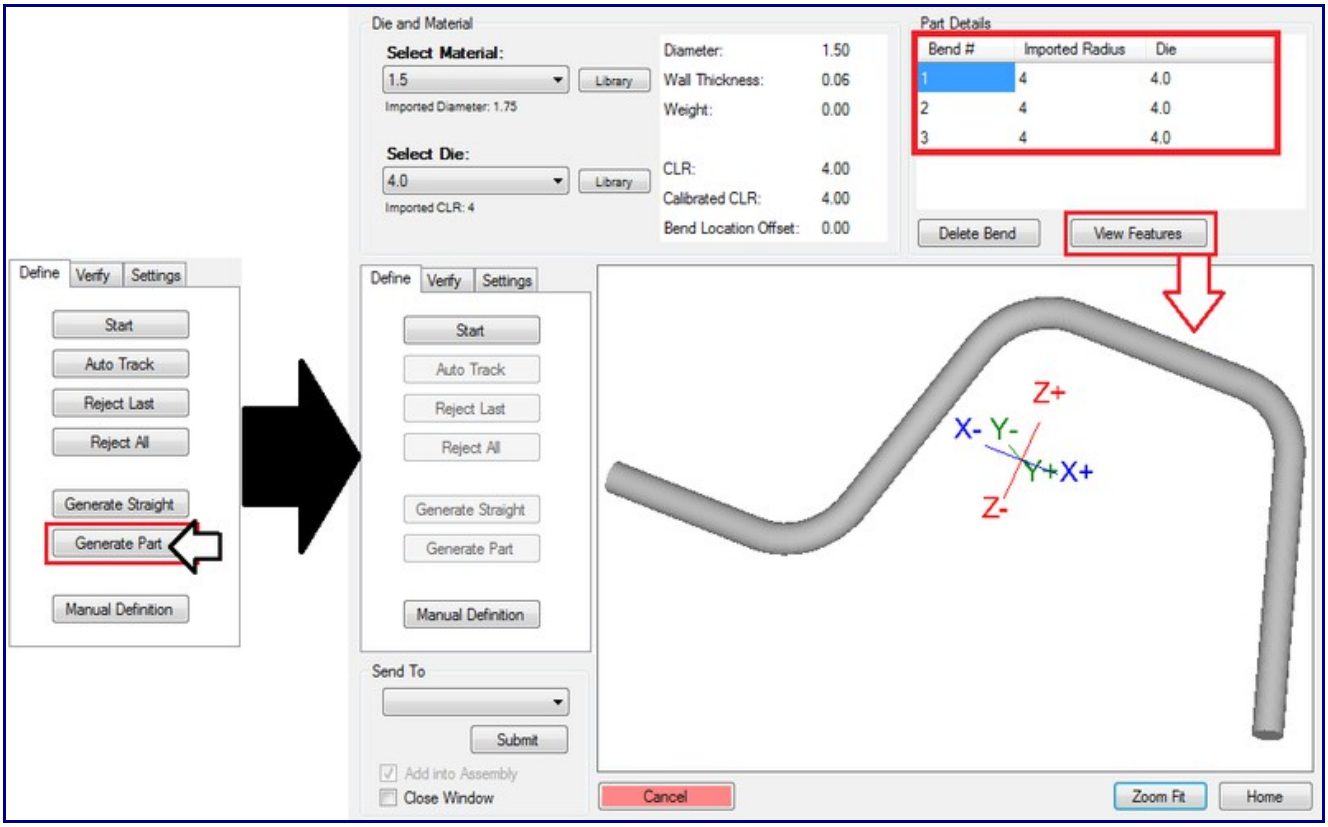
3.) Each **straight line** needs **one circle** at the **start *or* end**. Straight lines can have circles on both ends, but it is not required. For example, in the image below, line 1 has the circle at the end (Circle 1) selected, but line 2 has both the start and end circles (Circle 2.1 and circle 2.2) selected because it is in between two arcs.



4.) Each arc needs a **circle on the start and end of each arc**.



To define a part using lines, circles, and arcs, click the **start** button and select the features. Once they have been selected, click the **Generate Part** button to generate the part. The bend information will be shown in the Part Details chart at the top of the window. This chart will list each bend with the imported radius value (based on the radius of the selected arc for that particular bend) and the die chosen for each bend. To toggle between viewing the part in the selected material and the part features, use the **Preview Part/View Features** button.


Define
Verify
Settings
Start
Auto Track
Reject Last
Reject All
Generate Straight
Generate Part
Manual Definition
Die and Material
Select Material:
1.5
Library
Imported Diameter: 1.75
Select Die:
4.0
Library
Imported CLR: 4
Diameter: 1.50
Wall Thickness: 0.06
Weight: 0.00
CLR: 4.00
Calibrated CLR: 4.00
Bend Location Offset: 0.00
Part Details
Bend # Imported Radius Die
1 4 4.0
2 4 4.0
3 4 4.0
Delete Bend
View Features
Send To
Submit
Add into Assembly
Close Window
Z+
X- Y-
Y+X+
Z-
Cancel
Zoom Fit
Home