

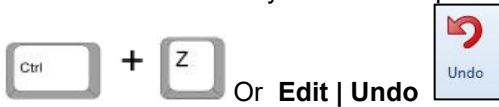
## Open



Displays the Open dialog to browse for and open an existing drawing file.  
To open a drawing, select the drawing file that you want to open and click the **Open** button.

## Undo

This command allows you to undo a previous command.



When this command is used, the previous change to the current drawing will be undone.

Undo will not work after using the following commands:

- Commands relating to the saving of data.
- Commands relating to the input and output of data.
- Commands that affect the view.
- Commands that affect work planes.
- Commands that report information.

## Cancelling A Command

To cancel a command, use one of the following methods:

- Right-click in the Drawing Area.

- Press  .

## Redo

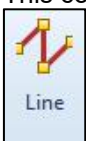
This command allows you to redo a previous command if the Undo command has been used.



When this command is used, the previous command will be redone.  
This command is only enabled after the Undo command has been used.

## Line

This command allows you to create one or more line segments in the Drawing Area.



### **Geometry | Line**

When this command is used, you will be prompted to enter the coordinates of the 'From' point. This point is the start of the line.

To select the point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to select the 'To' point. Repeat the method used for the 'From' point. Any additional points selected are 'To' points. Repeat the method used for the 'From' point.

If you do not know the value for a co-ordinate, press



or click



A dialog is displayed with alternative options to define the parameters of the line.

Depending on the options selected, alternative prompts will be displayed in the Input Bar.

## Circle - Centre + Diameter

This command allows you to create a circle in the Drawing Area by specifying the diameter and selecting the centre point.



Geometry | Circle ► Centre + Diameter

When this command is used, you will be prompted to enter the circle diameter. Type a value into the Input Bar and click **OK**.

You will then be prompted to enter the circle centre.

To select the point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Type the coordinate values into the Input Bar and click **OK**.

## Circle - Centre + Radius

This command allows you to create a circle in the Drawing Area by specifying the radius and selecting the centre point.



Geometry | Circle ► Centre + Radius

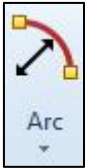
When this command is used, you will be prompted to enter the circle radius. Type a value into the Input Bar and click **OK**. You will then be prompted to enter the circle centre.

To select the point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Type the coordinate values into the Input Bar and click **OK**.

## Arc - 2 Points + Radius

This command allows you to create an arc in the Drawing Area by selecting the start and end points and defining the radius.



### Geometry | Arc ► 2 Points + Radius

When this command is used, you will be prompted to enter the start point of the arc.

To select the point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area or
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to enter the end point. Repeat the method used for the start point.

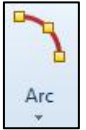
Arcs can be drawn by selecting the points in a counterclockwise or clockwise direction. Therefore, the relative direction in which you position the start and end points will determine the appearance of the arc.

You will then be prompted to enter the arc radius.

Type a value into the Input Bar and click **OK**.

## Arc - Start + Second + End Points

This command allows you to create an arc in the Drawing Area by selecting the start point, a second point with which the arc coincides and the end point.



### Geometry | Arc ► Start + Second + End Points

When this command is used, you will be prompted to enter the start point of the arc.

To select the point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a Snap function
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to enter the second point. Repeat the method used for the start point.

The second point can be positioned anywhere along the arc.

You will then be prompted to enter the end point.

## Rectangle

This command allows you to create a rectangle in the Drawing Area by specifying two points at diagonally opposite corners.



### Geometry | Rectangle

When this command is used, you will be prompted to 'Pick First Corner'.

To select the point, use one of the following methods:

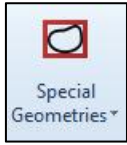
- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to 'Pick Second Corner'. Repeat the method used for the first corner.

The position of the second corner relative to the first corner determines the position and size of the rectangle.

## Enclosing Rectangle

This command allows you to enclose an existing element(s) in the Drawing Area inside a rectangle.



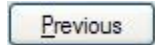
Geometry | Special Geometries ► Enclosing Rectangle

This command will only be enabled if there is existing geometry in the Drawing Area.

When this command is used, you will be prompted to 'Select Geometries'.

Select an element(s) in the Drawing Area using the mouse pointer.

The command buttons in the Input Bar can also be used to select elements:




Selects the most recently created element.



Selects all elements in the current drawing.

If more than one geometry is selected, a single rectangle will be created that encloses them all.

When the required elements have been selected, use one of the following methods to proceed:

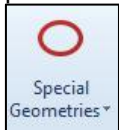
- Click .

- Press .

- Right-click in the Drawing Area.

## Special Geometries - Ellipse

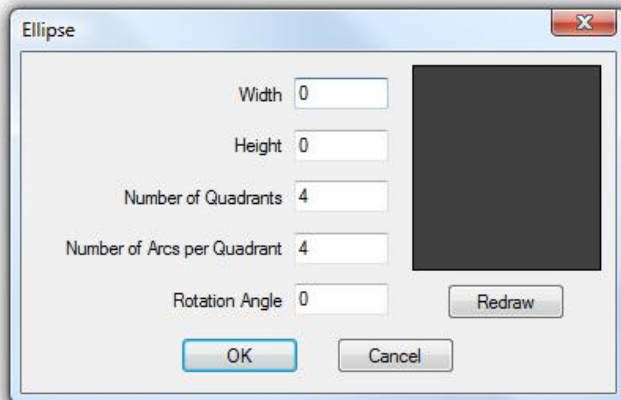
This command allows you to create an ellipse in the Drawing Area by defining the width, height, quadrant parameters and rotation angle.



Geometry | Special Geometries ► Ellipse

### Ellipse Dialog

When this command is used, the Ellipse dialog will be displayed:



This dialog contains the following options:

- **Width** - The width, in units, of the ellipse.

- **Height** - The height, in units, of the ellipse.
- **Number of Quadrants** - The number of quadrants of the ellipse that will be drawn. Enter a value between 1 - 4.

A quadrant is a quarter of the perimeter of the ellipse. Entering a value lower than 4 will result in a partial ellipse.

- **Number of Arcs per Quadrant** - The number of arcs that will make up each quadrant.

A higher value will result in a smoother curve. However, additional NC code will be generated for the machining of each arc.

- **Rotation Angle** - The angle, in degrees, by which the ellipse will be rotated. The ellipse will be rotated in an counterclockwise direction.

Entering a negative value will rotate the ellipse in a clockwise direction.

If the values for any parameters are changed, clicking **Redraw** refreshes the preview in the dialog.

## Creating an Ellipse

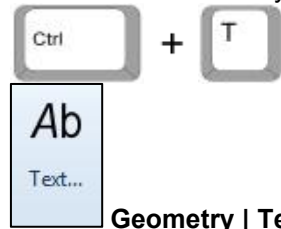
To create the ellipse as geometry in the Drawing Area, click **OK**.

To create the geometry, use one of the following methods to define the centre point:

- Click the mouse pointer at the required position in the Drawing Area.
- Type the coordinate values into the Input Bar and click **OK**.

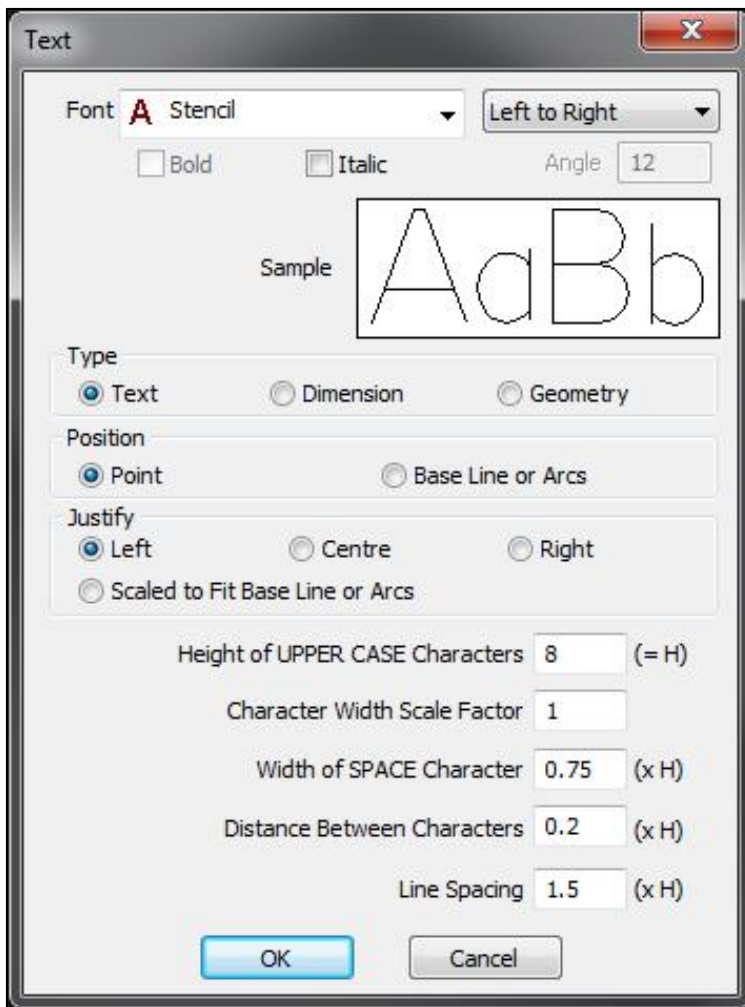
## Text

This command allows you to create text in the Drawing Area.



## Text Dialog

When this command is used, the Text dialog is displayed:



This dialog contains the following options:

**Font** - Enables you to define the font. Select an option from the drop-down menu. Script fonts contain overlapping geometries that cannot be used when engraving.

**Note:** The option selected in the drop-down menu next to the font determines the direction in which the text will be written:

- **Left to Right** - The text will be written from left to right.
- **Right to Left** - The text will be written from right to left.

**Bold** - If checked, the font will have a heavier line weight.

**Italic** - If checked, the text will be in italic.

**Angle** - The angle, in degrees, at which the text will be created in the Drawing Area.  
Text is rotated in a clockwise direction.

**Sample** - Displays a preview image of the text that will be created using the current settings.

**Type** - Allows you to select the layer on which the text will be created:

- **Text** - If selected, text will be created on the Text layer.
- **Dimension** - If selected, text will be created on the Dimension layer.
- **Geometry** - If selected, text will be created on the Geometry layer.

**Position** - Enables you to select the method by which the text will be created:

- **Point** - If selected, text will be created at a defined point.
- **Base Line or Arcs** - If selected, text will be created along an existing line or arc.

**Justify** - Allows you to specify the justification of the text:

- **Left** - If selected, text will be justified to the right of the point at which it is created.
- **Centre** - If selected, text will be centred on the point at which it is created.
- **Right** - If selected, text will be justified to the left of the point at which it is created.

- **Scaled to Fit Base Line or Arcs** - If selected, text will be scaled to fit along the selected line or arc.
- **Height of UPPER CASE Characters** - The height, in units, of uppercase characters.
- **Character Width Scale Factor** - The ratio of width to height. For example, a value of 1 will result in characters that have the same width and height. A value of 2 will result in characters that are twice as wide as their height.
- **Width of SPACE Character** - The width of space characters as a multiple of the text height.
- **Line Spacing** - The distance between separate lines of text as a multiple of the text height.

When the relevant options have been selected, click **OK**.

## Creating Text

The method for creating text will vary depending on the option selected for Position:

You will be prompted to select the "Start Point". Use one of the following methods:

- Click a point in the Drawing Area.
- Type the coordinates into the Input Bar.

You will then be prompted to "Enter Text".

Type text into the Input Bar and click **OK**.

### Base Line or Arcs

You will be prompted to select "Construction Base Line/Arcs".

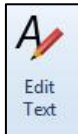
To select an element, click on it.

You will then be prompted to "Enter Text".

Type text into the Input Bar and click **OK**.

## Edit Text

This command enables you to edit existing text and dimensions in the Drawing Area.



**Geometry | Text | Edit Text**

**Note:** This command only applies to elements on the Text and Dimension layers. Therefore, if a text element has been changed to a geometry element, this command will not affect it.

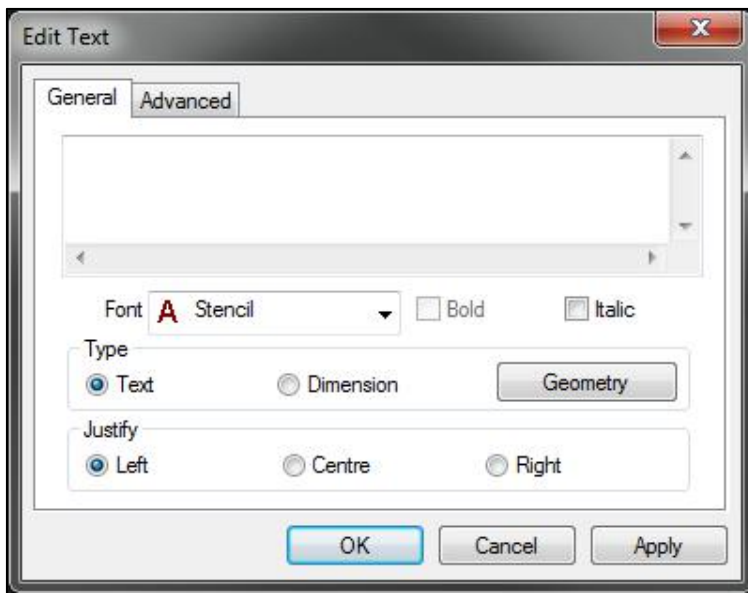
When this command is used, the Edit Text dialog is displayed:

This dialog has two tabs and contains the following options:

Ensure that all of the relevant options are selected from all tabs before clicking **OK**.

Clicking **Cancel** at any time aborts the command.

### General Tab



The text box contains the selected text which can be edited, if required.

**Font** - Enables you to define the font. Select an option from the drop-down menu.

**Bold** - If checked, the font will have a heavier line weight.

**Italic** - If checked, the text will be italic.

**Type** - Allows you to select the layer on which the text will be created:

- **Text** - If selected, text will be created on the Text layer.
- **Dimension** - If selected, text will be created on the Dimension layer.
- **Geometry** - Moves text to the Geometry layer.

**Justify** - Allows you to specify the justification of the text:

- **Left** - If selected, text will be justified to the right of the point at which it is created.
- **Centre** - If selected, text will be centred on the point at which it is created.
- **Right** - If selected, text will be justified to the left of the point at which it is created.

## Advanced Tab

Height of UPPER CASE Characters	8	(= H)
Character Width Scale Factor	2	
Width of SPACE Character	0.75	(x H)
Distance Between Characters	0.2	(x H)
Line Spacing	1.5	(x H)
Angle	12	

**Height of UPPER CASE Characters** - The height, in units, of uppercase characters.

**Character Width Scale Factor** - The ratio of width to height. For example, a value of 1 will result in characters that have the same width and height. A value of 2 will result in characters that are twice as wide as their height.

**Width of SPACE Character** - The width of space characters as a multiple of the text height.

Press the spacebar when typing to insert a space character.

**Distance Between Characters** - The distance between each character as a multiple of the text height.

**Line Spacing** - The distance between separate lines of text as a multiple of the text height.



## Distance/Angle

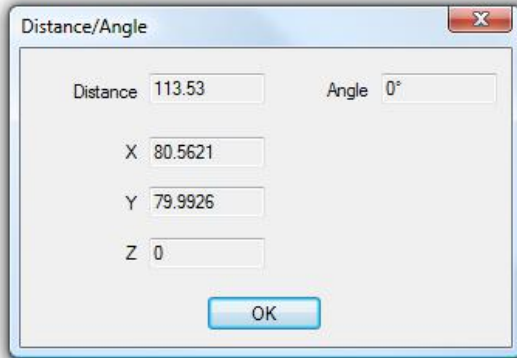
This command is used to measure the distances and angle between two selected points.

After you click this command, you will be prompted to select a first point. Use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

Once you have selected the first point, you must now do the same for the second point.

Upon selecting your second point, the following dialog is displayed:



This dialog has no options, it only displays the following information:

- **Distance** - this displays the distance (length) of line that was created by selecting the two points.
- **Angle** - this displays the Angle of the line that was created by selecting the two points
- **X** - this displays the distance the second points X coordinate is from the first point's X coordinate
- **Y** - this displays the distance the second points Y coordinate is from the first point's Y coordinate
- **Z** - this displays the distance the second points X coordinate is from the first point's Z coordinate

## Radius of

This command allows you to determine the radius of an Arc or Circle.

After you click this command, you will be prompted to select either an Arc or Circle using your mouse pointer.

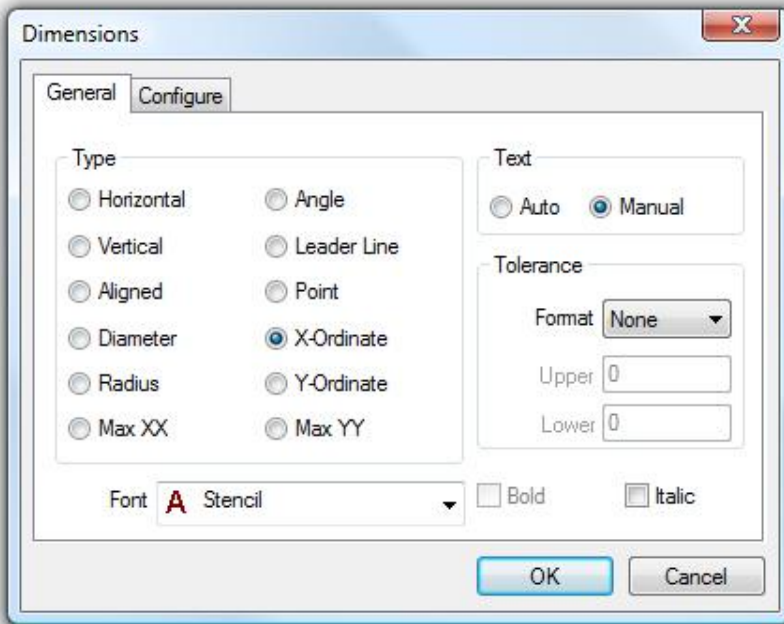
Once you have selected an Arc or Circle, a message will appear in the Numeric Entry area of ALPHACAM stating the Radius of the Arc or Circle.

## Dimension



This command allows you to draw a dimension line.

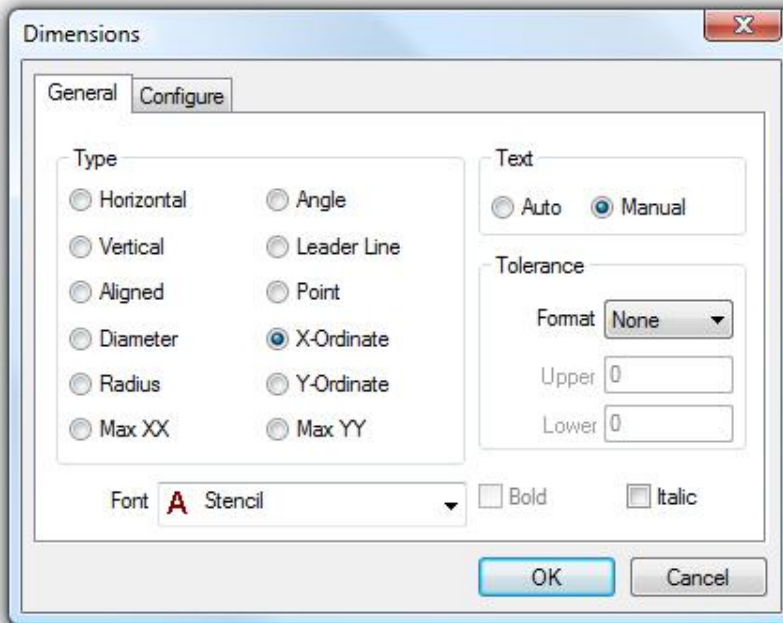
After you click this command, the following dialog is displayed:



This is the dimension dialog, it allows you to select the type of dimension that you can create as well as the setup of the dimension styles.

### Dimension Dialog

This dialog is split up by two tabs, the first one is the General tab:

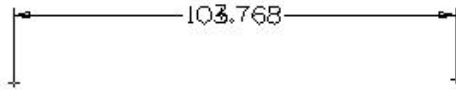


The general tab contains the following options:

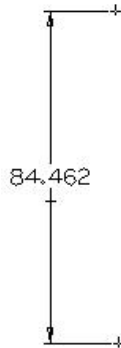
#### Type

The option selected here will determine the type of dimension that is drawn. Each option will create a different type of line

- **Horizontal** - this option allows you to draw a dimension line that will dimension out the distance between two points on the X axis.



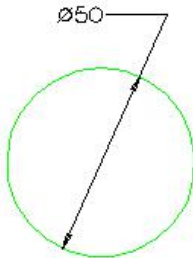
- **Vertical** - this option allows you to draw a dimension line that will dimension out the distance between two points on the Y axis



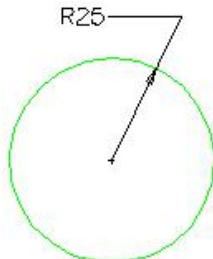
- **Aligned** - this option allows you to draw a dimension line that will dimension out the distance between two points, angling the dimension line to match the shortest distance between two points.



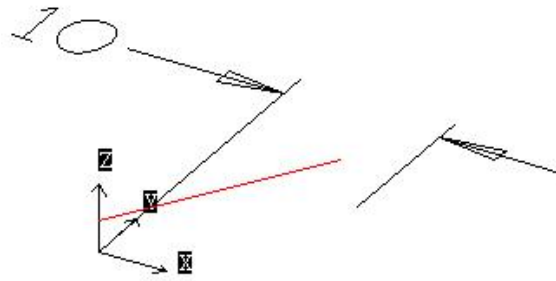
- **Diameter** - This option allows you draw a dimension that depicts the diameter of a Circle or Arc



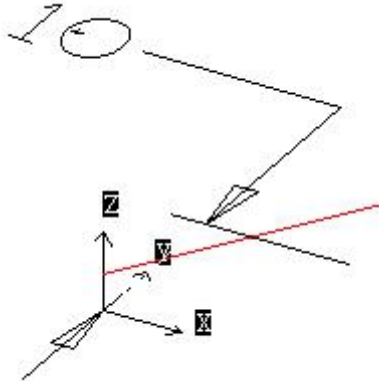
- **Radius** - This option allows you draw a dimension that depicts the radius of a Circle or Arc



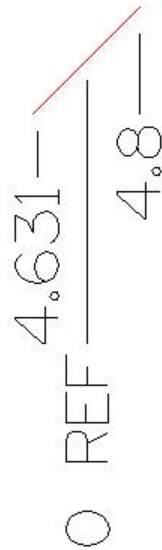
- **Max XX** - this option will dimension the overall dimension parallel to the X Axis (Z in ALPHACAM Lathe) of the geometry or geometries selected.



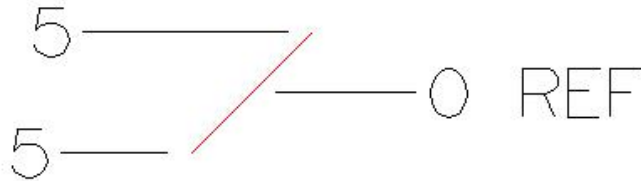
- **Max YY** - this option will dimension the overall dimension parallel to the Y Axis (X in ALPHACAM Lathe) of the geometry or geometries selected.



- **X-Ordinate** - this option allows you to mark the distance along the X axis from a specific 0 Ref.



- **Y-Ordinate** - this option allows you to mark the distance along the Y axis from a specific 0 Ref.



**Text**

This allows you to specify if the text created for the dimension line is automatically set (as the appropriate value) or whether or not you should be prompted to enter the text.

**Tolerance**

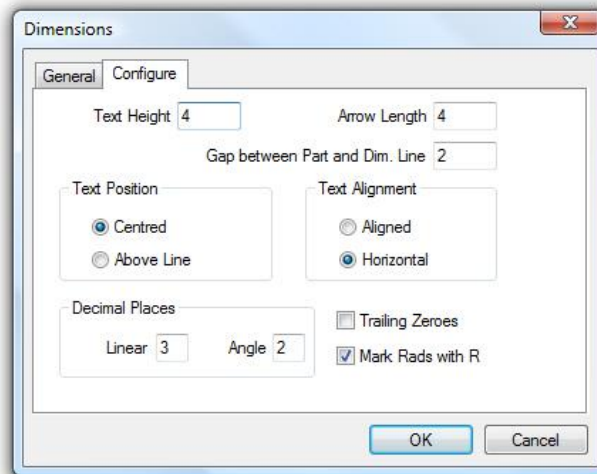
These options allow you to specify a tolerance used to create the dimension lines.

- **Format** - this is a list of the tolerance types, the selected format will determine how the tolerance is defined.
- **Upper** - this is the upper tolerance value that is used
- **Lower** - this is the lower tolerance value that is used

## Additional

- **Font** - this option allows you to specify the font that these dimension lines will be typed in
- **Bold** - this option allows you to specify if the text in the dimension line is to be bold
- **Italic** - this option allows you to specify if the text in the dimension line is to be italic

The next tab is the Configure tab:



This tab allows you to configure how the dimensions will be displayed in the drawing. This tab has the following options:

- **Text Height** - this is the value, in units, that the text that is used on the dimension line will be
- **Arrow Length** - this is the value, in units, that the arrows will be sized to
- **Gap between Part and Dim. Line** - this is the distance, in units, between the dimension line and the part

### Text Position

This option allows you to either set the text to be centered on the dimension line, or to place the text above the dimension line.

### Text Alignment

This option allows you to either set the text to run Horizontal (along the X axis), or to be aligned with the angle of the dimension line.

### Decimal Places

- **Linear** - this determines the number of decimal places displayed on linear dimensions
- **Angle** - this determines the number of decimal places displayed on angular dimensions

### Additional

- **Trailing Zeros** - this flag allows you to determine if ALPHACAM will display trailing zeros (where applicable) in the dimension line's text
- **Mark Rads with R** - this flag allows you to determine if Radians are marked with an 'R' in the dimension text

## Common Line Removal

This command allows you to remove elements that are overlapping.

Selected geometries are broken into individual elements and any elements that overlap will be deleted, leaving a single element.



## Geometry | Special Functions ► Common Line Removal

### Selecting the Geometries

When this command is used, you will be prompted to select the geometries from which common lines will be removed.

To select geometries, click on them.

The command buttons in the Input Bar can also be used to select elements:



Selects the most recently created element.



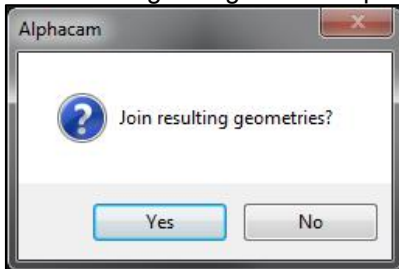
Selects all elements in the current drawing.

If there are overlapping lines, one of them will be removed.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

The following dialog will be displayed:



This dialog contains the following options:

- **Yes** - Joins the resulting elements into a single geometry.
- **No** - Leaves the resulting geometries as separate elements.

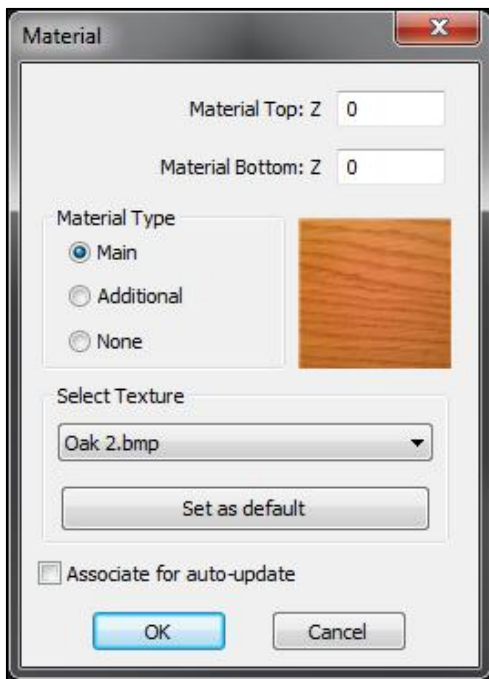
### Set Materials

This command allows you to create material from a closed geometry. A texture can be applied to display a realistic appearance when simulating machining operations. You can specify that material should be automatically updated at the end of a simulation, to be used on subsequent operations.



## 3D | Set Materials

When this command is used, you will be prompted to select a geometry to use as the material using your mouse pointer. After selecting the geometry, the Material dialog is displayed:



This dialog contains the following options:

**Material Top: Z** - This value defines the distance from the work plane of the geometry ( $Z = 0$ ) to the top of the material.

**Material Bottom: Z** - This value defines the distance from the work plane of the geometry ( $Z = 0$ ) to the bottom of the material.

**Material Type** - These options allow you to define the category of the material:

- **Main** - Makes this the main material in the current drawing.
- **Additional** - Makes this material separate from the main material in the current drawing.
- **None** - Removes the material from the drawing.

Only one "Main" material can be set, but you can set as many "Additional" materials as needed to create the material shape that you require.

**Select Texture** - These options enable you to apply textures to the set material, to display a more realistic appearance when simulating.

- To apply a texture, select an option from the drop-down menu. A preview of the selected texture will be shown on the dialog.
- You can add custom texture image files to the C:\ALPHACAM\LICOMDAT\Textures.Alp folder. These will then be added to the drop-down menu.

**Set as Default** - This button makes the selected texture the default option for the Set Materials command.

**Associate for auto-update** - Only used for STL files

To apply the material settings to the selected geometry, click the  button.

## Delete

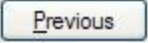
This command allows you to delete elements in the Drawing Area.




Edit | Delete



## Selecting the Element(s)

When this command is used, you can use the mouse pointer to select elements in the Drawing Area for deletion. The command buttons in the Input Bar can also be used to select elements:

 Selects the most recently created element.

 Selects all elements in the current drawing.

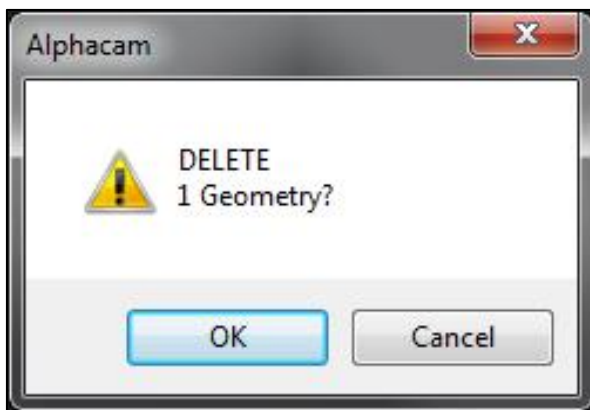
When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

If machining operations have been applied to any of the selected geometries, you will be prompted to select either the geometries, toolpaths or both.

### Deleting the Element(s)

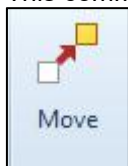
A dialog is displayed prompting you to confirm the selection:



Click **OK** to delete the selected element(s).

### Move

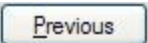
This command allows you to move existing elements in the Drawing Area.



**Edit | Move**

### Selecting the Element(s)

When this command is used, you can use the mouse pointer to select elements to be moved in the Drawing Area. The command buttons in the Input Bar can also be used to select elements:

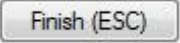

 Selects the most recently created element.





Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

If machining operations have been applied to any of the selected geometries, you will be prompted to select either the geometries, toolpaths or both.

### Selecting the Base Point

You will be prompted to select the base point.

The base point is the datum point relative to the element to be moved.

To select the base point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

### Moving the Element(s)

The Base Point is attached to the mouse pointer.

To move the element(s), use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

### Copy

This command allows you to create single copies of existing elements in the Drawing Area.

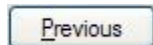


Edit | Move, Copy etc | Copy

### Selecting the Element(s)

When this command is used, you will be prompted to use the mouse pointer to select elements to copy.

The command buttons in the Input Bar can also be used to select elements:





Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

## Selecting the Base Point

You will be prompted to select the base point.

The base point is the datum point relative to the element to be moved.

To select the base point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

The base point is attached to the mouse pointer.

To copy the element(s), use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

## Rotate

This command allows you to create multiple copies of existing elements in the Drawing Area in a circular pattern.

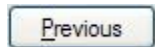


Edit | Rotate

### Selecting the Element(s)

When this command is used, you will be prompted to use the mouse pointer to select elements to rotate.

The command buttons in the Input Bar can also be used to select elements:

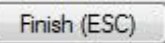



Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

### Rotating the Element(s)

The following steps are different depending on which version of the command is used; 2D Rotate or 3D Rotate.

You will be prompted to select the Base Point.

The base point is the point relative to the element that will be moved.

To select the base point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

You are then prompted to define the Rotation Angle.

The rotation angle can be defined using one of the following methods:

- Move the mouse pointer in the Drawing Area until the desired angle is displayed and click the mouse.
- Enter a value into the Input Bar.

To rotate an element in a clockwise direction, enter a negative value into the Input Bar.

You will then be prompted to enter the number of copies to create.

The value for the number of copies includes the original element. Entering "1" will rotate the original element. If additional copies are required, entering a higher number will retain the original element and create copies that are rotated as specified.

## **Mirror**

This command allows you to create a mirrored copy of an existing element in the Drawing Area.

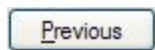


**Edit | Mirror**

### **Selecting the Element(s)**

When this command is used, you can use the mouse pointer to select elements to be mirrored in the Drawing Area.

The command buttons in the Input Bar can also be used to select elements:



Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

If machining operations have been applied to any of the selected geometries, you will be prompted to select either the geometries, toolpaths or both.

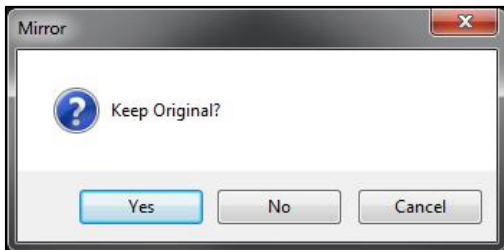
### **Mirroring the Element(s)**

You will be prompted to define the first point of the mirror line using one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to select the second point of the mirror line using one of the same methods.

When the mirror has been defined, the following dialog is displayed:



To keep the original element after mirroring, click **Yes**.

To delete the original element after mirroring, click **No**.

The mirrored copy of the original element will be created on the opposite side of the mirror line.

## **Scale**

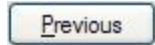
This command allows you to scale existing elements in the Drawing Area. For example, from imperial to metric measurements.



**Edit | Scale**

### **Selecting the Element(s)**

When this command is used, you will be prompted to use the mouse pointer to select elements to be scaled. The command buttons in the Input Bar can also be used to select elements:



Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

Tool diameters cannot be scaled.

If machining operations have been applied to any of the selected geometries, you will be prompted to select either the geometries, toolpaths or both.

### **Scaling the Element(s)**

You will be prompted to select the base point.

The base point is the point relative to the element that will be scaled.

To select the base point, use one of the following methods:

- Click the mouse pointer on the required point in the Drawing Area.
- Use a snap function
- Type the coordinate values into the Input Bar and click **OK**.

You will then be prompted to enter the scale factor. If scaling from mm to inches use .0394 or 1/25.4. Enter the required value into the Input Bar.

The scale factor is the value by which the dimension of the current element is multiplied to create the scaled element. For example, a scale factor of "2" will double the size of the element. A scale factor of "0.5" will halve the size of the element.

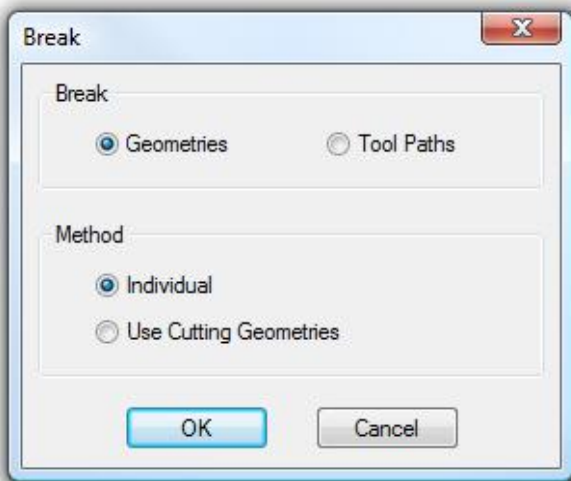
## **Break**

This command allows you to break a single element in the Drawing Area at a specified point, creating two or more separate elements.



**Edit | Break...**

When this command is used, the Break dialog is displayed:



This dialog contains the following options:

**Break** - The options in this section define which type of elements to break:

- **Geometries** - Applies the command only to geometry elements.
- **Tool Paths** - Applies the command only to toolpath elements.

**Method** - The options in this section allows you to select which method will be used to apply the break:

- **Individual** - Individual breakpoints will be inserted into the required positions using the mouse pointer.
- **Use Cutting Geometries** - Geometries will be selected which cut through the element that you want to break.

Click **OK** and follow the prompts in the Input Bar to make selections.

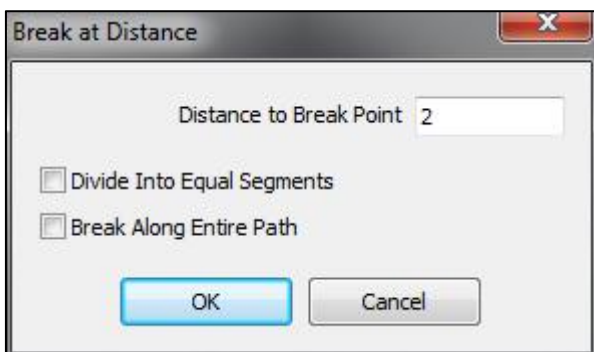
## **Break at Distance**

This command allows you to break an element at a defined distance from the start point.



**Edit | Break at Distance...**

When this command is used, the Break at Distance dialog is displayed:



This dialog contains the following options:

- **Distance to Break Point** - The distance from the start point of the element, at which a break will be applied.
- **Divide Into Equal Segments** - If checked, breaks will be applied to the element, dividing it into equal segments. The total number of segments is the value in the Distance to Break Point field.
- **Break Along Entire Path** - If checked, breaks will be applied along the entire path of the geometry. The

distance between breaks is the value in the Distance to Break Point field.

## **Trim**

This command allows you to remove a section of an element in the Drawing Area using a cutting geometry.

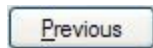


**Edit | Trim**

### **Selecting the Cutting Geometry**

When this command is used, you will be prompted to select the cutting geometry (or the cutting boundary). Multiple cutting geometries can be selected.

The command buttons in the Input Bar can also be used to select elements:





Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to proceed:

- Click .

- Press .

- Right-click in the Drawing Area.

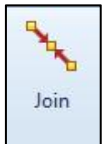
### **Selecting the Element(s) to Trim**

You will be prompted to select the element to trim by clicking it.

The section of the element that is clicked will be trimmed to the point where it coincides with the cutting geometry. If a geometry is selected that has a machining operation applied to it, this command will trim the toolpath first. To trim the geometry, click it again.

## **Join**

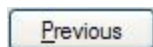
This command allows you to join separate elements in the Drawing Area into a single profile.



**Edit | Join**

### **Selecting the Elements**

When this command is used, you will be prompted to use the mouse pointer to select elements to be joined. The command buttons in the Input Bar can also be used to select elements:



Selects the most recently created element.



Selects all elements in the current drawing.

## Joining the Elements

To join the selected elements, use one of the following methods:

- Click .
- Press .
- Right-click in the Drawing Area.

## Special Rules

The following special rules apply when using the Join command:

- When joining elements that all exist in the same user layer, the joined profile will remain in that layer.
- When joining elements that exist in different user layers, the joined profile will be moved to the Geometry layer.
- If different types of elements are joined, the resulting joined element will inherit the type that has the greatest precedence. The order of precedence of elements is as follows:
  1. Toolpath.
  2. Geometry.
  3. Construction.
  4. User

For example, joining a geometry element to a construction geometry element will result in all joined elements being in the geometry layer.

## Extend

This command allows you to extend an open element to a specified boundary.



Edit | Extend

## Selecting the Boundary Geometry

When this command is used, you will be prompted to select the boundary geometry.

The boundary geometry defines the boundary to which elements will be extended.

Select the boundary geometry using the mouse pointer.

The command buttons in the Input Bar can also be used to select elements:

When the required elements have been selected, use one of the following methods to proceed:

- Click .
- Press .
- Right-click in the Drawing Area.

## Extending the Element(s)

You will be prompted to select the elements to extend.

Click an element to extend it to the boundary geometry.

**Note:** When extending an open geometry, the selected location on the geometry determines the direction in which the element will be extended.

For example, clicking the left side of a horizontal line will extend only the left side. Clicking the right side will extend the right side of the line.

If machining operations have been applied to any of the selected geometries, both geometry and toolpath

elements must be extended separately.

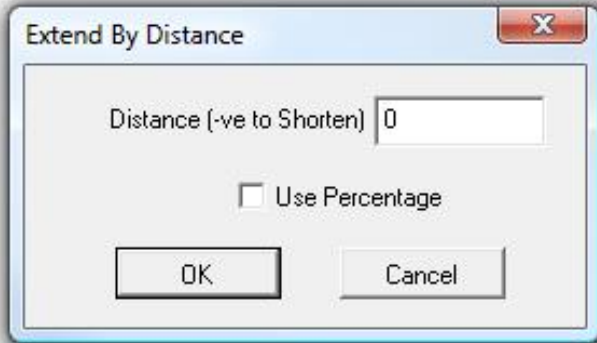
## **Extend by Distance**

This command allows you to extend elements in the Drawing Area by a specified value.



**Edit |Extend by Distance...**

When this command is used, the Extend By Distance dialog is displayed:



Enter the distance to extend in the Distance field.

A positive number will result in the element being extend by that distance

A negative entry will result in the element being shortened by that distance

You will be prompted to select the elements to extend.

Click on the end of the geometry you wish to extend or shorten

When finished click OK

Note: You can also lengthen or shorten the element by a percentage rather than a measured distance by putting a checkmark in the Use percentage box. For example, by putting a check mark in the box and typing 10 in the distance you can lengthen/shorten the geometry by 10 percent.

## **Fillet**

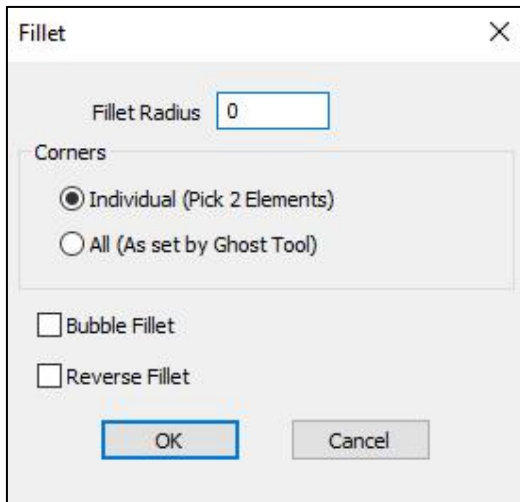
This command allows you to trim or extend two selected geometries or toolpaths and apply a specified blending radius.



**Edit |Fillet**

When this command is used, the Fillet dialog is displayed:





This dialog contains the following options:

**Fillet Radius** - The radius of the fillet.

**Corners** - The options in this section determines how the geometry or tool path will be selected:

- **Individual (Pick 2 Elements)** - Allows two elements to be selected in the Drawing Area, enabling you to apply a single fillet.
- **All (As set by Ghost Tool)** - Selects all elements in the selected geometry or toolpath, enabling you to apply multiple fillets.

**Bubble Fillet** - If checked, a bubble fillet will be applied. For an example of a bubble fillet,

**Reverse Fillet** - If checked, a reverse fillet will be applied. For an example of a reverse fillet,

### Applying the Fillet

When you click **OK** on the dialog, follow the prompts in the Input Bar to select elements to fillet in the Drawing Area.

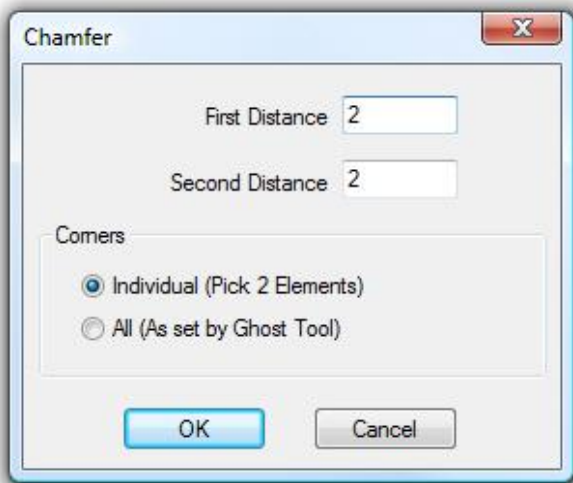
## Chamfer

This command allows you to trim or extend two selected geometries or toolpaths and apply a chamfer.



**Edit | Chamfer...**

When this command is used, the Chamfer dialog is displayed:



This dialog contains the following options:

**First Distance** - The dimension of the chamfer along the axis of the first selected element.

**Second Distance** - The dimension of the chamfer along the axis of the second selected element.

**Corners** - The options in this section determine how the geometry or toolpaths will be selected:

- **Individual (Pick 2 Elements)** - Allows two elements to be selected in the Drawing Area, enabling you to apply a single chamfer.
- **All (As set by Ghost Tool)** - Selects all elements in the geometry or toolpaths that you select, enabling you to apply multiple chamfers.

## Applying the Chamfer

Click **OK** on the dialog and follow the prompts in the Input Bar to select elements in the Drawing Area to chamfer.

The order in which the geometries in the Drawing Area are selected is important.

The value in the First Distance field is applied to the first element that is selected; the value in the Second Distance field is applied to the second element.

If the selected geometries do not intersect, they will be extended before the chamfer is applied.

The command will fail if the specified chamfer distances are too large to locate on the selected geometry.

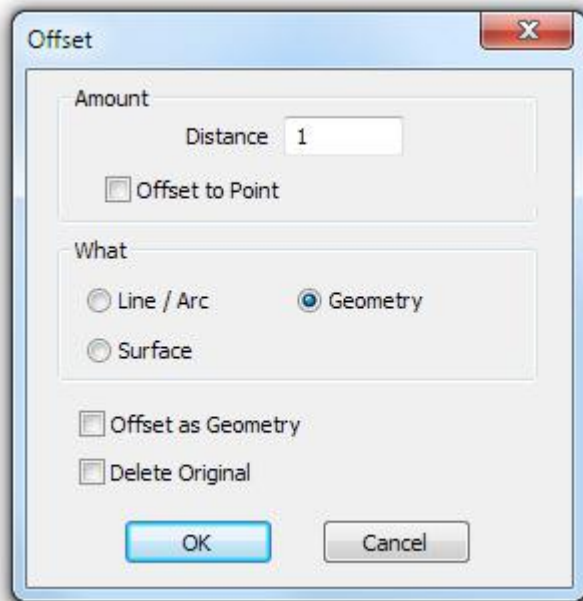
## Offset

This command allows you to create a copy of a geometry or surface that is offset from the original by a set distance or coincident with a specific point.



Edit | Break, Join etc | Offset...

When this command is used, the Offset dialog is displayed:



This dialog contains the following options:

**Amount** - The options in this section allow you to specify the amount that the copy will be offset from the original element:

- **Distance** - The distance by which the new element will be offset from the original element.
- **Offset to Point** - If checked, a point can be selected with which the offset copy will coincide.

**What** - The options in this section allow you to specify the type of element to offset:

- **Line / Arc** - Enables you to select a single element in a geometry.
- **Geometry** - Enables you to select a geometry.
- **Surface** - Enables you to select a surface.

**Offset as Geometry** - If checked, the offset copy is created in the alphacam geometry layer.

If not checked the offset copy is created in the alphacam construction layer..

**Delete Original** - If checked, the original will be deleted when the offset copy is created.

This option is only available if the option in the "What" section is set to either Geometry or Surface.

## Offsetting the Element(s)

Click **OK** on the dialog and follow the prompts in the Input Bar to select elements in the Drawing Area to offset.

When prompted, click anywhere in the Drawing Area that is on the side of the original element where the copy is required.

## Explode

This command allows you to break a geometry into multiple elements.



**Edit | Explode**

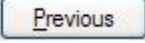
This command can only be applied to geometries


This command cannot be applied to toolpaths.

## Selecting the Geometries

When this command is used, you can use the mouse pointer to select elements to explode in the Drawing Area.

The command buttons in the Input Bar can also be used to select elements:

 Selects the most recently created element.

 Selects all elements in the current drawing.

## Exploding the Geometries

When the required geometries have been selected, use one of the following methods to explode the geometries:

- Click .
- Press .
- Right-click in the Drawing Area.

## Change

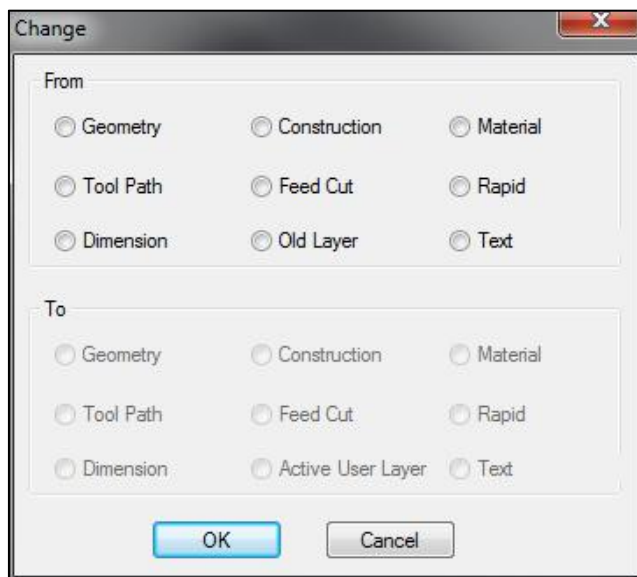
This command allows you to change an entity from one type to another.

 + 



**Edit | Change...**

When this command is used, the Change dialog is displayed:



This dialog is divided into two sections:

- **From** - The option selected in this section is the type of element from which you want to change.
- **To** - The option selected in this section is the type of element to which you want to change.
- 

## Changing an Element

Click an option in the "From" section.

Click an option that becomes available in the "To" section.

The options that are available in the "To" section will vary because the type of element that you can change to is dependent on the type of element that you are changing from.

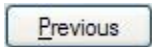
Click **OK** when an option has been selected in both sections.

## Selecting the Element(s) to Change

You will be prompted to 'Select Contour'.

It is only possible to select elements of the type that was selected in the "From" section of the Change dialog if you choose to select them individually. Use the mouse pointer to select all of the elements you want to change.

The command buttons in the Input Bar can also be used to select elements:



Selects the most recently created element.



Selects all elements in the current drawing.

When the required elements have been selected, use one of the following methods to apply the command:

- Click .
- Press .
- Right-click in the Drawing Area.

## Ghost Tools

This command will display an arrow that represents the direction that tool paths will take along a geometry.2D

Arrow: 