

## Lettering>About (Plug-In TSTEXT1)

This plug-in can only be used if you have purchased a personal unlock code. You can order the unlock code for this plug-in directly from TommySoftware®. For further information see [Order & Unlock](#).

### Menu Commands

Lettering

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## Lettering>Surface Finish ISO 1302 (Plug-In TSTEXT1)

### General

This command is used to create a surface finish symbol based on ISO 1302. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

#### 1. *Enter symbol position*

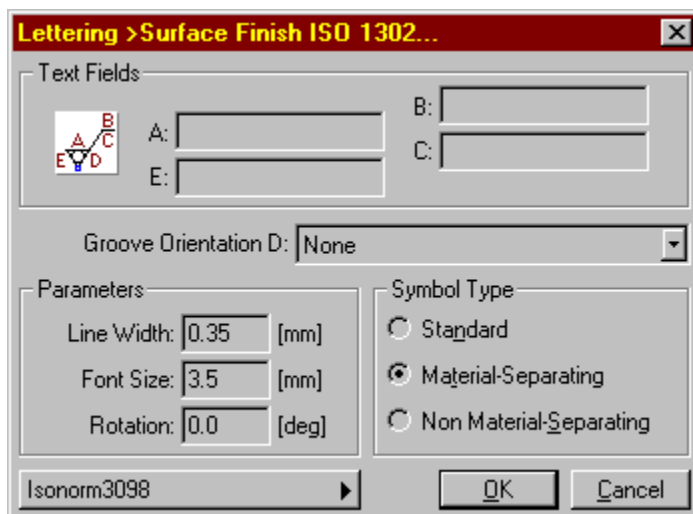
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

How can I access information on this dialog?



## Lettering>Shape / Position Tolerance ISO 1101 (Plug-In TSTEXT1) General

This command is used to create a shape / position tolerance symbol based on ISO 1101. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

### 1. *Enter symbol position*

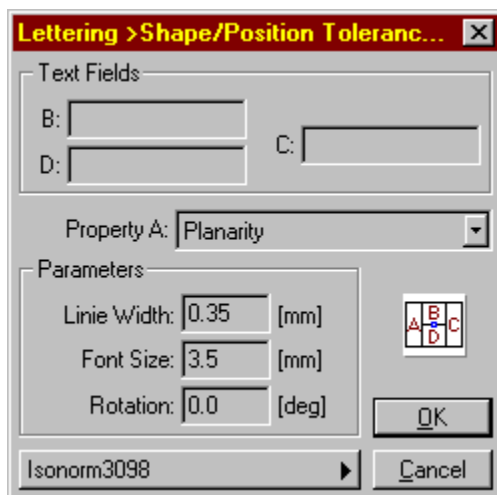
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

## Options

How can I access information on this dialog?



## Lettering>Shape / Position Tolerance ISO 1101 with Line (Plug-In TSTEXT1)

### General

This command is used to create a shape / position tolerance symbol based on ISO 1101 with a reference line. After entering the desired parameters in the options dialog (see below), first a reference object is chosen, then the resulting symbol is placed.

#### 1. *Identify reference object*

Any part of an existing object can be used as a reference object. This can be a rectangle side, the arc of an ellipse segment or a circular arc within a surface.

Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

#### 2. *Enter symbol position*

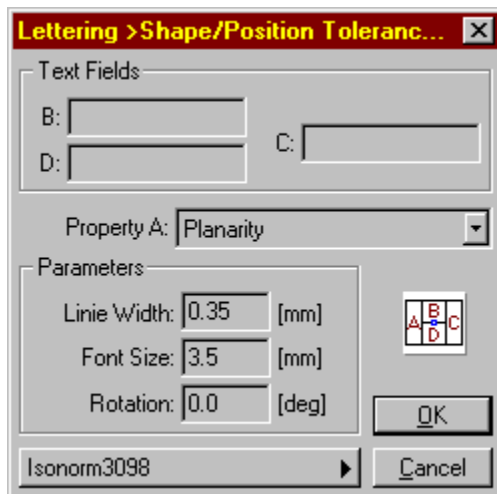
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

How can I access information on this dialog?



## Lettering>Tolerance Datum ISO 1101 (Plug-In TSTEXT1)

### General

This command is used to create an arbitrary oriented tolerance datum symbol based on ISO 1101. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

1. *Enter reference point*

The position of the tolerance datum is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

2. *Enter symbol position*

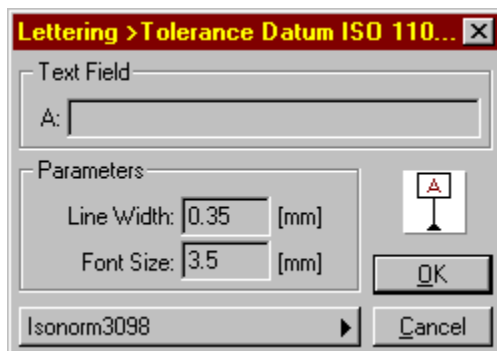
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

[How can I access information on this dialog?](#)



## Lettering>Tolerance Datum ISO 1101, Perpendicular (Plug-In TSTEXT1)

### General

This command is used to create a tolerance datum symbol based on ISO 1101 that is perpendicular to a reference object. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

#### 1. *Identify reference object*

Any part of an existing object can be used as a reference object. This can be a rectangle side, the arc of an ellipse segment or a circular arc within a surface.

Click on the relevant object (part object) with the mouse to identify it. If several objects lie very close to the point clicked on, then a small dialog appears at the bottom of the screen, with the help of which the correct object can be chosen.

#### 2. *Enter symbol position*

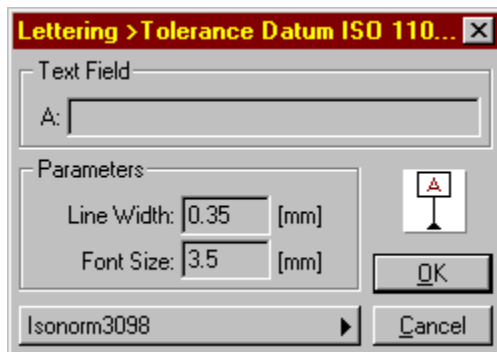
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

How can I access information on this dialog?



## Lettering>Workpiece Edge DIN 6784 (Plug-In TSTEXT1)

### General

This command is used to create a workpiece edge symbol based on DIN 6784. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

#### 1. *Enter symbol position*

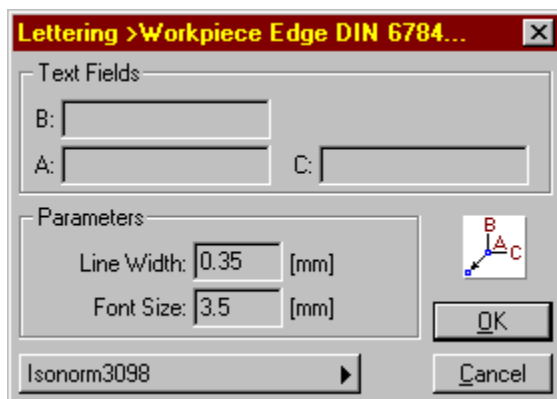
The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

How can I access information on this dialog?



## Lettering > Center Hole DIN 332 T10 (Plug-In TSTEXT1)

### General

This command is used to create a center hole symbol based on DIN 332 T10. After entering the desired parameters in the options dialog (see below), the resulting symbol is placed.

#### 1. *Enter symbol position*

The point at which the symbol shall be inserted is chosen by clicking the mouse at any chosen point within the drawing.

Optionally, precise coordinates can be specified. To do this, press the F8 or the ENTER key. You can specify absolute or polar coordinates in the dialog which appears. For information about entering coordinates, refer to [Coordinate Entry \(F8\)](#).

After specifying the insertion point, the symbol is created and inserted to the drawing.

### Options

How can I access information on this dialog?





Clicking on this button will close the dialog accepting all changes. Any changes or operations specified will be carried out.

Clicking on this button will close the dialog, without accepting any changes. Any following operation will not be carried out.

The text in this edit field will be displayed in the created symbol at the position A (see icon).

The text in this edit field will be displayed in the created symbol at the position B (see icon).

The text in this edit field will be displayed in the created symbol at the position C (see icon).

The text in this edit field will be displayed in the created symbol at the position D (see icon).

The text in this edit field will be displayed in the created symbol at the position E (see icon).

This list contains all possible groove orientation symbols available. The symbol selected here will be displayed in the created symbol at the position D (see icon).



The value in this edit field determines the line width of the symbol created.

The value in this edit field determines the font size of all texts in the symbol created.

The value in this edit field determines the rotation angle of the symbol created. This includes both the symbol and all texts.

This button displays the current font's name. Clicking on this button will display a dialog that allows to select a font which will be used for all texts in the symbol created.

If this radio box is set, the symbol will have the "Standard" form.

If this radio box is set, the symbol will have the "Material-Separating" form.

If this radio box is set, the symbol will have the "Non Material-Separating" form.

This list contains all possible property symbols available. The symbol selected here will be displayed in the created symbol at the position A (see icon).



If this radio box is set, the symbol will have the "Required" form.

If this radio box is set, the symbol will have the "Must Not Remain" form.



