# Schematic Text String

**Contents**

- Description
- Availability
- Placement
- Editing
  - Editing via an associated properties dialog
  - Special Strings
  - Special Strings for use with Component Parameters
  - Editing via the SCH Inspector panel
  - Editing via the SCH List panel
  - Graphical editing
- Notes

Altium

# Text

## Description

A text string (also referred to as an annotation) is a non-electrical drawing primitive. It is a single line of free text that can be placed on a schematic sheet. Uses might include section headings, revision history, timing information or some other descriptive or instructive text.

## Availability

Text strings are available for placement in both Schematic and Schematic Library Editors by:

- Choosing **Place » Text String** [P, T] from the main menus
- Clicking the A button on the Utility Tools drop-down of the Utilities toolbar.

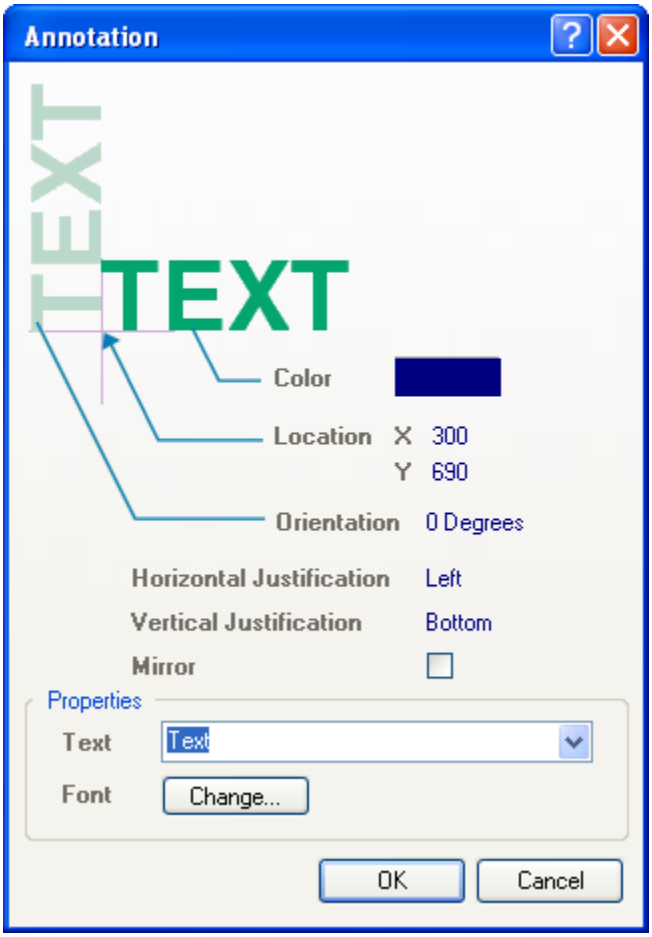
## Placement

After launching the command, the cursor will change to a cross-hair and you will enter text string placement mode.
Position the cursor on the sheet and click or press Enter to effect placement of the string.
Continue placing further text strings or right-click or press Esc to exit placement mode.
Press the Spacebar while in placement mode to rotate the text string. Rotation is anti-clockwise and in steps of 90°.
Press the X or Y keys while in placement mode to flip the text string along the X-axis or Y-axis respectively.

## Editing

The properties of a text string object can be modified before, during and after placement. Editing itself falls into two categories - graphical and non-graphical.
The following three methods of non-graphical editing are available:

### Editing via an associated properties dialog

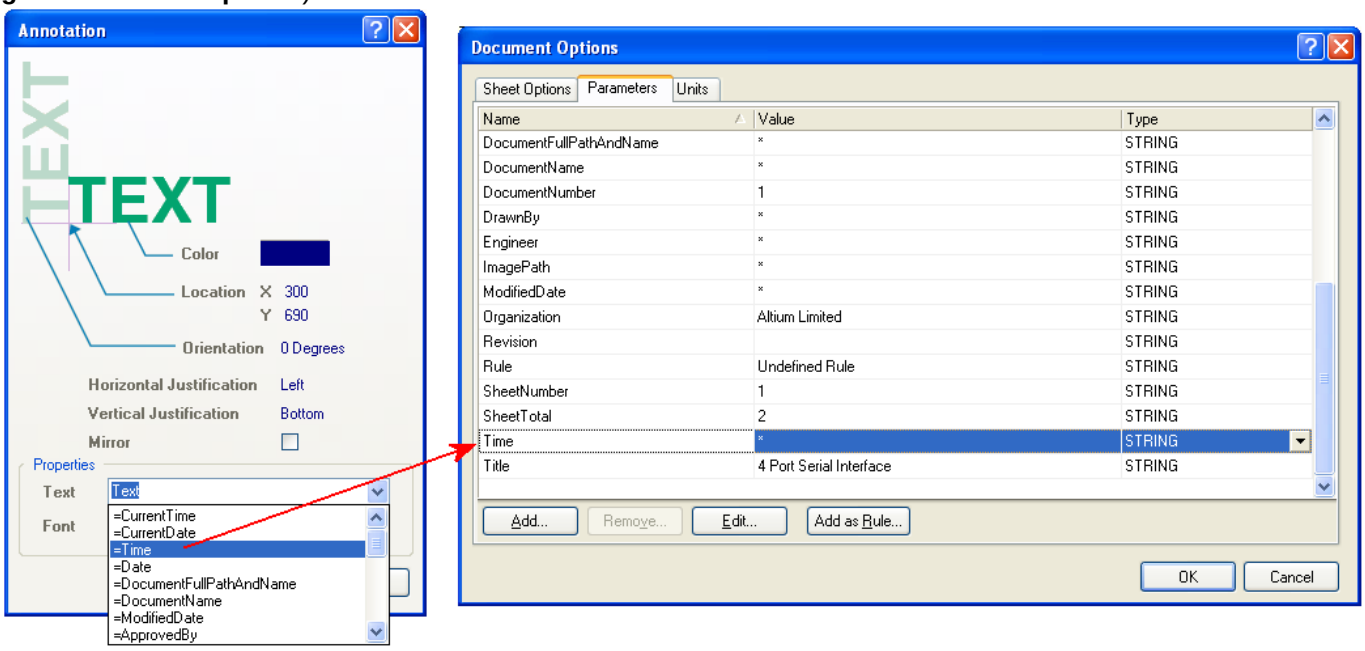



This method of editing uses the following dialog to modify the properties of a text string object.
Use the dialog's 'What's This Help' feature to obtain detailed information about each of the options available. Click on the question mark button at the top right of the dialog and then click over a field or option to pop-up information specific to that field or option.
The Annotation dialog can be accessed prior to entering placement mode, from the Schematic - Default Primitives page of the Preferences dialog (Tools » Schematic Preferences). This allows you to change the default properties for the text string object, which will be applied when placing subsequent text strings.
During placement, the Annotation dialog can be accessed by pressing the Tab key.
After placement, the Annotation dialog can be accessed in one of the following ways:

- double-clicking on the placed text string object
- selecting the text string object and choosing Properties from the right-click pop-up menu (Schematic Editor only)
- choosing the Change command from the Edit menu and then clicking once over the placed text string object.

The actual content of the text string can be entered/edited via the Annotation dialog using the Text field. Type the required text directly into the field. The Text field also provides a drop-down list of special strings that link to

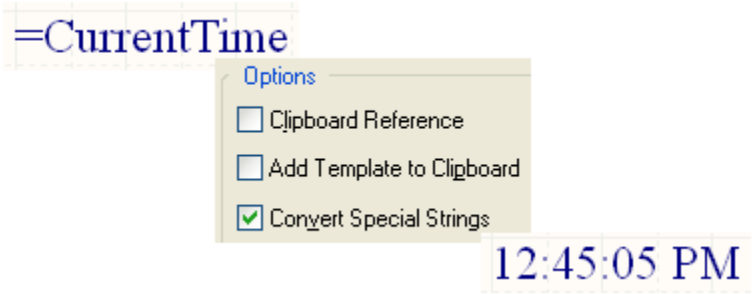
parameter information defined for the active document on the Parameters tab of the Document Options dialog (**Design » Document Options**).




## Special Strings

Special strings are text strings which are recognized and interpreted when the sheet is printed or plotted.

Certain special strings provide current information, such as date and time, which are inserted only at the time of printing. Some special stings can be displayed on the document prior to printing, by enabling the **Convert Special Strings** option on the **Schematic – Graphical Editing** page of the *Preferences* dialog (**Tools»Schematic Preferences**).




Special Strings are preceded by the '=' character. For a new schematic document, the default list of supported special strings for use on a schematic can be found by placing a Text String object (**Place»Text String**), then double-clicking the string to access the *Annotation* dialog. The list can be seen by clicking the drop-down arrow associated to the **Text** field. The majority of special string entries link to default parameter information defined for the active document on the **Parameters** tab of the *Document Options* dialog (**Design»Document Options**).

> **Note:** On a schematic sheet, special strings are characterized by the prefix '=' (`=CurrentTime`, `=CurrentDate`, etc). On a PCB document, special strings are characterized by the prefix '.' (`.Legend`, `.Net_Count`, etc).

The following lists the default set of special strings, as viewed in the *Annotation* dialog for a placed string (free text):

- **=Address1** – displays the value specified for the default parameter `Address1`, on the Parameters tab of the *Document Options* dialog.
- **=Address2** – displays the value specified for the default parameter `Address2`, on the Parameters tab of the *Document Options* dialog.
- **=Address3** – displays the value specified for the default parameter `Address3`, on the Parameters tab of the *Document Options* dialog.
- **=Address4** – displays the value specified for the default parameter `Address4`, on the Parameters tab of the *Document Options* dialog.
- **=ApprovedBy** – displays the value specified for the default parameter `ApprovedBy`, on the Parameters tab of the *Document Options* dialog.
- **=Author** – displays the value specified for the default parameter `ApprovedBy`, on the Parameters tab of the *Document Options* dialog.
- **=CheckedBy** – displays the value specified for the default parameter `CheckedBy`, on the Parameters tab of the *Document Options* dialog.
- **=CompanyName** – displays the value specified for the default parameter `CompanyName`, on the Parameters tab of the *Document Options* dialog.
- **=CurrentDate** – the current date, automatically calculated from your system settings and in the format `dd/mm/yyyy`, updated upon editing your schematic or on refresh/redraw. Example: `13/01/2010`.
- **=CurrentTime** – the current time, automatically calculated from your system settings and in the format `h:mm:ss AM/PM`, updated upon editing your schematic or on refresh/redraw. Example: `2:39:47 PM`.
- **=Date** – used to display static date information. Displays the value specified for the default parameter `Date`, on the Parameters tab of the *Document Options* dialog. Unlike the `=CurrentDate` special string, which is automatically calculated and presented in a set format, you can enter static date information in any format you like.
- **=DocumentFullPathAndName** – used to display the full path and name of the document into which the string is placed. Example: `C:\Program Files\Altium Designer Summer 09\Examples\Reference Designs\Multi-Channel Mixer\Mixer.SchDoc`.
- **=DocumentName** – used to display the schematic's file name only (without the file path). Example: `Mixer.SchDoc`.
- **=DocumentNumber** – displays the value specified for the default parameter `DocumentNumber`, on the Parameters tab of the *Document Options* dialog. The source parameter can also be updated through the *Sheet Numbering For Project* dialog, when using the Tools»Number Schematic Sheets command.
- **=DrawnBy** – displays the value specified for the default parameter `DrawnBy`, on the Parameters tab of the *Document Options* dialog.
- **=Engineer** – displays the value specified for the default parameter `Engineer`, on the Parameters tab of the *Document Options* dialog.
- **=ImagePath** – displays the value specified for the default parameter `ImagePath`, on the Parameters tab of the *Document Options* dialog.
- **=ModifiedDate** – the modified date stamp of the schematic, automatically populated. Example: `19/01/2010`.

- **=Organization** – displays the value specified for the default parameter `Organization`, on the Parameters tab of the *Document Options* dialog.
- **=ProjectName** – displays the actual name of the project (including extension). For example, for a project with filename `MyPCB.PrjPcb`, this special string will display `MyPCB.PrjPcb`.
- **=Revision** – displays the value specified for the default parameter `Revision`, on the Parameters tab of the *Document Options* dialog.
- **=Rule** – displays the value specified for the default parameter `Rule`, on the Parameters tab of the *Document Options* dialog. The value for this parameter will initially be `Undefined` (appearing as * on the schematic). Double-click on the parameter value to access the ability to define a rule type and edit its constraint(s).
- **=SheetNumber** – the sheet number of the current schematic. This value is calculated when using the Number Schematic Sheets or Annotate Compiled Sheets commands from the Tools menu:
    - *Number Schematic Sheets* – the assigned sheet number will be entered into the value for the default parameter `SheetNumber`, on the Parameters tab of the *Document Options* dialog. The special string, when used on the Editor tab for the schematic sheet will source its information from here.
    - *Annotate Compiled Sheets* – the special string will display the assigned Compiled Sheet Number – on the compiled tab view of the schematic sheet – provided the option to expand Sheet Number is enabled, in the Compiled Names Expansion section, on the Schematic – Compiler page of the *Prefere nces* dialog (DXP»Preferences).
- **=SheetTotal** – the sheet total for the project. This value is calculated when using the Number Schematic Sheets or Annotate Compiled Sheets commands from the Tools menu:
    - Number Schematic Sheets_ – the assigned sheet number will be entered into the value for the default parameter `SheetTotal`, on the Parameters tab of the *Document Options* dialog. The special string, when used on the Editor tab for the schematic sheet will source its information from here.
    - *Annotate Compiled Sheets* – the special string will display the Compiled Sheet Number total – on the compiled tab view of the schematic sheet – provided the option to expand Sheet Number is enabled, in the Compiled Names Expansion section, on the Schematic – Compiler page of the *Preferences* dialog (DXP»Preferences).
- **=Time** – used to display static time information. Displays the value specified for the default parameter `Time`, on the Parameters tab of the *Document Options* dialog. Unlike the `=CurrentTime` special string, which is automatically calculated and presented in a set format, you can enter static time information in any format you like.
- **=Title** – displays the value specified for the default parameter `Title`, on the Parameters tab of the *Document Options* dialog.
- **=VariantName** – this is the name of an assembly variant, as specified in the *Assembly Variant Management* dialog (Project»Assembly Variants). The value displayed is dependent on the assembly variant selected. On the Editor tab view of the schematic sheet, this special string is not interpreted until output is generated. On the Compiled tab view of the schematic, this string will display the assembly variant name, provided you have selected to view that assembly variant from the Current Project Variant drop-down field of the Assembly Variants toolbar.
- **=[VariantParameterName]** – enter the actual name of a variant parameter as the special string name – so for a variant parameter named `Model`, simply enter `=Model`. The value of the variant parameter displayed is dependent on the assembly variant selected. You could, for example, specify the same variant parameter

name for all of your assembly variants and enter a different value for each. As with `=VariantName`, this special string is not interpreted until output is generated, when viewed on the Editor tab view of the schematic sheet. On the Compiled tab view of the schematic, this string will display the value for the parameter, provided you have selected to view an assembly variant with that parameter, from the Current Project Variant drop-down field of the Assembly Variants toolbar.

You can also add your own custom special strings by defining additional parameters on the **Parameters** tab of either the *Document Options* or *Project Options* dialogs. Document parameters are local to the current document in which they are defined, whereas project parameters are available for use across all schematic sheets in the project. Once a parameter has been added to either of these locations, it will appear as an additional special string entry in the *Annotation* dialog, when placing a string object.

### Special Strings for use with Component Parameters

Several additional special strings (or special interpretations of existing ones) are available when defining component parameters:

- **=CurrentFootprint** – displays the name of the currently assigned footprint for the component, as defined in the Models region of the associated *Component Properties* dialog.
- **=Comment** – displays the value appearing in the component's Comment field, as defined in the Properties region of the associated *Component Properties* dialog.
- **=Description** – displays the value appearing in the component's Description field, as defined in the Properties region of the associated *Component Properties* dialog.
- **=[ParameterName]** – displays the value defined for a specified component parameter. Enter the actual name of a component parameter as the special string name – so for a component parameter named `Manufacturer`, simply enter `=Manufacturer`. When defining the Comment property for a component, the associated drop-down field will be populated with special strings for all existing component parameters – enabling you to quickly use any defined parameter's value for the Comment.

### Editing via the SCH Inspector panel

The SCH Inspector panel enables you to interrogate and edit the properties of one or more design objects in the active document. Used in conjunction with appropriate filtering, the panel can be used to make changes to multiple objects of the same kind, from one convenient location.

# For more information on the SCH Inspector panel, press F1 when the cursor is over the panel.

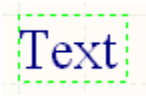
### Editing via the SCH List panel

The SCH List panel allows you to display design objects from one or more documents in tabular format, enabling you to quickly inspect and modify object attributes. When used in conjunction with the SCH Filter panel, it enables you to display just those objects falling under the scope of the active filter - allowing you to target and edit multiple design objects with greater accuracy and efficiency.

# For more information on the SCH List and SCH Inspector panels, press F1 when the cursor is over a panel.

### Graphical editing

This method of editing allows you to select a placed text string object directly in the workspace and change its location graphically. Text strings can only be adjusted with respect to their size by changing the size of the font used (accessed through the Annotation dialog). As such, editing handles are not available when the text string object is selected:




Click anywhere inside the dashed box and drag to reposition the text string as required. The text string can be rotated or flipped while dragging.
If the Enable In-Place Editing option is enabled on the Schematic - General page of the Preferences dialog (**Tools » Schematic Preferences**), you will be able to edit the content for the text string directly in the workspace. Select the text string and then click once to invoke the feature. Type the new text as required and then click away from the text string or press Enter to effect the change.
If you attempt to graphically modify a text string object that has its Locked property enabled, a dialog will appear asking for confirmation to proceed with the edit.
If the Protect Locked Objects option is enabled in the Schematic - Graphical Editing page of the Preferences dialog ( **Tools » Schematic Preferences**), and the Locked option for this design object is enabled as well then this object cannot be selected or graphically edited.
You will have to double click on this locked object directly and disable the Locked property or disable the Protect Locked Objects option in the Schematic - Graphical Editing Preferences dialog to graphically edit this object.
If you attempt to select locked objects along with other objects, only those objects that are unlocked can be selected and moved as a group if the Protect Locked Objects option is enabled. Otherwise if the Protect Locked Objects option is disabled you will be prompted for confirmation to proceed with the edit of the objects including locked objects.

## Notes

A newly placed text string will initially have a default name of Text. Edit the properties of the text string to change the text to that required.

Any changes made to object properties during placement will cause the default properties for the object to be updated, unless the Permanent option - on the Schematic - Default Primitives page of the Preferences dialog is enabled. When this option is enabled, changes made will affect only the object being placed and subsequent objects placed during the same placement session.