Using Altium Documentation
Modified by by Admin on Apr 11, 2017

Parent page: Schematic Commands

The following pre-packaged resources, derived from this base command, are available:

Add Port References to Current Sheet

Applied Parameters: `Action=AddToDocument`

Summary

This command is used to add port cross-referencing information to the current schematic document.

Access

This command is accessed from the Schematic Editor by choosing the **Reports » Port Cross Reference » Add To Sheet** command, from the main menus.

> The project must be compiled before this command becomes available to use.

Use

After launching the command a string will be included, where applicable, next to each port, detailing the sheet name and grid reference of the entity to which the port connects - other ports of the same name (when the **Net Identifier Scope** is `Flat` or `Global`), or a sheet entry in a parent sheet symbol (when the **Net Identifier Scope** is `Hierarchical`, or `Strict Hierarchical`).

Tips

1. Cross reference information is added as a parameter to a port. The name of the parameter is simply `CrossRef`. As a standard parameter, you can modify its display properties as required - for more information, refer to the editing section of the Parameter object page.

Add Port References to Current Sheet (Context Sensitive)

Applied Parameters: `ContextSensitive=True|Action=AddToDocument`

Summary

This command is used to add port cross-referencing information to the current schematic document.

Access

This command is accessed from the Schematic Editor by right-clicking in the workspace - away from any design objects - and choosing the **Sheet Actions » Port Cross Reference » Add To Sheet** command, from the main menus.

> The project must be compiled before this command becomes available to use.

Use

After launching the command a string will be included, where applicable, next to each port, detailing the sheet name and grid reference of the entity to which the port connects - other ports of the same name (when the **Net Identifier Scope** is `Flat` or `Global`), or a sheet entry in a parent sheet symbol (when the **Net Identifier Scope** is `Hierarchical`, or `Strict Hierarchical`).

Tips

1. Cross reference information is added as a parameter to a port. The name of the parameter is simply `CrossRef`. As a standard parameter, you can modify its display properties as required - for more information, refer to the editing section of the [Parameter object](#) page.

[Add Port References to Project](#)

Applied Parameters: `Action=AddToProject`

Summary

This command is used to add port cross-referencing information to all source schematic documents in the active project.

Access

This command is accessed from the Schematic Editor by choosing the **Reports » Port Cross Reference » Add To Project** command, from the main menus.

> The project must be compiled before this command becomes available to use.

Use

After launching the command a string will be included, where applicable, next to each port, across all sheets in the active project, detailing the sheet name and grid reference of the entity to which the port connects - other ports of the same name (when the **Net Identifier Scope** is `Flat` or `Global`), or a sheet entry in a parent sheet symbol (when the **Net Identifier Scope** is `Hierarchical`, or `Strict Hierarchical`).

Tips

1. Cross reference information is added as a parameter to a port. The name of the parameter is simply `CrossRef`. As a standard parameter, you can modify its display properties as required - for more information, refer to the editing section of the [Parameter object](#) page.

[Add Port References to Project (Context Sensitive)](#)

Applied Parameters: `ContextSensitive=True|Action=AddToProject`

Summary

This command is used to add port cross-referencing information to all source schematic documents in the active project.

Access

This command is accessed from the Schematic Editor by right-clicking in the workspace - away from any design objects - and choosing the **Sheet Actions » Port Cross Reference » Add To Project** command, from the main menus.

The project must be compiled before this command becomes available to use.

Use

After launching the command a string will be included, where applicable, next to each port, across all sheets in the active project, detailing the sheet name and grid reference of the entity to which the port connects - other ports of the same name (when the **Net Identifier Scope** is `Flat` or `Global`), or a sheet entry in a parent sheet symbol (when the **Net Identifier Scope** is `Hierarchical`, or `Strict Hierarchical`).

Tips

1. Cross reference information is added as a parameter to a port. The name of the parameter is simply `CrossRef`. As a standard parameter, you can modify its display properties as required - for more information, refer to the editing section of the [Parameter object](#) page.

[Remove Port References from Current Sheet](#)

Applied Parameters: `Action=RemoveFromDocument`

Summary

This command is used to remove port cross-referencing information from the current schematic document.

Access

This command is accessed from the Schematic Editor by choosing the **Reports » Port Cross**

Reference » Remove From Sheet command, from the main menus.

Use

After launching the command, the cross reference parameter (`CrossRef`) will be removed from each applicable port on the current sheet.

[Remove Port References from Current Sheet (Context Sensitive)](#)

Applied Parameters: `ContextSensitive=True|Action=RemoveFromDocument`

Summary

This command is used to remove port cross-referencing information from the current schematic document.

Access

This command is accessed from the Schematic Editor by right-clicking in the workspace - away from any design objects - and choosing the **Sheet Actions » Port Cross Reference » Remove From Sheet** command, from the main menus.

Use

After launching the command, the cross reference parameter (`CrossRef`) will be removed from each applicable port on the current sheet.

[Remove Port References from Project](#)

Applied Parameters: `Action=RemoveFromProject`

Summary

This command is used to remove port cross-referencing information from all source schematic documents in the active project.

Access

This command is accessed from the Schematic Editor by choosing the **Reports » Port Cross Reference » Remove From Project** command, from the main menus.

Use

After launching the command, the cross reference parameter (`CrossRef`) will be removed from each applicable port, across all sheets in the active project.

[Remove Port References from Project (Context Sensitive)](#)

Applied Parameters: `ContextSensitive=True|Action=RemoveFromProject`

Summary

This command is used to remove port cross-referencing information from all source schematic documents in the active project.

Access

This command is accessed from the Schematic Editor by right-clicking in the workspace - away from any design objects - and choosing the **Sheet Actions » Port Cross Reference » Remove From Project** command, from the main menus.

Use

After launching the command, the cross reference parameter (`CrossRef`) will be removed from each applicable port, across all sheets in the active project.