Server Process Reference

Summary

Technical Reference
TR0124 (v1.5) August 4, 2008

This reference manual describes the server processes available in Altium Designer.

This reference details:

- *Server Processes*
- *Client (System) Processes*
- *Integrated Library Processes*
- *PCB Processes*
- *FPGAFlow Processes*
- *Schematic Processes*
- *WorkSpace Manager Processes*.

Server Processes Reference

Servers

A server provides its services in the Altium Designer environment (the client side). The Client module of the Altium Designer interprets the tasks in terms of processes and then delegates these processes to the appropriate servers.

For example when a user is clicking on the Schematic menu to place a wire, the Client interprets this action as a **'PlaceWire'** process and delegates the process to the Schematic Editor server. The Schematic server responds by executing the process. The functionality of a server that is installed in the Altium Designer is exposed by that server's processes and its exposed functions.

Generally a process is executed by selecting a packaged process launcher (such as clicking on a toolbar button, or pressing a hot key or selecting a menu item) called as a command in Altium Designer, however you may wish to manually run a process: Up to three different types of process launchers can be used to launch the same process.

Server Processes

Each server process has a process identifier. The process identifier is made up of two parts separated by a colon. The first part of the process identifier indicates the server that defines the process, and the second part is the process name.

For example, the process `Sch:ZoomIn` is provided by the Schematic Editor server. When this process is launched, either by selecting a menu item, pressing a hot key or activating a toolbar button (which are all defined as process launchers in the Altium Designer), it will perform the task of zooming in on the currently active schematic sheet.

A process is implemented as a `server name:server process` string. Processes are stored in a command launcher table maintained by the server. Every time you execute a process via the user interface in Altium Designer, it consults the appropriate server's command table to fetch the process string and then sends this string over to the server for the server to determine which process to execute. These processes are stored in corresponding server install files (`*.ins`).

Parametric Processes

A parametric server process allows the information, a process needs, to be passed when the process is called. This ability to be able to pass process parameters allows direct control over the operation of a process. For parametric processes, each parameter has a value assigned and this parameter / value block is represented as Parameter = Name.

- For example `FileName = C:\Program Files\TestFile.Txt`.

To concatenate several parameters as a whole string, each parameter / value block is separated by the pipe | symbol.

- For example `Parameter1 = Name1 | Parameter2 = Name 2` etc.

Server Process Reference

There are two ways you can execute a process in a script

To execute a server process in a script, you need to use commands such as `ResetParameters` and `RunProcess` procedures or invoke the `Client.SendMessage` function.

RunProcess Example

```
ResetParameters;

AddStringParameter('OpenMode','NewFromTemplate');

AddStringParameter('ObjectKind,'Project');

RunProcess('WorkSpaceManager:OpenObject);
```

Client.SendMessage Example

```
Client.SendMessage('WorkspaceManager:OpenObject','OpenMode=NewFromTemplate |
ObjectKind=Project',1024,Nil);
```

Manipulating the Parameters of a Process in a Script

For parametric processes, each parameter has a value assigned and this parameter name / parameter value block is represented as Parameter = Name.

For example `FileName = C:\Program Files\TestFile.Txt`.

To concatenate several parameters as a whole string, each parameter name / value block is separated by the pipe | symbol.

For example `Parameter1 = Name1 | Parameter2 = Name 2` etc.

The `TParameterList` object helps you deal with parameter strings that consist of multiple parameters (names and values) in a script.

Example

```
//Parameters := Parameters + 'Orientation=1|Location.X=10000000|Location.Y=20000000'; //
overwrites the original parameter values.

P := TParameterList.Create;

P.ClearAllParameters;

P.SetState_FromString(Parameters);

P.SetState_AddParameterAsString ('Orientation','1');

P.SetState_AddParameterAsString ('Location.X' ,'10000000');

P.SetState_AddParameterAsString ('Location.Y' ,'20000000');

P.SetState_AddParameterAsString ('Designator' ,'dB1');

P.SetState_AddParameterAsString ('Comment'    ,'50pF');

Parameters := P.GetState_ToString;


IntegratedLibraryManager.PlaceLibraryComponent(SchLibRef,SchLibpath,Parameters);

P.Free;
```

Consult the *System API Reference* for more information on the `TParameterList` object and its methods.

Server Process Routines

To execute server processes from a script, you can use process routines provided by Altium Designer's Run Time Library in order to execute these server processes and their parameters.

Generally you would just use three `ResetParameters`, `AddStringParameter` and `RunProcess` routines to execute a server process with parameters in your script.

AddColorParameter

Declaration

Procedure `AddColorParameter`(Const Name: String; Red: Integer; Green: Integer; Blue: Integer);

Description

This procedure adds a color value parameter to the parameter buffer. This procedure is used to define a color for use by a process that requires a color parameter. The Color is a value where value = RedVal + 256*(GreenVal + 256*BlueVal) and Name is the name representing this color value.

AddIntegerParameter

Declaration

```
Procedure AddIntegerParameter(Const Name: String; Value: Integer);
```

Description

The `AddIntegerParameter` procedure defines a parameter with an Integer data type to the parameter buffer for use by a server / Process.

Example

```
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','Netlist');
    AddIntegerParameter('Index',5);
    AddStringParameter('ReturnGeneratedDocuments', 'True');
    RunProcess('WorkspaceManager:GenerateReport');
End;
```

AddLongIntParameter

Declaration

```
Procedure AddLongIntParameter(Const Name: String; Value: LongInt);
```

Description

The `AddLongIntParameter` procedure defines a parameter with a longint data type to the parameter buffer for use by a server / Process.

Example

```
Begin
    ResetParameters;
    AddLongIntParameter('LongIntValue',5);
    // code here
End;
```

AddSingleParameter

Declaration

```
Procedure AddSingleParameter(Const Name: String; Value: Single);
```

Description

The `AddSingleParameter` procedure defines a parameter with a single data type to the parameter buffer for use by a server Process.

Server Process Reference

Example

```
Begin
    ResetParameters;
    AddSingleParameter('SingleValue',5);
    // code here
End;
```

AddStringParameter

Declaration

```
Procedure AddStringParameter(Const Name, Value: String);
```

Description

This procedure adds a parameter with a string value to the parameter buffer. The Name parameter represents the name of the process parameter and the Value parameter represents the value of the process parameter.

Example

```
    ResetParameters
    Call AddStringParameter("Object","JumpToLocation10")
    Call RunProcess("PCB:Jump")


    ResetParameters
    Call AddStringParameter("ZoomLevel","2.0")
    Call RunProcess("PCB:Zoom")
```

AddWordParameter

Declaration

```
Procedure AddWordParameter(Const Name: String; Value: Word);
```

Description

The `AddWordParameter` procedure defines a parameter with a Word data type to the parameter buffer for use by a server / Process.

Example

```
Begin
    ResetParameters;
    AddWordParameter('WordValue',5);
    // code here
End;
```

GetColorParameter

Declaration

```
Procedure GetColorParameter(Const Name: String; Var Red: Integer; Var Green: Integer; Var
Blue: Integer);
```

Description

The `GetColorParameter` procedure retrieves the values of a color parameter as RGB values from the parameter buffer after running a process that returns a color value.

GetIntegerParameter

Declaration

```
Procedure GetIntegerParameter(Const Name: String; Var Value: Integer);
```

Description

The `GetIntegerParameter` procedure retrieves the value of an integer type parameter from the parameter buffer. This procedure after a process has been executed can return a resultant word value.

Example

```
Var
    ErrorCode : Integer;
    CommandLine : String;
    Result : Integer;
    NetlistName : String
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','Netlist');
    AddIntegerParameter('Index',5);
    AddStringParameter('ReturnGeneratedDocuments', 'True');
    RunProcess('WorkspaceManager:GenerateReport');
    GetIntegerParameter('Result', Result);
    If Result = 0 Then Exit;
    NetListName := GetStringParameter('File1', Result);
End;
```

GetLongIntParameter

Declaration

```
Procedure GetLongIntParameter(Const Name: String; Var Value: LongInt);
```

Description

The `GetLongIntParameter` procedure retrieves the value of a long int type parameter from the parameter buffer. This procedure after a process has been executed can return a resultant long int type value.

GetSingleParameter

Declaration

```
Procedure GetSingleParameter(Const Name: String; Var Value: Single);
```

Description

The `GetSingleParameter` procedure retrieves the value of a single type parameter from the parameter buffer. This procedure after a process has been executed can return a resultant single type value.

GetStringParameter

Declaration

```
Procedure GetStringParameter(Const Name: String; Var Value: String);
```

Description

The `GetStringParameter` procedure retrieves the value of a string type parameter from the parameter buffer. This procedure after a process has been executed can return a resultant string type value.

Example

```
Var
    ErrorCode : Integer;
    CommandLine : String;
    Result : Integer;
    NetlistName : String
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','Netlist');
    AddIntegerParameter('Index',5);
    AddStringParameter('ReturnGeneratedDocuments', 'True');
```

Server Process Reference

```
    RunProcess('WorkspaceManager:GenerateReport');
    GetIntegerParameter('Result', Result);
    If Result = 0 Then
        Exit;
    NetListName := GetStringParameter('File1', Result);
End;
```

GetWordParameter

Declaration

```
Procedure GetWordParameter(Const Name: String; Var Value: Word);
```

Description

The `GetWordParameter` procedure retrieves the value of a word type parameter from the parameter buffer. This procedure after a process has been executed can return a resultant integer value.

ResetParameters

Declaration

```
Procedure ResetParameters;
```

Description

The `ResetParameters` procedure clears the parameter buffer. Execute the procedure to reset the parameter buffer before setting parameters used by a process.

When you use any of the AddXXX Parameter procedures, the parameter declared is appended to the parameter buffer. When you run a process, any parameters that need to be passed to the process are read from the parameter buffer. Running a process, however, DOES NOT clear the parameter buffer. Therefore, it is important to use the `ResetParameters` procedure to clear the buffer of old values before placing a new series of parameters into the buffer.

Example in Delphiscript

```
Var
    ErrorCode : Integer;
    CommandLine : String;
    Result : Integer;
    NetlistName : String
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','Netlist');
    AddIntegerParameter('Index',5);
    AddStringParameter('ReturnGeneratedDocuments', 'True');
    RunProcess('WorkspaceManager:GenerateReport');
    GetIntegerParameter('Result', Result);
    If Result = 0 Then
        Exit;
    NetListName := GetStringParameter('File1', Result);
End;
```

RunProcess

Declaration

```
Procedure RunProcess(Const Command: String);
```

Description

The `RunProcess` procedure allows you to execute a server process. If the process invoked by this extension requires parameters to be passed to it, you must add the parameters to the parameter buffer using the AddXXXParameter functions before running the process.

If the process returns values, these will be placed in the return buffer and can be read using the GetXXXParameter functions.

The Command string takes on the following form: `Server:Process`

where Server is the name of the server the process is supplied by, and Process is the command name of the process. An example PCB:Zoom

Example in Delphiscript

```
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','Netlist');
    AddIntegerParameter('Index',5);
    AddStringParameter('ReturnGeneratedDocuments', 'True');
    RunProcess('WorkspaceManager:GenerateReport');
    GetIntegerParameter('Result', Result);


    If Result = 0 Then Exit;


    ShowMessage(GetStringParameter('File1', Result));
End;
```

Server Process Reference

Scripting System Processes

This section covers the Scripting processes and their parameters (if any).

RunScript Process

Description

This process is to used to run a script from the Run Process dialog (**DXP » Run Process**). There are two parameters in this case: the `ProjectName` and the `ProcName`. For the `ProcName` parameter, you need to specify the script filename and the main procedure in this script.

So the format is as follows: `ProcName = ScriptFileName>ProcedureName`. Note the GreaterThan symbol used between the script file name and the procedure name.

Parameters

ProjectName (string) Full path to the script project.

ProcName (string) A string containing two blocks separated by the >, Greater Than symbol. The first block is the script file name, and the second block is the procedure name within this script file.

Example

Process: ScriptingSystem:RunScript

Parameters : ProjectName = C: Program Files\Altium Designer\Examples\Scripts\DelphiScript Scripts\General\HelloWorld.PrjScr | ProcName = HelloWorldDialog>RunHelloWorld

RunScriptFile process

Description

Execute a DelphiScript unit script from Altium Designer (with a pas extension). Note, only DelphiScript unit scripts can be used - not these scripts that have forms.

Parameters

FileName (String) The path to a DelphiScript file (not the script project).

ProcName (String) The procedure name within the DelphiScript file.

Example

Process: ScriptingSystem:RunScriptFile

Parameters : FileName = c:\scripts\testascript.pas | ProcName = ProcedureName

RunScriptText process

Description

This process can be used to execute a series of commands within the Begin End; block.

Parameters

Text (String)

Example

Process: ScriptingSystem:RunScriptText

Parameters : Text = Begin RunApplication('Notepad.exe'); End;

Client Processes

This section covers the Client (System) processes and their parameters (if any). The Client module is the backbone of Altium Designer.

ArrangeAllWindows process

Description

The ArrangeAllWindows process can arrange opened windows (documents) in Altium Designer horizontally or vertically.

Parameters

Parameter	Value	Description
How	Horizontally, Vertically	Specifies how opened windows in Altium Designer can be arranged vertically or horizonally.

Example

Process: Client:ArrangeAllWindows

Parameters : How = Vertically

CascadeAllOpenWindows process

Description

The CascadeAllOpenDocuments process is not implemented

Parameters

N/A

ChangeTransparency process

Description

The ChangeTransparency process controls the transparency of floating windows.

Parameters

Parameter	Value	Description
Action	Toggle, On	If the Action parameter is set to Toggle, the status of the transparency of floating windows in Altium Designer is toggled. If parameter is set to On, the floating windows become transparent when cursor is hovering near.

Example

Process: Client:ChangeTransparency

Parameters : Action=Toggle

CustomizeResources process

Description

The CustomizeResources process allocates menus, shortcut keys and toolbar resources for current Editor. You can also toggle the visibility of certain panels, such as New Document, Differences List and Message List panels.

Running the CustomizeResources process with no parameters will invoke the Customizing XXX Editor dialog based on which document editor is active. For example if you run the process on a DelphiScript script, the *Customizing EditScriptDS Editor* will be invoked.

Parameters

Parameter	Value	Description
Action	Toggle, Show, None, Hide, ChangeType, New, SubMenu	

Server Process Reference

ObjectKind	Panel, Window,Tree, None	
Index	Integer	Indexes are based on the Editor's Resource file (*.RCS)
ID	TNewDocumentForm,TProjectGroupForm,TCompiledProjectForm,TCompiledErrorForm,TMessageListForm,TDifferencesListForm, TLSchematicTools, TLWiringTools,PowerObjects, DigitalObjects, TLAdvSimTools, TLSignalIntegrity, TLSimulationSources, TLSchLibTools, TLSChLibIEEETools, TLSchLibDrawingTools, SchObjectInspector, Filter, SchLibObjectInspector, LibFilter, LibraryBrowser, TSchLibPanel	

Example

Process: Client:CustomizeResources

Parameters: Action = Show | ObjectKind = Window

Favorites

Description

The Favorites process displays the Favorites view in Altium Designer.

Parameters

N/A

Example

Process: Client:Favorites

HelpAbout process

Description

The HelpAbout process displays the About dialog with the version number and copyright of Altium Designer.

Parameters

N/A

Example

Process: Client:HelpAbout

Licensing process

Description

The Licensing process invokes the Licensing dialog where you can setup or modify the license for the current copy of your Altium Designer application.

Example

Process: Client:Licensing

Navigate process

Description

The Navigate process is used to navigate or open documents in the Altium Designer application.

Parameters

Parameter	Value	Description
Mode	Go, Back, Forward, Stop, Refresh, Home	Specifies how the Navigation of open documents be carried out.
Address	String	File name or the Universal Resources Locator of the document.

Opening a PDF based document example

```
Process: Client : Navigate
Parameters: Mode = Go | Address = c:\dxp\help\AR0125 Honey, I Shrunk the Board.pdf
```

Opening a web page example

```
Process: Client : Navigate
Parameters: Mode = Go | Address = http://www.altium.com
```

QuitFromEDAClient process

Description

The QuitFromEDAClient process closes the current Altium Designer application

Parameters

N/A

Example

Process: Client:QuitFromEDAClient

RestoreLayout process

Description

The RestoreLayout process restores the Altium Designer desktop from a layout configuration file. Also accessed as a **View » Desktop Layouts** menu item on the menu.

Parameters

Type (Startup, Default)

The Default parameter denotes the Altium Designer default layout. The Startup parameter denotes the current saved layout.

Index (1..n)

Index = 1..n is one of the Altium Designer Workspace layout files.

RunCommonDialog process

Description

The RunCommonDialog process is used to execute a common dialog in Altium Designer such as Color dialog, File Open and File Save dialogs. The dialog when invoked also returns values such as the path of the file selected, whether dialog Ok or Cancel button clicked etc.

Parameters

Parameter	Value	Description
Dialog	FileOpenSave, Color	Specify the type of dialog. Two types of dialogs are supported. When Dialog = FileOpenSave, you need to specify which type of Open or Save dialog with the mode parameter.
Mode	0..4	0 = OpenFile dialog, 1 = File Save As dialog, 2 = Open File List, 3 = Open Document dialog, 4 = Open Expand List. The Dialog parameter has to be set to FileOpenSave.
Path	String	The dialog opens in this specified and valid path.
Prompt	String	Specifies the prompt string for this dialog
FileType1..FileTypeN	String	Specify the type of files to open/save. Example SetState_Parameter(Params, 'FileType1', 'Comma Separated Values (*.csv)|*.csv');

Parameters returned from Altium Designer after running the process

Parameter	Value	Description
Result	Boolean	If the result returned is false, then all other parameters in this table are not returned. If true, all other parameters can be returned. If the Dialog was set to Color, the Color

Server Process Reference

		parmater will be returned. If the Dialog was set to FileOpenSave, the Path, SelectedType, FileX parameters are returned.
Path	String	The full path to the document selected from the dialog.
SelectedType	String	When a file is selected, the selected type is returned denoting what type of file it is. For example when a PCB document is selected, PCB files (*.PCBDoc) is returned. This is based on the FileTypeX parameter in the Parameters table above.
FileX	String	Where X = 1..n for multiple files selected from the dialog.
Color	String	If the Dialog parameter was set to Color, then the color value will be returned.

Example

```
Process: Client:RunCommonDialog
Parameters : Dialog = FileOpenSave | Mode = 1 | Prompt = "Select a document and then click ok"
```

Example using VBscript

```
Var
  Path : String;
Begin
  ResetParameters
  AddStringParameter "Dialog", "FileOpenSave"
  AddStringParameter "Mode", "1"
  AddStringParameter "Prompt", "Select a document then click OK"
  RunProcess "Client:RunCommonDialog"
  GetStringParameter "Path", Path
End;
```

Example using DelphiScript to Run a Color dialog

```
Procedure RunADialogProcess;
Var
    S     : String;
    Value : Integer;
Begin
    ResetParameters;
    AddStringParameter('Dialog','Color'); // color dialog
    AddStringParameter('Color', '0');     // black color
    RunProcess('Client:RunCommonDialog');

    //Result value obtained from the RunCommonDialog's Ok or Cancel buttons.
    GetStringParameter('Result',S);
    If (S = 'True') Then
    Begin
        GetStringParameter('Color',S);
        ShowInfo('New color is ' + S);
    End;
End;
```

Example using DelphiScript to run a FileOpenSave dialog

```
SetState_Parameter(Params, 'Dialog',      'FileOpenSave');
SetState_Parameter(Params, 'Mode',        '3');  //Open document dialog.
SetState_Parameter(Params, 'Prompt',      'Open Comma Separated File');
```

```
SetState_Parameter(Params, 'FileType1',    'Comma Separated Values (*.csv)|*.csv');
MessageRouter_SendCommandToModule('Client:RunCommonDialog', Params, $FFFF, Nil);
GetState_Parameter(Params, 'Path',Filename);
```

Example using Delphi

```
StrPCopy(Parameter,
'Dialog=FileOpenSave|Mode=0|Prompt=PCAD ASCII File|DocumentType=All|FileType1=PCAD PCB File
(*.pcb)|' +
'FileType2=All Files (*.*)|Path='+FileName);
MessageRouter_SendCommandToModule('Client:RunCommonDialog',Parameter,4096,Nil);
GetState_Parameter(Params, 'Path',Filename);
```

RunProcess process

Description

The RunProcess process invokes the Run Process dialog.

SaveLayout process

Description

The SaveLayout process saves the current Altium Designer workspace layout as a layout file. Also accessed as a **View » Desktop Layouts** menu item on the DXP menu.

SetupEdaServers process

Description

The SetupEDAServers process invokes the EDA Servers dialog, where you can obtain more information about the installed servers.

Example

Process: Client:SetupEdaServers

SetupPreferences process

Description

The process invokes the EDA Servers dialog, where you can obtain more information about the installed servers.

Example

Process: Client:SetupPreferences

Parameters: Server=SCH

SwitchDocumentAndPanel process

Description

The SwitchDocumentAndPanel process switches the focus between a panel and its associated active document in Altium Designer.

Example

Process: Client:SwitchDocumentAndPanel

SwitchViews process

Description

The SwitchViews process opens the next or previous open (tabbed) window in Altium Designer.

Parameters

Parameter	Value	Description
ViewKind	Editor	This ViewKind parameter views certain types of documents. In this case, the Editor type denotes the editor documents such as Schematic, PCB documents.
Direction	Forward, Back	If Direction is specified to be Forward, then the next document is displayed. Otherwise the previously tabbed document (the document behind the current document) is

Server Process Reference

		displayed. To tab to the next window, the short cut keys are Shift Ctrl TAB.

Example

Process: Client:SwitchViews

Parameters : ViewKind=Editor | Direction = Forward

TileAllOpenDocuments process

Description

The TileAllOpenDocuments process tiles all open documents in the Altium Designer depending on which direction the tiling process is.

Parameters

Parameter	Value	Description
Mode	Horizontal, Vertical	If the parameter is not set, all open documents in Altium Designer are tiled optimally. If parameter is set to horizontal, then all documents are tiled horizontally, otherwise all documents are tiled vertically if the parameter is set to vertical.

Example

Process: Client:TileAllOpenDocuments

Parameters : Mode = Horizontal

ToggleClientStatusBar process

Description

The ToggleClientStatusBar process toggles the Command Status Bar on or off on the status section of Altium Designer.

Example

Process: Client:ToggleClientStatusbar

ToggleFloatingToolbars process

Description

This ToggleFloatingbars process toggles the visibility of Floating panels in Altium Designer when switching to different applications other than Altium Designer.

Parameters

Kind (Panels)

ToggleProcessStatusBar process

Description

The ToggleProcessStatusBar process toggles the Command Status Bar on or off on the status section of Altium Designer.

Example

Process: Client:ToggleProcessStatusbar

FPGA Flow Processes

This section covers the FPGAFlow processes and their parameters (if any).

BrowsePhysicalDevices

Description

Example

Process: FPGAFlow:BrowsePhysicalDevices

Configure

Description

The Configure process can configure a device list, synchronize with hard devices or nanoboards, edit the FPGA Workspace preferences, remove the current hard device, add a hard device or edit the current hard device.

Parameters

Parameter	Value	Description
Action	EditDeviceList, SynchronizeWIthHardWare, SynchronizeWithNanoBoards, EditPreferences, RemoveCurrentHardDevice, AddHardDevice, EditCurrentHardDevice	

Example

Process: FPGAFlow:Configure

Parameters : Action = EditPreferences

CurrentCoreCombo

Description

The CurrentCoreCombo process views and can change the current core in a FPGA project.

Example

Process: FPGAFlow:CurrentCoreCombo

DeviceAction

Description

The DeviceAction process can do a variety of device action tasks depending on the parameters supplied. For example, resetting all devices, pausing the processor, continuing the processor, single step in the processor code, download program to all on chip processors.

Parameters

Parameter	Value	Description
Target	BoardDevice, HardDevice, HardChain, SoftChain)	
Action	ShowAboutDialog, ShowViewer, ResetDevice, ChooseAndDownload, ResetDevice, PauseProcessor, ContinueProcessor, ResetProcessor, SingleStepProcessor,	

Server Process Reference

	DownloadProgram	
Index	integer	

Command Example

Process: FPGAFlow:DeviceAction

Parameters : Target = SoftChain | Action = PauseProcessor

Script Example

```
Procedure ShowInstrument(AIndex : Integer);

Begin

    ResetParameters;

    AddStringParameter('Target', 'SoftDevice'   );

    AddStringParameter('Action', 'ShowViewer'    );

    AddStringParameter('Index' , IntToStr(AIndex));

    RunProcess('FPGAFlow:DeviceAction');

End;


Procedure ShowAllInstruments;

Var

    DeviceIndex    : Integer;

    NexusWorkBench : INexusWorkbench;

Begin

    NexusWorkBench := GetNexusWorkbench;

    If NexusWorkBench = Nil Then Exit;

    If NexusWorkBench.GetSoftDeviceCount > 0 Then

        Count := NexusWorkBench.GetSoftDeviceCount;


    For DeviceIndex := 1 to Count Do

        ShowInstrument(DeviceIndex);

End;
```

FlowAction

Description

The FlowAction process can compile or rebuild a current hard device file and download to device if necessary or stop the current process flow actions.

Parameters

Parameter	Value	Description
Action	CompileHardDeviceFile, RebuildHardDeviceFile, CompileHardDeviceFileAndDownLoad, RebuildHardDeviceFileAndDownLoad, DownloadExistingHardDeviceFiles, CompileAllHardDeviceFiles, RebuildAllHardDeviceFiles, CompileAllHardDeviceFilesAndDownload, RebuildAllHardDeviceFilesAndDownload, DownloadAllExistingHardDeviceFiles, StopHardDeviceFlow	

Example

Process: FpgaFlow:FlowAction

Parameters : Action = StopHardDeviceFlow

RunDiagnostic

Description

The RunDiagnostic process depending on the action parameter can be invoked to scan a hard device chain, soft device chain, nanoboard chain, test cable connections, run the parallel port debugger or refresh the status of hard and soft devices.

Parameters

Parameter	Value	Description
Action	ScanFpga, ScanNexusChain, ScanNanoBoardChain, TestCableConnections, ParallelPortDebugger, RefreshStatus	

Example

Process: FPGAFlow:RunDiagnostic

Parameters : Action = RefreshStatus

Integrated Library Processes

This section covers the Integrated Library processes and their parameters (if any).

AddRemoveLibraries

Description

The `AddRemoveLibraries` process invokes the Available Libraries dialog allowing you to move up or down the order of installed libraries and add or remove installed libraries.

Parameters

N/A

CreateLibraryFromProject

Description

The CreateLibraryFromProject creates an integrated library of Schematic and PCB components from the current design project in Altium Designer.

EditParameterColumns

Description

FindComponent

Description

The `FindComponent` process invokes the Search Libraries dialog and you can define the search criteria to narrow the search for specific components in libraries.

Parameters

N/A

GenerateComponentReport

Description

Generates a report of components for the current library.

Parameters

N/A

PlaceCurrentComponent

Description

The `PlaceCurrentComponent` process places a currently selected component onto a document.

Parameters

N/A

See also

PlaceLibraryComponent process

PlaceLibraryComponent

Description

The `PlaceLibraryComponent` places a schematic, PCB or Integrated Library component from a specified library onto a current document (SCH or PCB documents).

Parameters

Parameter	Value	Description
LibReference	String	Specifies the LibReference of the Symbol.
Library	String	Specifies the path of the library. Can be just the filename, relative path to the filename or a full absolute path of the library.

ModelType	String	Specifies the model type. If modeltype = Sim, then ModelParameterNameX and ModelParameterValueX (where X = 1..N) values have to be specified.
ModelParameter Name0	String	Specifies the parameter name for the designated model.
ModelParameter Value0	String	Specifies the parameter value for the designated model.
Orientation	0..3	Specifies the orientation of the library component. 0 = 0 degrees, 1 = 90 degrees, 2 = 180 degrees and 3 = 270 degrees.

Example 1

Process: IntegratedLibrary:PlaceLibraryComponent

Parameters : LibReference=SN74F04D |Library=Texas Instruments\TI Logic Gate 2.IntLib|Orientation=0

Example 2

Process: IntegratedLibrary:PlaceLibraryComponent

Parameters : LibReference=CAP POL1 |Library=Miscellaneous Devices.IntLib |ModelType=SIM| ModelParameterName0=Value|ModelParameterValue0=10uF | Orientation=1

RefreshInstalledLibraries

Description

The `RefreshInstalledLibraries` process refreshes a current installed library or all installed libraries in Altium Designer to bring them up to date.

Parameters

Parameter	Value	Description
AllLibraries	True, False	If true, all installed libraries are refreshed, otherwise the current installed library is refreshed.

Example

Process: IntegratedLibrary:RefreshInstalledLibraries

Parameters : AllLibraries=True

Server Process Reference

PCB Processes

AlignComponents process		PlaceRoom process
Annotate process	GroupPrimitives process	PlaceSplitPlane process
ApertureLibrary process	FlipSelectedObjects process	PlaceString process
ArrangeComponents process	GotoLibraryComponent process	PlaceTrack process
AutopositionComponentTexts process	GroupPrimitives process	PlaceVia process
AutoRoute process	HideConnections process	PreviousComponent process
BackAnnotate	IdentifyNet process	PrimitiveSelect process
BoardInformation process	Import process	PrintDocument process
BreakTrack process	Jump process	ReAnnotate process
ChangeComponentName process	LastComponent process	Redo process
ChangeObject process	LibraryBrowse process	ReportBoardSpecs process
Clear process	ListAllSelectedPins process	ReportBOM process
ComponentRuleCheck process	ListComponents process	ReportLayerStackUpCompatibility process
ConfigureHeadsUpDisplay process	ListInternalPlanePins process	
ConnectSubnets process	ListNets process	ReportNetlistStatus process
ConvertSelected process	Manage Unions	ReportPickPlace process
Copy process	ManageLayerSets	ResetAllErrorMarkers process
CopyComponent process	ManualRoute process	ResetOrigin process
CreateApertureList	MeasureDistance process	ResetCamera process
CreateComponent	MeasureSelectedObjects process	RotateSelectedObjects process
CrossProbeChoose process	MoveAllComponentsToGrid process	RunQuery process
CrossProbeNotify process	MoveCursor process	RunQueryBuilder process
Cut process	MoveObject process	RunConvertedDrilledSMTPadsToMultiplayer
DeleteComponentFromLibrary process	Netlist process	
DeleteObjects process	NextComponent process	RunPolygonManager
DensityMap process	OffsetSelection process	RunScissors process
DeSelect process	OutlineSelectedObjects process	RunSpecctraForDos
DesignRuleCheck process	Paste process	RunSpecctraForWindows
DocumentPreferences process	PasteComponent process	Select process
EditClasses process	PinSwap process	SelectionMemory process
EditFromTo process	PlaceArc process	SetComponentReference process
EditInternalPlanes process	PlaceBoardOutline process	SetCurrentLayer process
EditRules process	PlaceComponent process	SetOrigin process
EngineeringChangeOrder process	PlaceComponentBody process	Setup process
EqualizeNetLengths process	PlaceComponentFromLibraryEditor process	SetupPreferences process
ExplodeComponent process		SetupPrinter process
Export process	PlaceComponentsFromFile process	ShowApplicableRules process
Fanout process	PlaceCoordinate process	ShoveComponents process
FilterSelect process	PlaceDifferentialPair process	ShowConnections process
FindTestPoints process	PlaceDimension process	ShowNetlistLength process
FirstComponent process	Place EmbeddedBoard	SnapGrid process
FlipSelectedObjects process	PlaceFill process	SwitchTo2D3D
GenerateLibraryReport	PlacePad process	TearDropSelectedPads process
GotoLibraryComponent process	PlacePolygonPlane process	TestGraphicsSystem process

		ToggleSelection process
		Undo process
		UnRoute process
		UpdateFootprints process
		UpdateRotationOnSelectedComponents process
		Zoom process

AlignComponents process

Description

The AlignComponents process aligns selected objects on a PCB document using specified parameters. There are different alignment parameters.

Parameters

Parameter	Value	Description
Alignment	MoveComponentsToGrid, MoveRoomsToGrid, Bottom, Left, Right, Top, CenterHorizontal, CenterVertical, ExpandHorizontal, ExpandVertical, ContractHorizontal, ContractVertical, SpreadHorizontal, SpreadVertical	If the Alignment parameter and its value is not supplied, then the Align Components dialog appears.

Example

Process: PCB:AlignComponents

Parameters : Alignment = CenterHorizontal

ArrangeComponents process

Description

Re arrange components within a room, rectangle, outside the board on a PCB document.

Parameters

Parameter	Value	Description
Action	ArrangeWithinRoom, ArrangeWithinRectangle, ArrangeOutsideBoard	The Action parameter with one of the values specifies how components can be arranged within a room on the PCB document.

Example

Process: PCB:ArrangeComponents

Parameters : Alignment = ArrangeWithinRoom

AutopositionComponentTexts process

Description

The AutopositionComponentTexts process auto positions component texts on the current PCB document

Parameters

Parameter	Value	Description
TextType	Name, Comment	If TextType value is not specified or invalid, then the Autoposition dialog appears.

Server Process Reference

Autoposition	LEFT-ABOVE, LEFT-CENTER, LEFT-BELOW, CENTER-ABOVE, CENTER-CENTER, CENTER-BELOW, RIGHT-ABOVE, RIGHT-CENTER, RIGHT-BELOW	Specifies how strings can be auto positioned in respect to components on the PCB document.

Autoroute process

Description

Perform an autoroute of the PCB board, or a section of the PCB board, or by a specified net etc. You can also save the current routing process and exit from Altium Designer.

Parameters

Parameter	Value	Description
Action	Start,Net,Connection,Component, SingleComponent,Area,Room,SingleRoom,Setup,Stop,Reset,Pause ,Restart, ExportRoutes	If SingleComponent value is specified for the Action parameter, you will need to specify the ContextObject which is usually Component If SingleRoom value is specified for the Action parameter, you will need to specify the ContextObject which is usually Room.
SeeFile	RenameDSN	If Action=ExportRoutes, you need to specify the SeeFile parameter.

Example

Process: PCB:AutoRoute

Parameters : Action=Setup

BoardInformation process

Description

Generates a board information report based on the current PCB or PCB library document.

Parameters

N/A

Example

Process: PCB:BoardInformation

BreakTrack process

Description

Breaks a whole focussed track into track segments on a PCB document.

Parameters

N/A

Example

Process: PCB:BreakTrack

ChangeComponentName process

Description

Renames a component name and some of its properties.

Parameters

N/A

Example

Process: PCB:ChangeComponentName

ChangeObject process

Description

Obtains object properties dialog where you can change the properties for the object on a PCB document.

Parameters

Parameter	Value	Description
Action	RepourAllPolygons, RepourSelectedPolygons,Repour SinglePolygon, RepourViolatingPolygons, ConvertHatchedPolygonsToSolid	Assign ContextObject = Polygon first for Single Polygon, Selected Polygons or All Polygons.
ContextObject	Polygon, Net	If ContextObject = Polygon, then only these single polygon, all polygons or selected polygons will be repoured. If ContextObject = Net, then Change Properties dialog for a Single Net will appear.

Example

Process: PCB:ChangeObject

Parameters : ContextObject = Polygon | Action=RepourAllPolygons

Clear process

Description

The Clear process is used to remove the selected objects from the current PCB document. The objects in the clipboard are not affected.

Parameters

N/A

ComponentRuleCheck process

Description

Checks whether components of a current PCB library are valid.

Parameters

N/A

Example

Process: PCB:ComponentRuleCheck

ConvertSelected process

Description

Convert either selected pads to vias or selected vias to pads.

Parameters

Action (PadsToVias, ViasToPads)

Example

Process: PCB:ConvertedSelected

Parameters : Action = ViasToPads

Copy process

Description

The Copy process is used to copy all selected objects to the clipboard. The Paste process can be used to place a copy of the selection back into any PCB document. However Copy with the Action=RoomFormat parameter can be used to copy a room format to other similar rooms.

Parameters

Server Process Reference

Action = RoomFormat

Example

Process PCB:Copy

CopyComponent process

Description

Copy a library component from a PCB library document which can be pasted as new components.

Parameters

N/A

Example

Process: PCB:CopyComponent

CrossProbeChoose process

Description

Cross probes or references a selected text string (such as a net identifier) in a linked document such as a schematic document from a PCB document.

Parameters

N/A

Example

Process: PCB:CrossProbeChoose

Cut process

Description

Cuts a selected object permanently from the PCB into the Clipboard. The original object that is cut is erased.

Parameters

N/A

Example

Process: PCB:Cut

DeleteComponentFromLibrary process

Description

Removes a currently focussed library component from the PCB library.

Parameters

N/A

Example

Process: PCB:DeleteComponentFromLibrary

DeleteObjects process

Description

The DeleteObjects process deletes any object from the current PCB document whether the objects are focussed or you are prompted to delete depending on the parameters.

Parameters

Parameter	Value	Description
Object	Prompt, Focused	Specifies how objects are deleted; whether they are focused or you are prompted to delete them.

Example

Process: PCB:DeleteObjects

Parameters : Object = Prompt

DensityMap process

Description

Obtains the density of the nets of a PCB document. The different colors indicate how dense the nets are within the regions of a PCB document.

Parameters

N/A

Example

Process: PCB:DensityMap

Deselect process

Description

The Deselect process is used to de-select objects in the current PCB and library editor window. Using parameters, all objects in the current document may be de-selected, all objects of a certain kind, objects inside or outside a specified area, all free objects, or all objects on a specific layer.

Parameters

Parameter	Value	Description
Scope	All, InsideArea, OutsideArea, Free, Layer	Defaults to All if no parameters are specified
InsideArea	Location1.X, Location1.Y, Location2.X, Location2.Y	All four parameters must be supplied as integers otherwise you are prompted to define the select rectangle.
OutsideArea	Location1.X, Location1.Y, Location2.X, Location2.Y	All four parameters must be supplied as integers otherwise you are prompted to define the select rectangle.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Keepout,Mechanical1,Mechanical2,Mechanical3,Mechanical4,Mechanical5,Mechanical6, Mechanical7,Mechanical8,Mechanical9,Mechanical10,Mechanical11,Mechanical12,Mechanical13,Mechanical14,Mechanical15,Mechanical16,Mid1,Mid10,Mid11,Mid12,Mid13,Mid14,Mid15,Mid16,Mid17,Mid18,Mid19,Mid20,Mid21,Mid22,Mid23,Mid24,Mid25,Mid26,Mid27,Mid28,Mid29,Mid30,Mid2,Mid3,Mid4,Mid5,Mid6,Mid7,Mid8,Mid9,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1,Plane2,Plane3,Plane4,Plane5,Plane6,Plane7,Plane8,Plane9,Plane10,Plane11,Plane12,Plane13,Plane14,Plane15,Plane16, Toppaste,Topsolder	However, you can only place arcs on available used layers of the current PCB document. The layer list is a list of all possible layers that the PCB editor can support. If no layer is specified then this process defaults to the current layer.

DesignRuleCheck process

Description

Either run the design rule checker to check the integrity of PCB rules on a PCB board, or obtain information on violations of the current PCB board.

Parameters

Parameter	Value	Description
InspectViolation	True, False	Defaults to false. If True, and Index not specified, all violations are shown.
Index	(1..n)	Specify which violation to display. If no value supplied, all violations are shown.

Server Process Reference

Example

Process: PCB : DesignRuleCheck

Parameters : InpsectViolation = True | Index = 1

DocumentPreferences process

Description

The `DocumentPreferences` process is used to define various PCB and library document settings, such as toggling the display of PCB layers and changing the visible grid values.

Parameter	Value	Description
'NoLayer','TopSignal','Mid1', 'Mid2', 'Mid3', 'Mid4', 'Mid5', 'Mid6', 'Mid7', 'Mid8', 'Mid9', 'Mid10','Mid11','Mid12','Mid13','Mid 14','Mid15', 'Mid16', 'Mid17', 'Mid18', 'Mid19', 'Mid20', 'Mid21', 'Mid22', 'Mid23', 'Mid24', 'Mid25', 'Mid26','Mid27','Mid28','Mid29','Mid 30', 'BottomSignal','TopOverlay','Botto mOverlay','TopPaste','BottomPast e','TopSolder','BottomSolder', 'Plane1', 'Plane2', 'Plane3', 'Plane4', 'Plane5', 'Plane6', 'Plane7', 'Plane8','Plane9', 'Plane10', 'Plane11', 'Plane12', 'Plane13','Plane14','Plane15','Plan e16', 'DrillGuide','KeepOut', 'Mechanical1', 'Mechanical2' ,'Mechanical3' ,'Mechanical4' ,'Mechanical5' ,'Mechanical6' ,'Mechanical7' ,'Mechanical8', 'Mechanical9', Mechanical10', 'Mechanical11', 'Mechanical12', 'Mechanical13', 'Mechanical14', 'Mechanical15', 'Mechanical16', 'DrillDrawing','MultiLayer','ShowD RCErrors','ShowVisibleGrid1','Sho wVisibleGrid2','ShowPadHoles','S howViaHoles'	True, False, Toggle	One of the three states sets the visibility of the specified layer. If true, the specified layer is displayed, if false, the layer is not displayed. If the Toggle parameter is used, the visibility of the layer is toggled.
SnapGrid	Real	Denotes the size of the snap grid. X and Y sizes are the same. The snap grid is used to define the alignment grid for manual movement and placement.
SnapGridX	Real	Denotes the snap grid X size (horizontal value set)
SnapGridY	Real	Denotes the snap grid Y size (horizontal value set)
ComponentGrid	Real	The ComponentGridSize is set in internal coordinates. (X and Y values are the same for this parameter).
ComponentGridX	Real	Denotes the component grid in X size in current PCB Units.
ComponentGridY	Real	Denotes the component grid in Y size in current PCB Units.
RoutingTrackGrid	Real	Denotes the track grid size in current PCB Units.

RoutingViaGrid	Real	Denotes the via grid size in current PCB Units.
VisibleGrid1	Real	Denotes the size of the first visible grid size in current PCB Units.
VisibleGrid2	Real	Denotes the size of the second visible grid size in current PCB Units.
VisibleGridMultFactor1	Real	Denotes the size of the visible grid multi factor1 in current PCB Units.
VisibleGridMultFactor2	Real	Denotes the size of the visible grid multi factor 2 in current PCB Units.
ElectricalGridRange	Real	Denotes the electrical grid size in current PCB Units. See MeasurementUnit parameter.
ElectricalGridMultFact	Real	Denotes the electrical grid multi factor value in current PCB Units. See MeasurementUnit parameter.
ElectricalGridEnabled	True, False, Toggle	Denotes whether the electrical grid is enabled or not.
MeasurementUnit	Imperial, Metric, Toggle	This denotes the default measurement units for the current PCB document. Defaults to imperial units. The default units are used to display any distance related information on screen or in reports. They are also used if units are not specified when editing a distance value in an object dialog field. The `Toggle` value changes from one unit to the other.
VisibleGridKind	Dots, Lines, Toggle	Denotes the visible grid type on the PCB document. By default the grid contains lines.
ShowSheet	True, False, Toggle	Denotes whether to display the sheet behind the board on the PCB document.
MaskLevel	Increase, Decrease	When the INCREASE parameter is applied, the MaskLevel is increased by one unit and when DECREASE parameter is applied, the MaskLevel is decreased by one unit.
ChangeFromLegacyToDXPPlane Mode	True, False	Change from legacy mode to DXP plane mode when importing from legacy designs and updating them.
Tab	LayerStack, DrillPairs, Mechanical, Layers, <empty>	When Tab is set to one of the following values, the appropriate dialog is displayed, for example, Tab = LayerStack, the Layer Stack manager dialog is invoked and displayed. Tab = DrillPairs, Drill Pairs dialog appears. Tab = Mechanical or Layers, the View Configurations dialog appears. When Tab is empty or invalid string, the Board Options dialog appears.

EditClasses process

Description

Display ObjectClasses dialog and you can manipulate object classes.

Parameters

N/A

Example

Process: PCB:EditClasses

EditRules process

Description

Displays the PCB design rules where you can manipulate and create new PCB rules.

Parameters

Server Process Reference

N/A

Example

Process: PCB : EditRules

EngineeringChangeOrder process

Description

Not implemented.

Parameters

N/A

EqualizeNetLengths process

Description

Equalize or make net lengths similar where possible in consideration of signal runs on a PCB document.

Parameters

N/A

Example

Process: PCB:EqualizeNetLengths

Export process

Description

This Export process exports a current PCB document open in Altium Designer into a different file format to be used in other applications other than Altium Designer.

Parameters

Parameter	Value	Description
Format	PROTEL NETLIST, SPECCTRA DESIGN, DXF, HYPERLINX, IPC, NETLIST, SHAPE, SELECTED	Specifies the file format for the current PCB document to be converted to.
Filename	String	Denotes the full path and filename of the PCB document to be exported.

Examples

Process: PCB:Export

Parameters: Format = HyperLynx | FileName = PCBBoard.PCBDoc

This automatically exports a file called PCBBoard.PCBDoc to the current directory in HyperLynx format.

Process: PCB:Export

Parameters: Format = Specctra Design| FileName = PCBBoard.dsn

This automatically exports a file called PCBBoard.PCBDoc to the current directory in HyperLynx format.

Fanout process

Description

Fanout process attempts to improve the quality of routing by arranging the pads and their tracks in a predetermined order.

Parameters

Parameter	Value	Description
Action	All, PowerPlaneNets, SignalNets, Room, Component, Selected, Net, Connection, Pad, SingleComponent, SingleRoom	If SingleRoom or SingleComponent values used, then specify the value (Room, Component) for the ContextObject parameter as well.
ContextObject	Component, Room	If Action = SingleRoom or SingleComponent, then

		ContextObject has to be assigned to Room or Component respectively.

Example

Process: PCB:Fanout

Parameters : Action = All

FilterSelect process

Description

Perform one of the many filtering processes and display the PCB Filter panel.

Parameters

Parameter	Value	Description
Edit	True, False	True to display the **PCB Filter** panel
Value	String	a valid expression or expressions.

Notes

Take note of the underscores _ for the _Edit_ and _Value_ name parameters.

Examples

Process: PCB:FilterSelect

Parameters : _Value_=IsTrack and OnBottom"

FindTestPoints process

Description

Find test points on a current PCB document or remove all test points from this PCB document. A testpoint is a point on a net that can be used for electrical continuity testing.

Parameters

Parameter	Value	Description
Action	ClearAllTestPoints	If no parameter supplied, test points are created.

Example

Process: PCB:FindTestPoints

Parameters: Action = ClearAllTestPoints

FirstComponent process

Description

Go to the first component in the library editor.

Parameters

N/A

Example

Process: PCB:FirstComponent

FlipSelectedComponents process

Description

Flip selected components across the axis (like a mirror).

Parameters

N/A

Example

Server Process Reference

Process: PCB:FlipSelectedComponents

GotoLibraryComponent process

Description

The GotoLibraryComponent process is used to go to the specified component in the specified library document in the Library Editor.

Parameters

FileName (String) The full path and file name of the library to be opened.

Footprint (String) Specifies the footprint.

GroupPrimitives process

Description

Group free primitives to an existing component on the current PCB document. You can also convert a group object into a set of free primitives, or create/break unions of components depending on parameters. Using component unions, unions are sets of components that you want to work as a block. The components in a union maintain their relative positions within the union as they are moved.

Parameters

Parameter	Value	Description
Action	Explode, CreateComponentUnion, BreakComponentUnion, BreakAllComponentUnions	If Action = Explode then use the following parameters; Object (Component, Coordinate, Dimension, Polygon)
Object	Component, Coordinate, Dimension, Polygon	By default, free selected primitives are grouped into a component you have selected.
ContextObject	Polygon, Component	To explode a single polygon or component, you need to specify the ContextObject.

Example

Process: PCB:GroupPrimitives

Parameters: Action=Explode | Object=Component

HideConnections process

Description

The HideConnections process is used to hide ratsnest connections for unrouted nets in the current PCB document.

Parameters

Parameter	Value	Description
Hide	All, Net, ComponentNets	All: Hides all ratsnest connections; Net: Hides a specified net and you will be prompted to choose which net to hide. ComponentNets: Hides nets connected to components only

IdentifyNet process

Description

The IdentifyNet process is used to display the net name for a chosen ratsnest connection or any object that belongs to a net on the status bar.

Parameters

N/A

Example

Process: PCB:IdentifyNet

Import process

Description

This Import process imports a document data of a different file format into a current PCB document in Altium Designer.

Parameters

Parameter	Value	Description
Format	PROTEL NETLIST, SPECCTRA DESIGN, DXF, HYPERLINX, IPC, NETLIST, SHAPE, SELECTED	Specifies the file format to be imported into Altium Designer.
Filename	String	Denotes the full path and filename of the PCB document to be imported.

Example

Process: PCB:Import

Parameters: Format = HyperLynx | FileName = PCBBoard.PCBDoc.

This automatically imports a file called PCBBoard.PCBDoc to the current directory.

Jump process

Description

The jump process can be used to jump to the reference point of a component, or jump to a selected primitive or group of primitives. You can also place up to 10 location markers and jump to one of them.

Parameters

Parameter	Value	Description
Object	Relative, Selected, Absolute, Location, Component, Net, Pad, String, DRCError,JumpToLocation1, JumpToLocation2, JumpToLocation3,JumpToLocation4,JumpToLocation5,JumpToLocation6,JumpToLocation7, JumpToLocation8,JumpToLocation9,JumpToLocation10, PlaceLocation1, PlaceLocation2,PlaceLocation3,PlaceLocation4,PlaceLocation5,PlaceLocation6,PlaceLocation7,PlaceLocation8,PlaceLocation9,PlaceLocation10	Relative: the jump process jumps to the reference point of the component Selected: depending on the Type parameter, the Jump process jumps to a selected primitive or a selected group of primitives. Absolute: jump to the absolute origin Location: jump to the specified location Component: jump to the specified component Net: jump to the specified net Pad: jump to the specified pad String: jump to the specified string object DRCError, jump to the specified string object. To define a location marker (1..n), execute a jump process with a location marker and then click on the screen to define the coordinates. To jump to this defined marker, execute the jump process with a JumpToLocationX parameter.
Type	First, Previous, Next, Last, FirstGroup, PreviousGroup, NextGroup, LastGroup	Jump to first, previous, next or last selected primitive or group only if Object = Selected.

Example

Process: PCB:Jump

Parameters : Object = Selected | Type = First

LastComponent process

Description

The LastComponent process is used to go to the last component in the current library document when in the Library Editor.

Server Process Reference

Parameters

N/A

Example

Process: PCB:LastComponent

LibraryBrowse process

Description

Used to graphically browse through the currently listed PCB libraries. The Library browse dialog box also allows placement of components.

Parameters

N/A

Example

Process: PCB:LibraryBrowse

ListAllSelectedPins process

Description

The ListAllSelectedPins process is used to list the component label and pin designators for all selected pads in the current PCB document. Entries are sorted by component designator then pad designator (e.g. U1-16).

Parameters

N/A

Example

Process: PCB : ListAllSelectedPins

ListComponents process

Description

The ListComponents process is used to generate a report listing all components placed on the current document. Components are listed by designators and comments. The report is automatically loaded and opened in the Text Editor.

Parameters

N/A

Example

Process: PCB:ListComponents

ListInternalPlanePins process

Description

This process when executed can display net information for an internal plane in Altium Designer. One of the 16 Internal planes can be listed.

Parameters

Parameter	Value	Description
InternalPlane	(1..16)	Only can display internal planes 1 to 16. Otherwise the Power Pins information dialog appears.

Example

Process: PCB:ListInternalPlanePins

Parameters : InternalPlane=2

ListNets

Description

This process when executed displays loaded nets information in a Nets Information dialog for the currently focussed PCB document.

Parameters

N/A

ManageLayerSets

Description

This process allows you to toggle the display of a group of layers in one go. These groups of layers are called Layer Sets. Altium Designer PCB Editor has 5 pre-defined layer sets, All Layers, Signal Layers, Plane Layers, Non SIgnal Layers and Mechanical layers.

Parameters

Parameter Name	Value	Description
SetIndex	0..4 (system) 5..X (user defined if any)	0 = All Layers 1 = Signal Layers 2 = Plane Layers 3 = Non Signal Layers 4 = Mechanical Layers 5 onwards correspond to any existing user defined layer sets.

Examples

Process: PCB:ManageLayerSets

Parameters : SetIndex=1

Notes

If there are no layers in the PCB document and you attempt to run the process with a setindex that displays those layers. Altium Designer will prompt you a warning dialog with "Can't Enable an empty layer Set! To rectify this issue, you need to display these layers first in the PCB document first. For example, some PCBs don't have internal planes and when you attempt to SetIndex to 2 for the ManageLayersSet process, you will get this warning dialog.

ManualRoute process

Description

Place a series of tracks to complete a connection from a pad to another pad.

Parameters

Parameter	Value	Description
mode	line	By default, interactive router is active when no parameters are supplied. To place a line, mode = line. To place a track (signal aware), mode = line | keepout = true.
keepout	True, False	Set it to true and when a signal aware track is being placed the keepout restrictions are observed. The keepout layer generally defines areas on the PCB document that you don't want automatically or manually routed, and this can include clearance areas around mounting hole pads or high voltage components for example.

Example

Process: PCB : ManualRoute

Parameters: mode = line \| keepout = true

MeasureDistance process

Description

Used to measure and display the distance between any two points on the PCB document. The measured distance will be displayed using the current units (mils or mm).

Parameters

Server Process Reference

Parameter Name	Value	Description
SnapToGrid	(True,False)	Defaults to True. If true, the cursor snaps to a grid point.
Repeat	(True,False)	Defaults to True
Primitives	(True,False)	Defaults to False. When Primitives is true, you are prompted to click on two primitives.

Example

Process: PCB:MeasureDistance

Parameters : Primitives = True

MeasureSelectedObjects process

Description

Used to calculate the total physical connection length of selected tracks within the current PCB document. Arcs will be included in the calculation (if selected), however the end point diagonal distance will be calculated, not the chord.

Parameters

N/A

Example

Process: PCB:MeasureSelectedObjects

MoveAllComponentsToGrid process

Description

Move all components to a specified grid.

Parameters

Parameter	Value	Description
Grid	Real	Specifies the grid value to move components to. If no parameters are used then a dialog box will prompt for a value.
GridX	Real	Specifies the grid X value to move components to. If no parameters are used then a dialog box will prompt for a value.
GridY	Real	Specifies the grid Y value to move components to. If no parameters are used then a dialog box will prompt for a value.

Example

Process: PCB:MoveAllComponentsToGrid

Parameters : Grid = 20

MoveCursor process

Description

Move a cursor across a PCB document programmatically.

Parameters

Parameter	Value	Description
Position	Up10, Down10, Left10, Right10, Up, Down, Left, Right	Up10 = Up 10 snap grid units. Up = Up 1 snap grid unit.

Example

Process: PCB:MoveCursor

Parameters : Position = Up

MoveObject process

Description

Move a specified group of objects across a PCB document.

Parameters

Parameter	Value	Description
Drag	True, False	If the Drag parameter is set to true, the object and its associated objects are moved together when it is being dragged.
Object	Component, Reroute, TrackEnd, Selection, PolygonVertices, Polygon, BoardOutlineVertices, BoardOutline, SheetAutoposition, Selection, Room, Room_Vertices	If TrackEnd, then the end of the track segment is repositioned. For the BoardOutlineVertices, you are prompted to edit the boardoutline, for the BoardOutline, you are prompted to move the outline.
ContextObject	Component, Polygon, Room	Specify the objects type when the MoveObject process is being carried out.

Example

Process: PCB:MoveObject

Parameters : Object = Drag

Netlist process

Description

Execute this process to edit nets, update free objects from nets, clear all nets, export netlist etc.

Parameters

Parameter	Value	Description
Action	EditNets, UpdateFreePrimitiveNets, ClearAllNets, ExportNetlistFromPCB, CreateNetlistFromConnectedCopper, CleanupNets, CleanUpSingleNets, AnalyseSingleNets	Specifies a specific action for the current netlist.

Example

Process: PCB:Netlist

Parameters: Action = EditNets

NextComponent process

Description

Go to the next component within the opened PCB library in the library editor.

Parameters

N/A

Example

Process: PCB:NextComponent

Paste process

Description

Paste the contents from the clipboard onto the current PCB board.

Parameters

Parameter	Value	Description

Server Process Reference

Mode	Special	Set the Mode to Special and this feature enables you to control what happens to certain object attributes when they are pasted back into the workspace. The special features allow you to create arrays of objects or create a panelized PCB layout. The *Paste Special* dialog appears allowing you to toggle the paste attributes.
OnCurrentLayer	True, False	If pasting on the current layer of the PCB document, specify true.
Array	True,False	If pasting an array of objects, specify true. An array of objects needs to be specified and copied to the clip board.
Action	RubberStamp	If Action = RubberStamp, you can click on the board multiple copies of the same object.

Example

Process: PCB:Paste

Parameters : OnCurrentLayer = True | Array = False | Action = RubberStamp

PasteComponent process

Description

Puts a selected component form a schematic into a library.

Parameters

N/A

Example

Process: PCB:PasteComponent

PinSwap process

Description

Invokes the Setup Pin Swapping dialog which attempts to minimise net crossovers and total routing length for FPGA based projects.

Parameters

N/A

Example

Process: PCB:PinSwap

PlaceArc process

Description

The PlaceArc process is used to place arc objects onto PCB and library editor documents, using the arc center or arc edge as the starting point. Arcs can be used to define component shapes on the overlay layers or on the mechanical and keepout layers to indicate the board outline, mounting holes or general documentation. Arcs can also be placed on signal layers as tracks to create curved corners.

Parameters

Parameter	Value	Description
Method	Circle, Edge,EdgeAnyAngle	Defaults to Center if no parameter supplied.
Location.X	Real	X-location of the arc center point.
Location.Y	Real	Y-location of the arc center point.
Width	Real	Specifies the width of the arc in current PCB units.
StartAngle	Real: 0-360	Specifies the starting angle of the arc
Radius	Real	Specifies the radius of the arc
EndAngle	Real: 0-360	Specifies the ending angle of the arc
Keepout	True, False	Defaults to False if no parameters supplied.

Selected	True, False, Toggle	Specifies whether the arc is selected or not.
DRCError	True, False, Toggle	Specifies whether the arc has a DRC error or not
Locked	True, False, Toggle	Specifies whether the arc is locked from being graphically edited or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Keepout,Mechanical1..16,Mid1..30, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,Toppaste,Topsolder	Specifies the layer the arc object is on.

Example

Process: PCB:PlaceArc

Parameters:

PlaceBoardOutline process

Description

Create and place a board outline on the PCB document either by deriving from the selected primitives on the board or by defining the vertices of the board outline.

Parameters

Parameter	Value	Description
Mode	BOARDOUTLINE_FROM_SEL_PRIMS	By default, you have to define the vertices on the PCB document to draw a board outline. If the mode is set to BOARDOUTLINE_FROM_SEL_PRIMS parameter then the board outline is generated by the selected primitives.

Example

Process: PCB:PlaceBoardOutline

PlaceComponent process

Description

The PlaceComponent process is used to place a library component from any open footprint library in the current PCB document window.

Parameters

Parameter	Value	Description
NameOn	True, False	Sets the designator visibility on or off.
CommentOn	True, False	Sets the comment visibility on or off.
NameAutoPosition(0..10)	Manual,TopLeft,CenterLeft, BottomLeft,TopCenter, CenterCenter, BottomCenter,TopRight, CenterRight, BottomRight CommentAutoPosition (0...10) Manual,TopLeft,CenterLeft, BottomLeft,TopCenter, CenterCenter, BottomCenter,TopRight,	Sets the auto position of the name (Designator) object relative to the component.

Server Process Reference

	CenterRight, BottomRight	
UnionIndex	Integer	Unions are sets of components that will be manipulated as a block for the PCB placement. Components in a union maintain their relative positions within the union as they are moved.
GroupNum	Integer	Not used internally. Can use for specific purposes such as a tag.
Height	Real	Sets the height of the component
Pattern	String	See the footprint parameter.
FileName	String	The filename of the component.
Footprint	String	Name of the component to be place from a library in the current library list.
Location.X	Real	Location of the component on the x axis.
Location.Y	Real	Location of the component on the y axis.
Rotation	Real: 0-360	Defaults to 0 degrees.
Layer	Top, Bottom	Defaults to Top layer.
Designator.Text	String	String of the designator
Designator.Visible	Boolean	Visibility of the designator
Designator.Location.X	Real	Location of the designator in X coordinate
Designator.Location.Y	Real	Location of the designator in Y coordinate
Designator.Height	Real	Height of the designator in the current PCB Units
Designator.Font	Default, SansSerif, Serif	Set to one of the available 3 fonts (default, SansSerif or Serif). You can use True Type font instead (UseTTFonts parameter instead).
Designator.Rotation	Real	Rotation of the designator in degrees
Designator.Mirror	Boolean	Set it to true to mirror the designator text. If Mirror is set to true, the text string is flipped across the vertical axis.
Designator.Width	Real	The width of the designator in pixels.
Designator.UseTTFonts	Boolean	Use True Type fonts if set to true, otherwise use Comment.Font parameter.
Designator.Bold	Boolean	Set it to true to set the designator font style to bold.
Designator.Italic	Boolean	Set it to true to set the designator font style to italic.
Designator.FontName	String	Specify the font name for the Designator if UseTTFont parameter is set to true.
Designator.WideString	String	Wide String of the Designator bypassing the character limit.
Comment.Text	String	String of the commentr
Comment.Visible	Boolean	Visibility of the comment
Comment.Location.X	Real	Location of the comment in X coordinate
Comment.Location.Y	Real	Location of the comment in Y coordinate
Comment.Height	Real	Height of the comment in the current PCB Units
Comment.Font	Default, SansSerif, Serif	Set to one of the available 3 fonts (default, SansSerif or Serif). You can use True Type font instead (UseTTFonts parameter instead).

Comment.Rotation	Real	Rotation of the comment in degrees
Comment.Mirror	Boolean	Set it to true to mirror the comment text. If Mirror is set to true, the text string is flipped across the vertical axis.
Comment.Width	Real	The width of the comment in pixels.
Comment.UseTTFonts	Boolean	Use True Type fonts if set to true, otherwise use Comment.Font parameter.
Comment.Bold	Boolean	Set it to true to set the designator font style to bold.
Comment.Italic	Boolean	Set it to true to set the designator font style to italic.
Comment.FontName	String	Specify the font name for the CommentTTFont parameter is set to true.
Comment.WideString	WideString	Wide String of the Comment bypassing the character limit.

Example

Process: PCB:PlaceComponent

Parameters : Footprint = RES10.55-7X2.8 | CommentAutoPosition = 6 | NameOn = False | Designator.Text = DesignatorText | Comment.Text = Commentary

PlaceComponentFromLibraryEditor process

Description

Places a currently selected component from a current library onto a PCB document.

Parameters

N/A

Example

Process: PCB : PlaceComponentFromLibraryEditor

PlaceComponentsFromFile process

Description

Position components based on the PIK (Pick and Place) file.

Parameters

N/A

Example

Process: PCB:PlaceComponentsFromFile

PlaceCoordinate process

Description

The `PlaceCoordinate` process is used to place coordinate markers onto the current PCB document.

Parameters

Parameter	Value	Description
Location.X	Real	The X location of the coordinate object.
Location.Y	Real	The Y location of the coordinate object.
Size	Real	The length of the marker of the coordinate object.
LineWidth	Real	The current line width used to draw the coordinate marker lines, which form a cross centered on the measurement point of the coordinate. Set a value in internal coordinates.
TextHeight	Real	The current height of the coordinate characters. The character width used to display or print the text is automatically

Server Process Reference

		proportioned to the height. A minimum height of 36mil (0.9mm) will allow the string to be legibly photoplotted. Set a value in internal coordinates
TextWidth	Real	The current coordinate text stroke width - the "thickness" of the lines used to produce the lettering. Set a value in internal coordinates for the text stroke width (range between 0.001 to 255mil).
Font	Serif, SansSerif, Default	The **Default** style is a simple vector font which supports pen plotting and vector photoplotting. The **Sans Serif** and **Serif** fonts are more complex and will slow down vector output generation, such as Gerber. Stroke-based fonts are built into the software and cannot be changed. All three fonts have the full IBM extended ASCII character set that supports English and other European languages.
Style	None, Normal, Brackets	The current setting for the display style of coordinate units. The available values are: **None**: No units are displayed, only the coordinates [eg. 1220, 3400]. **Normal**: Units are displayed with units after each value [eg. 1220mil, 3400mil]. **Brackets**: Units are displayed in brackets at the end of the coordinate [eg. 1220, 3400 (mil)].
Rotation	Real	Specifies the orientation of the coordinate object in degrees.
Selected	True, False, Toggle	Specifies the selected state of the object
DRCError	True, False, Toggle	Specifies whether this object has a DRC Error status or not.
Locked	True, False, Toggle	Specifies whether this object is locked from editing or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drill guide,Keepout,Mechanical1..16,Mid1..30,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,Toppaste,Topsolder	Specifies the layer the coordinate object will be on.

Example

Process: PCB:PlaceCoordinate

Parameters : Location.X = 3000 | Location.Y = 4000 | Size = 15 | Rotation = 45 | Layer = Top

PlaceDimension process

Description

The `PlaceDimension` process is used to place dimension objects onto the current PCB and library editor document. Dimensions are used for documentation and mechanical purposes to describe the physical dimensions of the PCB design. The dimension consists of arrows and lines made up of tracks and a string describing the actual distance measured between any two user-specified points. There are different types of dimension objects.

Parameters

Parameter	Value	Description
DimensionKind	Original, Linear, Angular, Radial, Leader, Datum, Baseline, Center, LinearDiameter, RadialDiameter	Specirfies the dimension type.
Location.X	Real	The X location of the dimension object.

Location.Y	Real	The Y location of the dimension object.
Size	Real	The length of the marker of the dimension object.
LineWidth	Real	The current line width used to draw the extension dimension lines. Set a value in internal coordinates.
TextHeight	Real	The current height of the characters in the dimension object. The character width used to display or print the text is automatically proportioned to the height. A minimum height of 36mil (0.9mm) will allow the string to be legibly photoplotted. Set a value in internal coordinates
TextWidth	Real	The current text stroke width - the "thickness" of the lines used to produce the lettering. Set a value in internal coordinates for the text stroke width (range between 0.001 to 255mil).
Font	Serif, SansSerif, Default	The **Default** style is a simple vector font which supports pen plotting and vector photoplotting. The **Sans Serif** and **Serif** fonts are more complex and will slow down vector output generation, such as Gerber. Stroke-based fonts are built into the software and cannot be changed. All three fonts have the full IBM extended ASCII character set that supports English and other European languages.
Style	None, Normal, Brackets	The current setting for the display style of dimension units. The available values are: **None**: No units are displayed, only the coordinates [eg. 1220, 3400]. **Normal**: Units are displayed with units after each value [eg. 1220mil, 3400mil]. **Brackets**: Units are displayed in brackets at the end of the coordinate [eg. 1220, 3400 (mil)].
Rotation	Real	Specifies the orientation of the dimension object in degrees.
Selected	True, False, Toggle	Specifies the selected state of the object
DRCError	True, False, Toggle	Specifies whether this object has a DRC Error status or not.
Locked	True, False, Toggle	Specifies whether this object is locked from editing or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing, Drillguide,Keepout,Mechanical1..16,Mid 1..30,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,Toppaste,Topsolder	Specifies the layer the dimension object will be on.

Example

Process: PCB:PlaceDimension

Parameters : DimensionKind = Radial

PlaceFill process

Description

The PlaceFill process is used to place a rectangular solid fill area onto PCB or library editor documents.

Parameters

Parameter	Value	Description

Server Process Reference

Location1.X	Real	The X location of the fill object from left bottom
Location1.Y	Real	The Y location of the fill object from left bottom
Location2.X	Real	The X location of the fill object from top right
Location2.Y	Real	The Y location of the fill object from top right
Rotation	Real	Specifies the orientation of the fill object in degrees.
Selected	True, False,Toggle	Specifies the selected state of the object
Keepout	True, False	Specifies whether this object is used as a keep out object or not.
DRCError	True, False,Toggle	Specifies whether this object has a DRC Error status or not.
Locked	True, False,Toggle	Specifies whether this object is locked from editing or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste, Bottomsolder, Drilldrawing, Drillguide, Keepout, Mechanical1..16, Mid1..30, Bottompaste, Bottomsolder, Drilldrawing, Drillguide, Plane1..16, Toppaste, Topsolder	Specifies the layer the fill object will be on.

Example

Process: PCB:PlaceFill

Parameters : Location1.X = 1000 | Location1.Y = 1000 | Location2.X = 2000 | Location2.Y = 2000 | Layer = Top | Rotation = 20

PlacePolygonPlane process

Description

The PlacePolygonPlane process is used to create polygon objects on the current document window. Polygon planes (or copper pours) are similar to area fills, except that they can fill irregularly shaped areas of a board and can connect to a specified net as they are poured.

Parameters

Parameter	Value	Description
PourOver	Integer	0 = None 1 = SameNet 2 = SameNetPolygons
RemoveDead	True, False,Toggle	Set it to true to remove any regions of "dead" copper within the polygon. Dead copper is created when an area of the polygon can not be connected to the selected net.
GridSize	Real	Specifies grid size used when pouring polygon
TrackWidth	Real	Specifies track width used when pouring polygon
MinPrimLength	Real	The current minimum allowable primitive length for the polygon pour. Polygons pours can contain many small primitives to create smooth edges around existing objects on the board. A larger value will give faster pour

		times, screen redraws and output generation, at the expense of the smoothness of curved polygon edges.
HatchStyle	90Degree, 45Degree, Vertical, Horizontal, None	Specifies the hatching style of the polygon pour.
Netname	String	Specifies the netname of the net that the polygon is connected to.
UseOctagons	True, False,Toggle	Set the UseOctagon parameter true to surround enclosed pads with octagons when pouring the polygon. This results in smaller Gerber files and faster photoplotting. Set it to false to use arcs instead.
PolygonType	Polygon, Split Plane	Set the Polygon type to Polygon or split plane.
Kind(n)	0, 1	Segment kind, 0 specifies track segment, 1 specifies arc segment
Vx(n)	Real	Starting point of vertex (n)
Vy(n)	Real	End point of vertex (n)
Cx(n)	Real	X center point of arc segment (n)
Cy(n)	Real	Y center point of arc segment (n)
SA(n)	Real	Starting Angle of arc segment (n)
EA(n)	Real	End Angle of arc segment (n)
R(n)	Real	Radius of arc segment (n)
Selected	True, False,Toggle	Specifies the selected state of the object
DRCError	True, False,Toggle)	Specifies whether this object has a DRC Error status or not.
Locked	True, False,Toggle	Specifies whether this object is locked from editing or not.
PrimitiveLock	True, False,Toggle	Set this to true to allow all tracks that form the polygon pour to be treated as a single object. If you want to individually edit the tracks that make up the polygon, set the value to false.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Keepout, Mechanical1..16,Mid1..30,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,,Toppaste,Topsolder	Specifies the layer the polygon object will be on.

Example

Process: PCB:PlacePolygonPlane

Parameters :

See also

PlaceSplitPlane process

PlaceRoom process

Description

Places a room object on the PCB document where specified objects are grouped together inside this room. This room often represents a channel from a schematic project.

Parameters

Parameter	Value	Description

Server Process Reference

ModeFit	Polygonal_room, Create_Centers, Create_NonOrtho, Create_Ortho, Create_Rectangle, Fit_Centers, Fit_NonOrtho, Fit_Ortho, Fit_Rectangle	Create_Centers: Creates a Room from Component Centers
		Create_NonOrtho: Create Non-Orthogonal Room from Components
		Create_Ortho: Create Orthogonal Room from Components
		Create_Rectangle: Create Rectangular Room from Components
		Fit_Centers: Wrap Room Around Member Component Centers
		FIt_NonOrtho: Wrap Non-Orthogonal Room Around Member Components
		Fit_Ortho: Wrap Orthogonal Room Around Member Components
		Fit_Rectangle Wrap Rectangular Room Around Member Components
ContextObject	Room	

Example

Process: PCB:PlaceRoom

Parameters :ModeFit = Fit_Rectangle

PlaceSplitPlane process

Description

Place a split power/ground plane on the current PCB document. This process is used to "split" internal power planes so that they can be shared between multiple power rails. You need to have internal planes set up in your PCB document first before you can place split planes.

Parameters

Parameter	Value	Description
PourOver	Integer	0 = None 1 = SameNet 2 = SameNetPolygons
RemoveDead	True, False,Toggle	Set it to true to remove any regions of "dead" copper within the polygon. Dead copper is created when an area of the polygon can not be connected to the selected net.
GridSize	Real	Specifies grid size used when pouring polygon
TrackWidth	Real	Specifies track width used when pouring polygon
MinPrimLength	Real	The current minimum allowable primitive length for the polygon pour. Polygons pours can contain many small primitives to create smooth edges around existing objects on the board. A larger value will give faster pour times, screen redraws and output generation, at the expense of the smoothness of curved polygon edges.
HatchStyle	90Degree, 45Degree, Vertical, Horizontal, None	Specifies the hatching style of the polygon pour.
Netname	String	Specifies the netname of the net that the polygon is connected to.
UseOctagons	True, False,Toggle	Set the UseOctagon parameter true to surround enclosed pads with octagons when pouring the polygon. This results in smaller Gerber files and faster photoplotting. Set it to false to use arcs instead.
PolygonType	Polygon, Split Plane	Set the Polygon type to Polygon or split plane.
Kind(n)	0, 1	Segment kind, 0 specifies track segment, 1 specifies arc segment
Vx(n)	Real	Starting point of vertex (n)
Vy(n)	Real	End point of vertex (n)
Cx(n)	Real	X center point of arc segment (n)

Cy(n)	Real	Y center point of arc segment (n)
SA(n)	Real	Starting Angle of arc segment (n)
EA(n)	Real	End Angle of arc segment (n)
R(n)	Real	Radius of arc segment (n)
Selected	True, False,Toggle	Specifies the selected state of the object
DRCError	True, False,Toggle)	Specifies whether this object has a DRC Error status or not.
Locked	True, False,Toggle	Specifies whether this object is locked from editing or not.
PrimitiveLock	True, False,Toggle	Set this to true to allow all tracks that form the polygon pour to be treated as a single object. If you want to individually edit the tracks that make up the polygon, set the value to false.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Keepout, Mechanical1..16,Mid1..30,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,,Toppaste,Topsolder	Specifies the layer the split plane object will be on.

Example

Process: PCB:PlaceSplitPlane

Parameters :

PlaceString process

Description

The PlaceString process is used to place a line of text onto PCB or library editor documents. Special strings allow the designer to place generic, non specific text which is interpreted when printing.

Parameters

Parameter	Value	Description
Height	Real	The height of the string. Defaults to 60mils
Width	Real	The width of the letters in a string. Defaults to 10mils
Font	String: Default, SansSerif, Serif	The type of font for the string object.
Rotation	Real: 0-360	Specifies the orientation of the string object in degrees.
Mirror	True, False	Specifies the mirrored status of the string. If Mirror is set to true, the text string is flipped across the vertical axis.
Text	String	Upto 255 characters. Special strings can be used. See footnote for more information on special strings.
Location.X	Real	The X location of the string object from left bottom
Location.Y	Real	The Y location of the string object from left bottom
Selected	True, False,Toggle	Specifies the selected state of the object
DRCError	True, False,Toggle	Specifies whether this object has a DRC Error status or not.

Server Process Reference

Locked	True, False,Toggle	Specifies whether this object is locked from editing or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Keepout, Mechanical1..16,Mid1..30,Bottompaste,Bottomsolder,Drilldrawing,Drillguide,Plane1..16,,Toppaste,Topsolder	Specifies the layer the string object will be on.

Notes

The available special strings for the Text field of the text object:

- .Application_BuildNumber – the version of Altium Designer that the PCB is currently loaded in. When generating
- .Gerber output, this string will record the software build that the design was created on
- .Arc_Count - the number of arcs on the PCB
- .Comment - the comment string for a component (used in designing component footprints)
- .Component_Count - the number of components on the PCB
- .ComputerName - The name of the machine that the PCB is currently loaded in
- .Designator - the designator string for a component (used in designing component footprints)
- .Fill_Count - the number of fills on the PCB
- .Hole_Count - the number of drill holes on the PCB
- .Layer_Name - the name of the layer the string is placed on
- .Legend - a symbol legend for mechanical drill plots. This string is only valid when placed on the Drill Drawing layer
- .Net_Count - the total number of different nets on the PCB
- .Net_Names_On_Layer - the names of all nets on the specific layer. This string is only valid when placed on an internal plane layer
- .Pad_Count - the number of pads on the PCB
- .Pattern - the names of the component footprints used on the PCB
- .Pcb_File_Name - the path and file name of the PCB document
- .Pcb_File_Name_No_Path - the file name of the PCB document
- .Plot_File_Name - When generating Gerber output, this string identifies the file name of the Gerber plot file. When generating printed output, this string identifies the layer depicted within the output. When generating ODB++ output, this string identifies the name of the parent folder in which the files are stored
- .Poly_Count - the number of polygons on the PCB (consisting of polygon pours, internal planes and split planes)
- .Print_Date - the date of printing/plotting
- .Print_Scale - the printing/plot scale factor
- .Print_Time - the time of printing/plotting
- .Printout_Name - the name of the printout
- .SlotHole_Count - the number of slotted holes on the PCB
- .SquareHole_Count - the number of square holes on the PCB
- .String_Count - the number of strings on the PCB
- .Track_Count - the number of tracks on the PCB
- .VersionControl_RevNumber - the current revision number of the document. Version control must be used for this string to contain any information
- .Via_Count - the number of vias on the PCB.

The .DESIGNATOR and .COMMENT special strings are added to the component in the library. Use these if you need to control the location of these attributes on a component. They can be placed on any layer. The standard designator and comment can be hidden if desired.

Example

Process: PCB:PlaceString

Parameters : Location.X = 3000 | Location.Y = 3000 | Layer = Top | Height = 50 | Width = 10 | Font = Default | Text = Testing | Rotation = 45

PlaceTrack process

Description

The PlaceTrack process places a free track on a current PCB document.

Parameters

Parameter	Value	Description
Width	Real	The Width of the track object
Location1.X	Real	The X location of the track object from left bottom
Location1.Y	Real	The Y location of the track object from left bottom
Location2.X	Real	The X location of the track object from top right
Location2.Y	Real	The Y location of the track object from top right
UserRouted	True, False,Toggle	Specifies whether this object has been manually routed or not.
TearDrop	True, False,Toggle	Specifies whether this object has been used for teardrop
Selected	True, False.Toggle	Specifies the selected state of the object
DRCError	True, False,Toggle	Specifies whether this object has a DRC Error status or not.
Locked	True, False,Toggle	Specifies whether this object is locked from editing or not.
Layer	Current, Top, Bottom, Topoverlay, Multilayer, Bottomoverlay, Connect, Bottompaste, Bottomsolder, Drilldrawing, Drillguide, Keepout, Mechanical1..16, Mid1..30, Bottompaste, Bottomsolder, Drilldrawing, Drillguide, Plane1..16, Toppaste, Topsolder	Specifies the layer the track object will be on.

Example

Process: PCB:PlaceTrack

Parameters : Location1.X = 1000 | Location1.Y = 1000 | Location2.X = 2000 | Location2.Y = 2000 | Layer = Top | Width = 20

PlaceVia process

Description

The PlaceVia process is used to place a free via onto the current PCB or library editor document.

Parameters

Parameter	Value	Description
Diameter	Real	Specifies the diameter of the via
HoleSize	Real	Specifies the holesize of the via

Server Process Reference

Location.X	Real	The X location of the via object from the center of this via
Location.Y	Real	The Y location of the track object from the center of this via
UserRouted	True, False,Toggle	Specifies whether this object has been manually routed or not.
StartLayer	Top, Mid1..Mid30, Bottom	Specifies the top layer the via object will be on.
EndLayer	Top, Mid1..Mid30,Bottom	Specifies the bottom layer the via object will be on.

Example

Process: PCB:PlaceVia

Parameters : Diameter = 40 | HoleSize = 28 | Location.X = 1000 | Location.Y = 1000 | StartLayer = Top | EndLayer = Bottom

PreviousComponent process

Description

Go to the previous component within the currently open PCB library.

Parameters

N/A

Example

Process: PCB:PreviousComponent

ReAnnotate process

Description

Reannotate components to update the designators on the PCB document. You can also reverse the component designators.

Parameters

Action (ReverseDesignators, '')

Example

Process: PCB:ReAnnotate

Parameters : Action = ReverseDesignators

Redo process

Description

Redoes the previous operation in Altium Designer.

Parameters

N/A

Example

Process: PCB:Redo

ReportBoardSpecs process

Description

The ReportBoardSpecs process is used to generate an ASCII report file of a library or a component.

Parameters

ReportKind (Component, Library)

Example

Process: PCB : ReportBoardSpecs

Parameters : ReportKind = Component

ReportNetlistStatus process

Description

Generates a report outlining the status of the netlist for a PCB document.

Parameters

N/A

Example

Process: PCB:ReportNetlistStatus

ResetAllErrorMarkers process

Description

Reset all error markers on the PCB document.

Parameters

N/A

Example

Process: PCB:ResetAllErrorMarkers

ResetOrigin process

Description

Resets the origin of the PCB board.

Parameters

N/A

RotateSelectedObjects process

Description

Rotate selected objects by 90 degree increments.

Parameters

N/A

Example

Process: PCB : RotateSelectedObjects process.

RunQuery process

Description

Execute a query statement to affect a group of objects on a PCB document.

Parameters

Parameter	Value	Description
Expr	string	The Expr refers to the valid expression statement or statements (with OR and AND keywords in the expression) that the Query engine parses first before taking action. Refer to the Query documentation for more details on numerous parameters for this RunQuery process.
Action	FindSimilar, FindSimilarUnderCursor	If Action is set to FindSimilar, you are prompted to choose a template PCB design object before the Find Similar Object dialog appears. If Action is set to FindSimilarUnderCursor, those objects that appear under the cursor
Zoom	True,False	If true, and the query expression is valid, the objects affected by the query are zoomed into.
Mask	True,False	If true, and the query expression is valid, the objects affected by the query are masked.
Select	True,False	If true, and the query expression is valid, the objects affected by the query are selected.
Clear	True,False	If true, the current query is cleared.
Source	Favorite, History,	To choose a favorite query and use it for querying, set the Source to Favorite /

Server Process Reference

	Example	History and specify the Index (indexed favorite), Apply, Zoom, Mask, Select, Clear parameters.
Apply	True, False	If true, the Query expression are processed immediately. If false or parameter not set, the Query expression is added to the PCB filter and you then can set the filter options and then press Apply button. You need to have the Expr, and other parameters set up first.

Example

Process: PCB:RunQuery

Parameters :Expr=IsDesignator And (Rotation <> 0) And (Rotation <> 360)|Select=True|Mask=True

Process: PCB:RunQuery

Parameters: Apply=True|Source=Example|Expr=IsComment And (Hide = True)|Zoom=True|Select=True'

There are existing query examples in the *Expression Manager* dialog from the **PCB Filter** panel.

An example of the PCB query can be found in the `Examples\Scripts\Delphi Scripts\PCB` examples folder of the Altium Designer installation.

RunQueryBuilder process

Description

Invokes a Query Builder dialog that simplify the process of building a query.

Parameters

Parameter	Value	Description
LaunchMode	UnderCursor	Defaults to blank. If UnderCursor, the QueryBuilder dialog appears with a query for the current state of selection of objects on a PCB document.
RunRuleWizard	True, False	If True, the Query Builder wizard appears and assist you with building a query.

Example

Process: PCB : RunQueryBuilder

RunScissors process

Description

Slice a polygon plane or a room on a PCB document.

Parameters

Parameter	Value	Description
Mode	Cut_Polygons, Cut_Rooms, SmartCut_Rooms	Cut_Polygons = slice polygons Cut_Rooms =Slice a room object SmartCut_Rooms = Slice a room object including those created by the Component Class Creation.
ContextObject	Room, Polygon	You need to provide the context for the Scissors process. Use Room or Polygon.

Example

Process: PCB : RunScissors

Parameters : Mode = Cut_Rooms

Select process

Description

Perform a selection on a specified group of objects within a specified boundary on a PCB document.

Parameters

Parameter	Value	Description
Scope	InsideArea, OutsideArea, All, Board, Net, ConnectedCopper, PhysicalConnection, Layer, Free,Locked, OffGridPads, RoomConnections, ComponentConnections, ComponentNets	If RoomConnections, ComponentConnections or ComponentNets then you might need to specify the ContextObject parameter where possible values are Room or Component.

Example

Process: PCB:Select

Parameters :Scope = All

SelectionMemory process

Description

Perform one of the many selection memory processes including displaying the Selection Memory dialog.

Parameters

Parameter	Value	Description
Action	ShowDialog, Store, Recall, StorePlus, RecallPlus, Clear, Apply	The action parameter specifies how the operation for the SelectionMemory is to be carried out. You will need to specify the Index parameter as well.
Index	1..8	Choose one of the 8 memory locations.

Notes

There are up to 9 Store and Recall memory states for the Selection Memory dialog.

Example

Process: PCB : SelectionMemory

Parameters : Action = ShowDialog | Index = 1

SetComponentReference process

Description

The SetComponentReference process sets the reference point of a component object, whether by the pin 1, center or the location. When a component is dragged or rotated, it is done by the reference point of this object.

Parameters

Parameter	Value	Description
Location	Pin, Center	Specify the location parameter to set the reference point of a component object.

Example

Process: PCB:SetComponentReference

Parameters: Location = Pin

SetCurrentLayer process

Description

Display next enabled signal layer, or an enabled layer from the current PCB document.

Parameters

Server Process Reference

Parameter	Value	Description
LayerName	NextSignal, Next, Previous, PreviousSignal	Specify the layername to the next signal layer, next available layer, previous layer or previous signal layer in respect to the current layer.

Example

Process: PCB:SetCurrentLayer

Parameters :LayerName =NextSignal

SetOrigin process

Description

The SetOrigina process sets the origin of the PCB board.

Parameters

Parameter	Value	Description
Location.X	Real	Specifies the X coordinate of the origin on the PCB document.
Location.Y	Real	Specifies the Y coordinate of the origin on the PCB document.

Example

Process: PCB:SetOrigin

Parameters : Location.X = 1000 | Location.Y = 1000

SetupPreferences process

Description

The SetupPreferences process configures system settings, display settings, single layer mode and routing mode etc. As therea re a large number of parameters for this process, so it is broken up into a number of tables according to their categories:

- General Parameters
- PCB Options Parameters
- PCB Layer Colors Parameters
- PCB Show/Hide Display Parameters

PCB General Parameters

Parameter	Value	Description
DefaultPrimitives	SAVE, LOAD	If the DefaultPrimitives parameter set to 'SAVE', the default primitives are saved to a default file, otherwise if the parameter is set to 'LOAD', the default primitives are loaded into the PCB Editor – Defaults page of the Preferences dialog. Note, the LOAD or SAVE values must be in Uppercase.
Tab	Display, Colors, SystemColors, PreferredWidths, PreferredVias, Show/Hide, Defaults, Board Insight Display, Board Insight Modes, Board Insight Lens, Interactive Routing, Truye Type Fonts, Mouse Wheel Configuration, Layer Colors, PCB Legacy 3D.	Defaults to running Preferences dialog with the Options page active if no parameters supplied. Otherwise the tab parameter sets the page of the preferences dialog active.

PCB Options Parameters Table

Parameter	Value	Description
SelectHidden	True, False, Toggle	Not Implemented.
		Use the Toggle value to toggle from True to False or False to True.
SnapToCenter	True, False, Toggle	If SnapToCenter parameter is set to True, the cursor jumps automatically to a defined reference point on the object when you select it and be "held" by that point as you reposition it. For free pads or vias, the cursor will snap to the center of the object, with components, the cursor snaps to the reference point of the component. For tracks, the cursor snaps to the vertex point.
		Otherwise if the parameter is set to False, objects will be "held" by the point at which you click on them. Use the Toggle value to toggle from True to False or False to True.
SmartComponentSnap	True, False, Toggle	If the parameter is set to True, the cross hair cursor appears on the nearest pad of this associated component in respect to where the cursor is. When False, the cross hair cursor always appears on the pad reference point of this component when it is clicked on. Use the Toggle value to toggle from True to False or False to True.
SmartDrag	True,False, Toggle	Set the SmartDrag parameter to true to preserve the angles of tracks and arcs when you are dragging them on the PCB document.
ClickClearsSelection	True,False, Toggle	Set the ClickClearsSelection parameter if you want to deselect all design objects by clicking any where on the PCB workspace.
		Otherwise set it to false if you do not want to have this click anywhere to deselect all ability and you can click on a selected design object to deselect it without affecting other selected design objects and the selection process is cumulative.
ToggleMustHoldShiftToSelect	True,False, Toggle	Set the parameter to true so that single design objects can be selected by Shift and Click. Set it to false and objects can clicked to be selected.
		See HoldShiftToSelect_<DesignObject> parameters for finer control.
MustHoldShiftToSelect	True,False, Toggle	Set the parameter to true so that single design objects can be selected by Shift and Click. Set it to false and objects can clicked to be selected.
		See HoldShiftToSelect_<DesignObject> parameters for finer control.
NearestComponent	True,False, Toggle	Not implemented.
AutoPan	True,False, Toggle	If the parameter is set to True, the Auto panning of the PCB document can be carried out. When False, the PCB document cannot be auto panned.. Use the Toggle value to toggle from True to False or False to True.
RemoveDuplicates	True,False, Toggle	When this parameter to check for and remove duplicate primitives when the system is preparing data for output. Enable this option when outputting to a vector device, such as a pen plotter or a vector photo-plotter.

Server Process Reference

AutoVia	True,False, Toggle	Not implemented.
OnlineDRC	True,False, Toggle	When this parameter is set to true, Altium Designer monitors all PCB design rules interactively as you work and immediately highlight any rule violations. If this option is disabled, design rule violations will not be highlighted as you work.
LoopRemoval	True,False, Toggle	Set this parameter to true to automatically remove any redundant loops that are created during manual routing on the current PCB document. This allows you to re-route a connection without having to manually remove redundant tracks.
RoutingMode	'IGNORE', 'AVOID', 'PUSH', 'CYCLE'	When parameter is set to Ignore, Interactive Router allows the track to completely ignore obstacles while routing. If set to Push, the router moves existing tracks out of the way while routing. If set to Avoid, the Router traces around existing tracks, pads and vias while routing. If set to cycle, the router cycles from a current routing mode to the next mode.
RestrictRoutingTo9045	True,False, Toggle	When the parameter is set to True, the changes of direction during interactive routing are restricted to using 45° or 90° angled tracks. When parameter is set to false, the full range of corner modes is available (any angle, 45°, 45° with arc, 90° and 90° with arc).
DuplicateDesignators	True,False, Toggle	If set to true, duplicate designators are allowed for the current PCB document.
ConfirmDelete	True,False, Toggle	Set the parameter to true to have a confirmation dialog appear before clearing a Selection Memory entry. Selection Memories can be used to store the selection state of a set of objects and are accessed via the *Selection Memory* dialog (open using the button to the left of the Mask Level button, lower right-hand corner of the main design window).
ConfirmGlobalEdit	True,False, Toggle	When the parameter is set to True, the confirmation dialog appears before you can commit or cancel a global editing action. If the parameter is set to False, global editing changes will be made as soon as you click the OK button in a global editing dialog.
ConfirmDragTracks	True,False, Toggle	When the parameter is set to True, the confirmation dialog appears before you can drag tracks.
RotationStep	A real value between 0-and 360 degrees.	The value is in real format. The Default value is 90.000. The Minimum angular resolution is 0.001degrees.
ComponentDrag	None, ConnectedTracks	When the parameter is set to None, and you drag a component, only the component moves. Any attached tracks will be disconnected and left in place. When the parameter is set to ConnectedTracks and you drag a component, any connected tracks will remain attached to the component. This affects how connected tracks are handled when you drag a component.
CursorType	Large90 Small90	If Parameter set to Small90 - Small crosshair cursor like a cross (+). This is the default. If set to Large90 - Cursor like a cross (+) spanning the width of

	Small45	the screen.
		If set to Small45 – a small crosshair cursor and the lines are at a 45° (eg. X).
ShowInvisibleObjects	True,False, Toggle	This parameter corresponds to the Origin Marker. If parameter is set to True, the coordinate origin marker (bottom left corner) is displayed. All objects in the PCB design are positioned relative to the origin marker. Use the Toggle value to toggle from True to False or False to True.
ConvertSpecialStrings	True,False, Toggle	If parameter is set to True, the special strings are converted to their literal values. Special strings act as place holders for various system data, eg. layers names, hole counts and drawing legends. Normally, special strings are interpreted during printing or plotting.
		Use the Toggle value to toggle from True to False or False to True.
HighlightInFull	True,False, Toggle	Set the parameter to true to have selected objects completely highlighted in the current selection color. Set this parameter to false and selected objects are only outlined in the current selection color.
UseNetColorForHighlight	True,False, Toggle	Set the parameter to true and the color is assigned to a particular net for highlighting when you select a net (**Edit » Select » Net in PCB**). Set this parameter to false and a selected net is highlighted in the default selection color.
CleanRedraw	True,False, Toggle	Set this parameter to true, the the document is refreshed with minimal corruption during redrawing but this option is computionally intensive. Turn this option off to use less computing resources.
RedrawLayers	True,False, Toggle	Set this parameter to true to redraw the screen each time you toggle to a different layer, with the current layer being redrawn last. Set this parameter to false and the screen is not redrawn when moving to a different layer.
SingleLayerMode	True,False, Toggle	Set the parameter to True/False the the current single layer mode is cycled. If Gray Scale Other layers were set then Not In Single layer Mode is set and vice versa.
TransparentLayers	True,False, Toggle	Set the parameter to true and when objects are masked, you can see through them to objects on layers underneath the mask.
UseColorDithering	True,False, Toggle	Set the parameter to true and the colors are dithered using least colors. Set it to false, and maximal colors offered by the PC are used.
HoldShiftToSelect_NoObject	True, False, Toggle	If the parameter is false, then all objects to be selected, you need to use the Shfit Click to select. If Parameter is true, you can select design objects nnormally.
HoldShiftToSelect_Arc	True,False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single arc object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Pad	True,False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single pad object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Via	True,False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single via object to select it. Set it to false and

55

Server Process Reference

		you can click on the object to select it.
HoldShiftToSelect_Track	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single track object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Text	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single text object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Fill	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single fill object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Connection	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single connection object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Net	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single net object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Component	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single component object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Poly	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single polygon object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_PolyRegion	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single region object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_ComponentBody	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single component body object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Dimension	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single dimension object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Coordinate	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single coordinate object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Class	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single class object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Rule	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single rule object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_FromTo	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single fromto object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_DifferentialPair	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single differential pair object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Violation	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single violation object to select it. Set it to false and you can click on the object to select it.

HoldShiftToSelect_Embedded	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single embedded object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_EmbeddedBoard	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single embedded board object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_SplitPlane	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single split plane object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Trace	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single trace object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_SpareVia	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single spare via object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_Board	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single board object to select it. Set it to false and you can click on the object to select it.
HoldShiftToSelect_BoardOutline	True, False, Toggle	If the parameter is true, you need to use the SHIFT key as well as clicking on a single board outline object to select it. Set it to false and you can click on the object to select it.

PCB Layer Colors Parameters Table

Each layer color parameter sets the color of a PCB layer. The color value is specified by a RGB value converted from 6 digit hexadecimal number.

For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value:

```
R+256*(G+(256*B)).
```

Parameter	Value	Description
TopSignalColor	Integer	Set the parameter to a RGB integer value to set the color for this top signal layer.
Mid1Color-Mid30Color	Integer	Set the parameter to a RGB integer value to set the color for this middle layer.
BottomSignalColor	Integer	Set the parameter to a RGB integer value to set the color for this bottom signal layer.
TopOverlayColor,BottomOverlayColor	Integer	Set the parameter to a RGB integer value to set the color for this top/bottom overlay layer.
TopPasteColor, BottomPasteColor	Integer	Set the parameter to a RGB integer value to set the color for this top/bottom paste layer.
TopSolderColor, BottomSolderColor	Integer	Set the parameter to a RGB integer value to set the color for this top/bottom solder layer.
Plane1Color – Plane16Color	Integer	Set the parameter to a RGB integer value to set the color for this internal plane layer.
DrillGuideColor	Integer	Set the parameter to a RGB integer value to set the color for this drill guide layer.
KeepOutColor	Integer	Set the parameter to a RGB integer value to set the color for this keep out layer.
Mechanical1Color – Mechanical16Color	Integer	Set the parameter to a RGB integer value to set the color for this mechanical layer.

Server Process Reference

DrillDrawingColor	Integer	Set the parameter to a RGB integer value to set the color for this drill drawing layer.
MultiLayerColor	Integer	Set the parameter to a RGB integer value to set the color for this multi layer.
ConnectLayerColor	Integer	Set the parameter to a RGB integer value to set the color for this connect layer.
SelectionColor	Integer	Set the parameter to a RGB integer value to set the color for this selection layer.
DRCErrorColor	Integer	Set the parameter to a RGB integer value to set the color for this DRC Error layer.
BackgroundColor	Integer	Set the parameter to a RGB integer value to set the color for this background layer.
PadHoleColor	Integer	Set the parameter to a RGB integer value to set the color for this pad hole layer.
ViaHoleColor	Integer	Set the parameter to a RGB integer value to set the color for this via hole layer.
VisibleGrid1Color	Integer	Set the parameter to a RGB integer value to set the color for this visible grid 1 layer.
VisibleGrid2Color	Integer	Set the parameter to a RGB integer value to set the color for this visible grid 2 layer.

PCB Show/Hide Display Table

Parameter	Value	Description
ArcQuality	Full, Draft, Hidden	The parameter sets the display mode for arcs.
FillQuality	Full, Draft, Hidden	The parameter sets the display mode for fills.
PadQuality	Full, Draft, Hidden	The parameter sets the display mode for pads.
PolygonQuality	Full, Draft, Hidden	The parameter sets the display mode for polygons.
DimensionQuality	Full, Draft, Hidden	The parameter sets the display mode for dimensions.
StringQuality	Full, Draft, Hidden	The parameter sets the display mode for strings.
TrackQuality	Full, Draft, Hidden	The parameter sets the display mode for tracks.
ViaQuality	Full, Draft, Hidden	The parameter sets the display mode for vias.
ComponentQuality	Full, Draft, Hidden	The parameter sets the display mode for components.
AllQuality	Full, Draft, Hidden	The parameter sets the display mode for All primitives
DraftTrackThreshold	Real	This parameter shows the current minimum track width that determines whether or not tracks are displayed as single lines or outlines in Draft display mode. Tracks of the defined width, or narrower, will be displayed as a single line. Tracks of greater width will be displayed as an outline when displayed in Draft Mode. Specify a value in Real type for the Parameter to change the width. The width is based on the current unit used by the PCB Editor. Default units (metric or imperial) are determined by the **Measurement Unit** setting in the *Board Options* dialog (**Design » Board Options**). When DirectX is being used this DraftTrackThreshold is ignored.
DraftStringThreshold	Integer	Set the parameter to the current minimum string height (in screen pixels) that determines whether or not the text on the document is displayed in full or as an outline box only. Strings equal to or greater than the set number of pixels in height at the current zoom level will be displayed as text. Otherwise the text is replaced by an outline box. The default is 4 pixels. When DirectX is being used this DraftStringThreshold is ignored.

ShowPadNumbers	True, False, Toggle	Set the parameter to true to display Pad Numbers for pad objects.
ShowPadNets	True, False, Toggle	Set the parameter to true to display Nets for pad objects.
ShowViaNets	True, False, Toggle	Set the parameter to true to display Nets for via objects.
ShowTextPoints	True, False, Toggle	Set the parameter to true to display Test Points on the PCB document. Notice the spelling "Text". ShowTextPoints parameter corresponds to the test points.
ShowStatusInfo	True, False, Toggle	Set the parameter to true and the status information for the current PCB object is displayed on the status bar of Altium Designer. The information displayed includes the location of this object on a PCB document, the layer it is on and the net it is connected to (showing the width and length of this net as well).
ShowComponentRefPoint	True, False, Toggle	Set the parameter to true to display the component reference point markers for components.
ShowComponentBodies	True, False, Toggle	Set the parameter to true to display extruded 3D bodies whenever the view configuration Show Simple 3D Bodies setting is set to Use System Settings.
ShowComponentStepModels	True, False, Toggle	Set the parameter to true to display 3D STEP models whenever the view configuration Show STEP Models setting is set to Use System Settings.
ShowComponentSnapMarkers	True, False, Toggle	Set the parameter to true to display Snap Markers for components when in 3D mode.
ShowBoardCore	True, False, Toggle	Set the parameter to true to display the board core when in 3D mode.
ShowBoardPrepreg	True, False, Toggle	Set the parameter to true to display the board prepreg when in 3D mode
ShowTopSilkScreen	True, False, Toggle	Set the parameter to true to display the top silk screen when in 3D mode
ShowBotSilkScreen	True, False, Toggle	Set the parameter to true to display the bottom silk screen when in 3D mode
PlaneDrawMode	0,1	If PlaneDrawMode parameter set to 0, outlined Layer Colored. If parameter is set to 1, Solid Net Colored.
DisplayNetNamesOnTracks	0,1,2	If DisplayNetNamesOnTracks parameter set to 0, Netnames are not displayed on tracks. If 1, Net names are Single And Centered and displayed on tracks. If 2, Net names are repeated on tracks.
OriginMarkerColor	Integer	Specifies the Origin Marker color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
ComponentRefPointColor	Integer	Specifies the Component Reference Point color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).

Server Process Reference

		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
PositiveTopSolderMask	True, False, Toggle	Set the PositiveTopSolderMask to true to display the top solder mask in positive. If False, this solder mask is in negative.
PositiveBottomSolderMask	True, False, Toggle	Set the PositiveBottomSolderMask to true to display the bottom solder mask in positive. If False, this solder mask is in negative.
FromTosDisplayMode	0,1,2	Set the parameter to 0, and the FromToDisplay Mode is set to automatic. If 1, the From To Display mode is hidden. If 2, the From To Display mode is shown.
PadTypesDisplayMode	0,1,2	Set the parameter to 0, and the Pad Types Display Mode is set to automatic. If 1, the Pad Types Display mode is hidden. If 2, the Pad Types Display mode is shown.
BoardThicknessScaling	Double	Define the board thickness in scale (0 to 100).
WorkspaceLuminanceVariation	Integer	Specifies the Workspace Luminance Variation color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
WorkspaceColor3D	Integer	Specifies the Workspace Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
BoardCoreColor3D	Integer	Specifies the Board Core Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
BoardPrepregColor3D	Integer	Specifies the Board Prepreg Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
TopSolderMaskColor3D	Integer	Specifies the Top Solder Mask 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
BotSolderMaskColor3D	Integer	Specifies the Bottom Solder Mask Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula

		may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
CopperColor3D	Integer	Specifies the Copper Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
TopSilkScreenColor3D	Integer	Specifies the Top Silk Screen Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
BotSilkScreenColor3D	Integer	Specifies the Bottom Silk Screen Color 3D color with the RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
WorkspaceColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Workspace Color 3D.
BoardCoreColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Board Core Color 3D.
BoardPrepregColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Board Prepreg Color 3D.
TopSolderMaskColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Top Solder Mask Color 3D.
BotSolderMaskColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Bottom Solder Mask Color 3D.
CopperColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Copper Color 3D.
TopSilkScreenColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Top Silk Screen Color 3D.
BotSilkScreenColor3D_Opacity	Single	Set opacity value between 0 and 1 to set the Bottom Silk Screen Color 3D.

Note the Single type is the fastest floating point type in Delphi. It also has the lowest storage requirements - 32 bits (1 for the sign 8 for the exponent, and 23 for the mantissa). It supports approximately 7 digits of precision in a range from 1.18×10^{-38} to 3.4×10^{38}.

Example

Process: PCB:SetupPreferences

Parameters : ShowComponentBodies = True

Example with DelphiScript

```
Procedure RefreshDisplay;
Begin
    ResetParameters;
    AddStringParameter('Action','Redraw');
    RunProcess('PCB:Zoom');
```

Server Process Reference

```
End;
{.........................................................................}
Procedure ShowComponentBodies(Enable : Boolean);
Begin
    ResetParameters;
    If Enable Then
        AddStringParameter('ShowComponentBodies','True')
    Else
        AddStringParameter('ShowComponentBodies','False');
    RunProcess ('Pcb:SetupPreferences');
End;
{.........................................................................}
Procedure TurnOnComponentBodiesIn3d;
Begin
    ShowComponentBodies(True);
    RefreshDisplay;
End;
{.........................................................................}
Procedure TurnOffComponentBodiesIn3d;
Begin
    ShowComponentBodies(False);
    RefreshDisplay;
End;
```

ShowApplicableRules process

Description

The ShowApplicableRules process is used to show which rules are applicable to this object if no parameter specified. If Binary = True, you are prompted to select two objects that the binary rules are applicable to. A report dialog is displayed showing which rules are applied to one object (or two objects).

Parameters

Parameter	Value	Description
Binary	True, False	Defaults to unary rule if no parameter is specified. If True, you are prompted to select two objects.

Example

Process: PCB : ShowApplicableRules

Parameters : Binary = True

ShoveComponents process

Description

Shove components and move the surrounding objects on the current PCB.

Parameters

Parameter	Value	Description
Method	SetShoveDepth	If Method = SetShoveDepth, you will be prompted with a Shove Depth dialog.

Example

Process: PCB:ShoveComponents

Parameters : Method = SetShoveDepth

ShowConnections process

Description

Make connections visible.

Parameters

Parameter	Value	Description
Show	Net, ComponentNets, All	If Net, then the specified net is shown only and you will be prompted to choose which net. If All, all connections are shown. If ComponentNets, only nets to components will be shown.

Example

Process: PCB:ShowConnections

Parameters: Show=Net

See also

HideConnections process

SnapGrid process

Description

Define the snap grid x and y simultaneously for the PCB document.

Parameters

Parameter	Value	Description
Size	1Mil, 5Mil, 10Mil, 20MIl, 25Mil, 50MIl, 100Mil, 0.025MM, 0.100MM, 0.250MM, 0.500MM, 1.000MM, 2.500MM	If Size not specified, you are prompted to define a snap grid.

Example

Process: PCB:SnapGrid

Parameters : Size = 5Mil

SeeAlso

SnapGridXY process

SnapGridXY

Description

Define the snap grid for the x axis or y axis for the current PCB document.

Parameters

Parameter	Value	Description
Axis	X,Y	When Axis = X or Y, you are required to also specify a value in current Units for the X or Y. If the Value is not specified for either X or Y, you will be prompted to enter a value for either X or Y.
Value	String	Possible values for the X or Y axis are: 1Mil, 5Mil, 10Mil, 20MIl, 25Mil, 50MIl, 100Mil, 0.025MM, 0.100MM, 0.250MM, 0.500MM, 1.000MM, 2.500MM. If Value not specified, you are prompted to define a snap grid.

Example

Process: PCB:SnapGridXY

Server Process Reference

Parameters : Axis = X | Value = 100Mil

See Also

SnapGrid process

TearDropSelectedPads process

Description

Add tear drops to selected pads on the PCB board for better electrical properties.

Parameters

N/A

Example

Process: PCB:TearDropSelectedPads

ToggleSelection process

Description

The ToggleSelection process toggles the selection state of PCB objects.

Parameters

Parameter	Value	Description
Object	Arc, Component, Fill, Pad, Track, Via, String, Coordinate, Dimension, Polygon, Net	The process prompts for a object to be selected depending on which object is targeted. If Object is not specified, the process selects any object you click on. If the **Click Clears Selection** option is enabled in the PCB Editor – General page then it clears previous seletions first.

Example

Process: PCB:ToggleSelection

Parameters : Object=Track

Undo process

Description

Undoes the current operation.

Parameters

N/A

Example

Process: PCB:Undo

Unroute process

Description

Unroutes all nets, a specific net, room or a component on a current PCB document.

Parameters

Parameter	Value	Description
Object	All, Net, Room, SingleRoom, Connection, Component, SingleComponent	When Object = SingleRoom or SingleComponent, you need to specify the value for the ContextObject (Room or Component respectively)
ContextObject	Room, Component	Specify the ContextObject if you wish to specify the Object parameter as a single room or single component.

Example

Process: PCB:Unroute

Parameters : Object=All

UpdateFootprints process

Description

Updates footprints on a PCBfrom a library based on the PCB components in a current PCB document.

Parameters

Parameter	Value	Description
Mode	All	If mode is set to All, all PCB footprints from the PCB library are updated. If mode is set to empty, the current PCB footprint from the PCB library is updated.

Example

Process: PCB : UpdateFootprints

Parameters : Mode = All

UpdateRotationOnSelectedComponents process

Description

Used to update selected components rotation field values, based upon the orientation of the component in the library and the pad positions in the placed component. Otherwise, components are assumed to have a 0 rotation value, whatever their placed orientation.

Zoom process

Description

The Zoom process is used to set the zoom level of the current PCB document. Depending upon the parameters, a number of zoom actions can be performed from refreshing the screen to displaying a specified region of the PCB document.

Parameters

Parameter	Value	Description
ZoomLevel	Real	Prompts for a zoom value if not specified.
Action	In, Out, All, Filtered, Board, Last, MicroIn, MicroOut, Pan, Point, Redraw, RedrawCurrent,Selected, Sheet, Window	If Action is set to area, then the four following parameters will be used (Location1.x, Location1.Y, Location2.X and Location2.Y)

Example

Process: PCB:Zoom

Parameters : ZoomLevel = 4.0

Common Example used in DelphiScript

```
Procedure RefreshDisplay;
Begin
    ResetParameters;
    AddStringParameter('Action','Redraw');
    RunProcess('PCB:Zoom');
End;
```

Schematic Processes

This section covers the Schematic processes and their parameters (if any).

Table of Schematic processes

AlignObjects process	LibraryDocumentDescription process	PlaceSheetEntry process
AskForXYLocation process	ListAllSelectedPins process	PlaceSheetSymbol process
BringObjectToFront process	MoveComponentToLibrary process	PlaceStimulus process
BringObjectToFrontOf process	MoveCursor process	PlaceTestVectorIndex process
ChangeComponentName process	MoveObject process	PlaceTextFrame process
ChangeCurrentTemplate process	MoveObjectToFront process	PlaceWire process
ChangeObject process	MoveSelectedObjects process	PreviousComponentLibraryEditor process
ChangeObjectGraphically process	MoveSingleObject process	PreviousComponentPart process
ChangeObjectGraphicallyOrSetFocus process	NextComponentLibraryEditor process	PrintDocument process
ChangeSingleObject process	NextComponentPart process	Redo process
Clear process	Paste process	RemoveComponentPart process
ClearLocationMark process	PlaceAnnotation process	RemoveDuplicateComponentNames process
ComponentRuleCheck process	PlaceArc process	RemoveTemplate process
Copy process	PlaceArray process	RenameObjectText process
CopyComponentToLibrary process	PlaceBezier process	ReportComponent process
CreateComponent process	PlaceBus process	ReportComponentLibrary process
CreateLibraryFromProject process	PlaceBusEntry process	ResetUniqueIds process
CreateSheetFromFGPAPart process	PlaceComponentFromLibraryEditor process	Select process
CreateSheetFromSheetSymbol process	PlaceEllipse process	SelectionMemory process
CreatSheetSymbolFromSheet process	PlaceEllipticalArc process	SendObjectToBack process
CrossProbeChoose process	PlaceGraphicImage process	SendObjectToBackOf process
CrossProbeNetOnSch process	PlaceIEEESymbol process	SetLocationMark process
CrossProbeNotify process	PlaceIntegratedComponent process	SetupArrayPlacement process
CrossReference process	PlaceJunction process	SetupPreferences process
Cut process	PlaceLine process	SetupPrinter process
DeleteComponentFromLibrary process	PlaceNetLabel process	SynchronizeHierarchy process
DeleteObjects process	PlaceNoErc process	ToggleComponentModeDisplay process
Deselect process	PlaceNote process	ToggleElectricalGrid process
DocumentPreferences process	PlaceOffSheetConnector process	ToggleHiddenPins process
DownHierarchy process	PlaceParameterSet process	ToggleSelection process
Drag process	PlacePart process	ToggleSingleObjectSelection process
ExportLibraryToDatabase process	PlacePartFromLibraryEditor process	ToggleSnapGrid process
ExportSchematicToDatabase process	PlacePartFromSchEditor process	ToggleVisibleGrid process
FilterSelect process	PlacePCBLayoutDirective process	Undo process
FindAndReplaceText process	PlacePieChart process	UpdateComponentsFromLibraryEditor process
FindNextText process	PlacePin process	UpdateCurrentTemplate process
FindText process	PlacePolygon process	UpdateLibraryMask process
FirstComponentLibraryEditor process	PlacePort process	UpdatePartDatabaseLinks process

IncrementComponentPartNumber process	PlacePowerPort process	UpdatePartFromLibraryEditor process
Jump process	PlaceProbe process	UpdatePartsFromLibraryList process
LastComponentLibraryEditor process	PlaceRectangle process	Zoom process
	PlaceRoundedRectangle process	

Server Process Reference

AddComponentPart process

Description

Adds a new component part to a component in the library.

Parameters

N/A

Example

Process: PCB:AddComponentPart

AlignObjects process

Description

The AlignComponents process aligns selected objects on a Schematic document using specified parameters. There are different alignment parameters.

Parameters

Parameter	Value	Description
Action	Dialog,Left, Right, CenterHorizontal, SpaceEquallyHorizontal,Top , Bottom, CenterVertical, SpaceEquallyVertical,Grid	If no parameters supplied, the Align Components dialog appears.

Example

Process: SCH:AlignComponents

Parameters : Alignment = CenterHorizontal

AskForXYLocation process

Description

Returns X and Y location values from a cursor click on a current schematic document. This is primarily used for scripts that need (X,Y) coordinates of the mouse click on the document in Altium Designer using Delphi Script for example.

Parameters

Parameter	Value	Description
Location.X	Integer	
Location.Y	Integer	
Result	Integer	

Example

Process: SCH:AskForXYLocation

Example 2

```
Function GetClickPosition;
Var
    X,Y : Integer;
    Result1, Result2 : Integer;
Begin
    ResetParameters;
    RunProcess('SCH:AskForXYLocation');
    Result1 := GetIntegerParameter('Location.X',X);
    Result2 := GetIntegerParameter('Location.Y',Y);
```

```
End;
```

BringObjectToFront process

Description

Bring a selected object graphically to the front of all other objects on a schematic document.

Parameters

N/A

Example

Process: SCH:BringObjectToFront

BringObjectToFrontOf process

Description

Bring an object to the front of another object on a schematic document.

Parameters

N/A

Example

Process: SCH:BringObjectToFrontOf

ChangeComponentName process

Description

Change the name of the current component in the current library.

Parameters

Parameter	Value	Description
Name	String	Name of the component to change.
Returns	True, False	Depending on whether the component name has changed or not.

Example

Process: SCH:ChangeComponentName

ChangeCurrentTemplate process

Description

The ChangeCurrentTemplate process is used to change the current template of a sheet to a new template file name. A template is a special graphical entity that holds user-defined sheet size, border and title block descriptions.

Parameters

Parameter	Value	Description
FileName	String	Specifies the full path and file name of the template file.

Example

Process: SCH:ChangeCurrentTemplate

ChangeObject process

Description

Select and use dialog to change objects on the schematic document if no parameters supplied. Otherwise you can change component parts to sheet symbols or to ports or break a polyline into segments if the Action parameter is specified.

Parameters

Parameter	Value	Description
Action	PartToSheetSymbol,,	

Server Process Reference

	PartToPorts, BreakPolyline	

Example

Process: SCH:ChangeObject

Parameters :

ChangeObjectGraphically process

Description

The ChangeObjectGraphically process is used to graphically change or move an object. If the object already has the focus then this process interactively changes the object by physically moving the position of the objects handles.

Parameters

N/A

ChangeObjectGraphicallyOrSetFocus process

Description

The ChangeObjectGraphicallyOrSetFocus process is used to set the focus on an object. If the object already has the focus then this process interactively changes the object by physically moving the position of the objects handles.

Parameters

Parameter	Value	Description
ZoomOnFocusedObject	True, False	If True then the focussed object is zoomed into, resizing the whole object on the screen.

Example

Process: SCH:ChangeObjectGraphicallyOrSetFocus

Parameters : ZoomOnFocusedObject = True

ChangeSingleObject process

Description

The ChangeSingleObject process is used to modify specific attributes of placed objects on the schematic or library document. This process is similar to the Change process, except that when an object has been edited you will not be prompted to select another object.

Parameters

Parameter	Value	Description
RunComponentDialog	True,False	Defaults to false. If true, Component Properties dialog appears.

Clear process

Description

Delete all selected objects from the current schematic document.

Parameters

N/A

ClearLocationMark process

Description

The ClearLocationMark process is used to clear a specific location marker. To set a new location use one of the SetLocationMark (1-10) processes. To move the cursor to a marked location use a Jump process with a specified LocationMark parameter.

Parameters

Parameter	Value	Description
LocationMark	1..10	To clear all location marks, leave the LocationMark parameter blank.

Example

Process: SCH:ClearLocationMark

Parameters : LocationMark = 1

ComponentRuleCheck process

Description

The ComponentRuleCheck process searches the current active library in the Library Editor for user-specified errors such as duplicate pin numbers or missing footprints.

Parameters

Parameter	Value	Description
Show	True,False	If True, the report of pin errors are displayed. Defaults to True.
DuplicateComponentNames	True,False	Checks for duplicate component names. Defaults to previous setting if none specified.
DuplicatePins	True,False	Checks for duplicate pins. Defaults to previous setting if none specified.
MissingDescription	True,False	Checks for empty description field. Defaults to previous setting if none specified.
MissingFootPrint	True,False	Checks for empty foot print field. Defaults to previous setting if none specified.
MissingDefaultDesignator	True,False	Checks for empty default designator field. Defaults to previous setting if none specified.
MissingPinName	True,False	Checks for empty pin name field. Defaults to previous setting if none specified.
MissingPinNumber	True,False	Checks for empty pin number field. Defaults to previous setting if none specified.
Returns	True,False	'Result=True' or 'Result=False' depending on whether the dialog or the parameters for this process has been processed or not.

Example

Process: SCH:ComponentRuleCheck

Parameters: DuplicateComponentNames=True | MissingPinName=False

Copy process

Description

The Copy process is used to copy the objects that are currently selected in the schematic or library document to the clipboard. The Paste process can be used to place a copy of the selection back into any open the Schematic editor document window or into any application which supports the .WMF (Windows MetaFile) clipboard format.

Parameters

Parameter	Value	Description
ClipboardMode	ClipboardRing,''	When you wish to do a general copy, assign ClipBoardMode to ClipboardRing. If you wish to copy as text, ClipboardMode=ClipboardRing|Action=CopyAsText
Action	CopyAsText, ''	If you wish to copy as text, you will need to assign values to 2 parameters; ClipboardMode=ClipboardRing|Action=CopyAsText

Server Process Reference

Example

Process: SCH:Copy

Parameters : ClipBoardMode = ClipboardRing | Action = CopyAsText

CopyComponentToLibrary process

Description

The CopyComponentToLibrary process is used in the Library Editor to copy a component from the current library file to another library file. This process is useful when creating customized libraries.

Parameters

N/A

CreateComponent process

Description

The CreateComponent process is used to create a new component within the current library document.

Parameters

Parameter	Value	Description
Name	String	Name of the component to create.

Example

Process: SCH:CreateComponent

Parameters : Name = ExprCapacitor

CreateLibraryFromProject process

Description

Creates a library document of symbols based on the components on a current schematic document.

Parameters

N/A

CreateSheetFromFPGAPart process

Description

Create a VHDL or Verilog file from a FPGA component.

Parameters

Parameter	Value	Description
DocumentKind	VHDL,Verilog	DocumentKind is set to one of the Hard Description Languages (VHDL or Verilog) based on which FPGA part.

CreateSheetFromSheetSymbol process

Description

Creates a new schematic document and adds ports for the sheet entries that are on the sheet symbol. Used in hierarchical designs. After creating a sheet symbol (if following a top down design methodology), automatically creates a new sheet with ports for all sheet entries present in the sheet symbol. Ports electrical characteristics and styles complement the sheet entries in the original sheet symbol.

Parameters

Parameter	Value	Description
DocumentKind	Schematic VHDL,Verilog, C	Specify the type of sheet when creating a new sheet. Defaults to a schematic sheet if no parameter supplied.

Example

Process: SCH:CreateSheetFromSheetSymbol

Parameters : Type = Schematic

See also

CreateSheetSymbolFromSheet process

CreateSheetSymbolFromSheet process

Description

Create a sheet symbol that represents the current schematic document. Used in hierarchical designs, to speed creation of sheet symbols for a "child" sheet. After creating a sheet, this process will automatically generate a sheet symbol, labeled with its file name, including Sheet Entries for each Port in the sheet. Sheet Entry electrical characteristics and styles complement the ports in the original sheet.

Parameters

N/A

See also

CreateSheetFromSheetSymbol process

CrossProbeChoose process

Description

The ChooseCrossProbe process is defined to allow a user to cross-probe from the Schematic editor server into the other servers that are currently running within the Altium Designer environment

Parameters

Parameter	Value	Description
Action	CrossSelect	

CrossReference process

Description

The CrossReference process adds port references to a document or a project, or removes port references from a document or a project.

Parameters

Parameter	Value	Description
Action	AddToDocument, AddToProject, RemoveFromDocument, RemoveFromProject	

Example

Process: SCH:CrossReference

Parameters : Action = AddToDocument

Cut process

Description

The Cut process is used to clear the current selected objects from the document and copies it to the clipboard. The Paste process can be used to place the selection back into any open the Schematic document window or to another application that supports the Windows .WMF clipboard format.

Parameters

N/A

Example

Server Process Reference

Process: SCH:Cut

DeleteComponentFromLibrary process

Description

The DeleteComponentFromLibrary process is used to remove components from the current Library document.

Parameters

Parameter	Value	Description
Name	String	Name of component to delete from current library.

Example

Process: SCH:DeleteComponentFromLibrary

Parameters : Name = 74LS00

DeleteObjects process

Description

The DeleteObjects process is used to delete objects from the schematic and library editor document windows.

Parameters

N/A

DeSelect process

Description

DeSelect objects inside/outside an area, all objects on a current schematic document or all open schematic documents

Parameters

Parameter	Value	Description
Action	InsideArea,OutsideArea,All, AllOpenDocuments	DeSelect objects inside/outside an area, all objects on a current schematic document or all open schematic documents

Example

Process: SCH:DeSelect

Parameters : Action= All

See also

DeSelect process.

DocumentPreferences process

Description

The DocumentPreferences process is used to define various document settings for either the schematic editor or library editor, such as sheet styles, size, orientation, background and border colors and other options that apply to the document. If no parameters are supplied, the *Document Options* dialog appears.

Parameters

Parameter	Value	Description
Tab	Sheet Options, Parameters, Units	If value is Sheet Options, the Sheet Options tab of the Document Options dialog appears. If Parameters, the Parameters tab appears allowing you to update existing Special String or add custom parameters. If Units, then the Units tab appears allowing you to choose which Unit System.
Action	SetSnapGrid, SetVisibleGrid,SetElectricalGrid ,EditProperties	A dialog appears allowing you to enter a value based on the current Unit System used by Schematic document depending on which grid you have specified. If Action is set to EditProperties, the Document Options dialog appears.

Example

Process: SCH:DocumentPreferences

Parameters : Tab = SheetOptions

DownHierarchy process

Description

Go down the hierarchy of a project.

Parameters

Parameter	Value	Description
Action	OpenAllDocumentsInHierarchy	If no parameter supplied, you are prompted with a cross hair to click on a sheet symbol or port to go up or down from a master sheet to a child sheet and vice versa.

Example

Process: SCH:DownHierarchy

Parameters : Action = OpenAllDocumentsinHierarchy

Drag process

Description

The DragObject process is used to move an object, selected objects or an object using focus rather than selection. If the object or group of objects has electrical characteristics such as a wire or part then connectivity will be maintained with other attached electrical objects.

Parameters

Parameter	Value	Description
Action	SingleObject, MultipleObjects, SelectedObjects	If Action is set to Single Object, the current object can be dragged. If MultipleObjects, then multiple objects can be dragged. if SelectedObjects then selected objects can be dragged.

Example

Process: SCH:Drag

Parameters : Action = SelectedObjects

ExportLibraryToDatabase process

Description

Not Implemented.

Parameters

N/A

ExportSchematicToDatabase process

Description

Not implemented.

Parameters

N/A

FilterSelect process

Description

Use this FilterSelect process to perform one of the many filtering processes including displaying the Find Similar Objects dialog on a schematic document.

Server Process Reference

Parameters

Parameter	Value	Description
Action	FindSimilar,FindSimilarUnderCursor, ShowFavorites	To invoke the Find Similar Objects dialog, use Action=FIndSimilar.
Clear	True, False	Set Clear to true to clear out the filter. Set it false to keep previous filter settings.
ClearUnderlines	True, False	Set ClearUnderlines to true to remove the underlines.
Expr	String	Expr is a valid expression string for the Filter. For example Expr = (ParameterName = ''Comment'') And (IsHidden = ''True'') looks for hidden parameters with comment names.
Zoom	True, False	When Zoom is set to true, those Schematic design objects passing through the filter will be zoomed into as a whole.
Mask	True, False	When Mask is set to true, those schematic design objects passing through the filter will not be masked.
Select	True,False	When Select is set to True, those Schematic design objects passing through the filter will get selected.

Examples

Process: SCH:FilterSelect

Parameters : Expr=IsPin And (PinElectrical = ''Power'') | Zoom=True | Mask = True | Select=True

Parameters : Expr=IsSchComponent And (PinsLocked = ''False'')

FindAndReplaceText process

Description

The FindAndReplaceText process searches for and replace text strings on a schematic document. If no parameters supplied, the Find and Replace Text dialog appears.

Parameters

Parameter	Value	Description
SearchText	String	Specifies the text to search for on the schematic document. Need the ReplaceText parameter.
ReplaceText	String	Specifies the text to replace with on the schematic document. Need the SearchText parameter.
DocumentScope	0..1	Specifies the sheet scope to search in. 0 - All Document, 1 - Current Document.
CaseSensitive	True, False	Depending on whether the search is case sensitive or not.
PromptOnReplace	True, False	Depending on whether to prompt for confirmation before replacing text.
RestrictToNet	True, False	Depending on whether to restrict search to net identifiers.
SelectionCriteria	0..2	Specifies objects to search according to their selection state. 0 - Selected Objects, 1 - Deselected Objects, 2 - All Objects

Example

Process: SCH:FindAndReplaceText

FindNextText process

Description

Search for and jump to the next matching text string which is the next occurence of the last text find that was specified using the FindText process.

Parameters

N/A

FindText process

Description

Search for and jump to a text string on a schematic document. If no parameters are supplied, the Find Text dialog appears prompting you to define the search criteria.

Parameters

Parameter	Value	Description
SearchText	String	Specifies the text to search for
DocumentScope	0..1	Specifies the sheet scope to search in. 0 - All Document, 1 - Current Document.
CaseSensitive	True, False	If true, search looks for case sensitive text only. Dog and DOG are different in this case. If false, search looks for text irrespective of case. Dog and DOG are same in this case.
PromptOnReplace	True, False	If true, you are prompted before you proceed to replace found text.
RestrictToNet	True, False	If true, you are restricted search to net identifiers.
SelectionCriteria	0..2	Specifies objects to search according to their selection state. 0 - Selected Objects, 1 - Deselected Objects, 2 - All Objects.

Example

Process: SCH:FindText

FirstComponentLibraryEditor process

Description

The FirstComponentLibraryEditor process is used to go to the first component in a current library document in the Library Editor.

Parameters

N/A

Example

Process: SCH:FirstComponentLibraryEditor

IncrementComponentPartNumber process

Description

Increments the component part number for a multi-part component.

Parameters

N/A

Jump process

Description

This parameter can be used to do a variety of jump actions such as jumping to the next error marker, origin, or a new location etc.

Parameters

Parameter	Value	Description
NextErrorMarker	True, False	If true, jump to the next existing error marker.
Origin	True, False	If true, jump to the origin of the schematic document
NewLocation	True, False	If true, jump to a new location
LocationMark	1..10	Jump to a location mark

Server Process Reference

Example

Process: SCH:Jump

Parameters : NewLocation = True

LastComponentLibraryEditor process

Description

The LastComponentLibraryEditor process is used to go to the last component in a current library document.

Parameters

N/A

LibraryDocumentDescription process

Description

The AlignComponents process aligns selected objects on a PCB document using specified parameters. There are different alignment parameters.

Parameters

Parameter	Value	Description
Alignment	MoveComponentsToGrid, MoveRoomsToGrid, Bottom, Left, Right, Top, CenterHorizontal, CenterVertical, ExpandHorizontal, ExpandVertical, ContractHorizontal, ContractVertical, SpreadHorizontal, SpreadVertical	If no parameters supplied, the Align Components dialog appears.

Example

Process: Sch:AlignComponents

Parameters : Alignment = CenterHorizontal

ListAllSelectedPins process

Description

The ListAllSelectedPins process is used to display a list of the part and pin designators currently selected on the worksheet. This process is useful to identify that your nets contain all the necessary pin information.

Parameters

N/A

MoveComponentToLibrary process

Description

The MoveComponentToLibrary process is used in the Library Editor to move a component from the current library file to another library file. This is similar to the CopyComponentToLibrary process, except that the component is physically removed from the current library and placed in the destination library. This process is useful when creating customized libraries.

Parameters

N/A

MoveCursor process

Description

Move the cursor across the screen by a specified unit.

Parameters

Parameter	Value	Description
Direction	Right, Left, Up, Down	Move cursor in the specific direction specified by the Direction parameter.
BigSteps	True, False	

Example

Process: SCH:MoveCursor

Parameters : Direction = Up | BigSteps = True

MoveObject process

Description

The MoveObject process is used to interactively change the location of placed objects on the schematic and library editor documents.

Parameters

N/A

See also

MoveSelectedObjects process

MoveObjectToFront process

MoveObjectToFront processs

Description

The MoveObjectToFront process is used to interactively change the location of placed objects on the schematic and library documents. This process is similar to the MoveObject process, except that when an object has been moved and placed on the document it will be positioned in front of all the other stacked objects..

Parameters

N/A

MoveSelectedObjects process

Description

The MoveSelectedObjects process is used to reposition an individual selected object or a complex selection containing many objects as a single entity on a current schematic library/document.

Parameters

N/A

MoveSingleObject process

Description

The MoveSingleObject process is used to interactively change the location of placed objects on the schematic and library editor document windows. This process is similar to the MoveObject process, except that when an object has been moved you will not be prompted to select another object.

Parameters

N/A

NextComponentLibraryEditor process

Description

The NextComponentLibraryEditor process is used to go to the next component in a current library document in the Library Editor.

Parameters

N/A

Example

Server Process Reference

Process: SCH:NextComponentLibraryEditor

NextComponentPart process

Description

The NextComponentPart process is used to show the next part of a multiple part component in the current library document.

Parameters

N/A

Paste process

Description

The Paste process is used to place the current clipboard contents into an open Schematic / Library document. However it can be configured as a rubber stamp, placing multiple copies of the same obejct on the schematic document.

Parameters

Parameter	Value	Description
Action	RubberStamp	

Example

Process: SCH:Paste

Parameters: Action=RubberStamp

PlaceAnnotation process

Description

Place a single line text on the current document.

Parameters

Parameter	Value	Description
S	String	Text string up to 255 characters in length. Example: S=.date specifies the special string for displaying the current date.
Location.X	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Location.Y	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
FontID	String	The font is specified by a description string. The string contains seven fields each separated by a space. "<Size> <Rotation> <Underline> <Italic> <Bold> <StrikeOut> <FontName>". Size is an integer; Rotation cannot be set, always 0; Underline, Italic, Bold and Strikeout are boolean flags 0=False, 1=True; FontName is the actual font name. Examples: FontID=8 0 0 0 1 0 Helv - produces a Helvetica bold font with 8pt size. FontID=12 0 0 1 0 0 Times New Roman - produces a Times New Roman bold, italic font with 12pt size.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceAnnotation

PlaceArc process

Description

Place graphical arcs on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units of .01 inch. Range is 0-6500. Example: Location.X=100
Location.Y	Integer	Specified in coordinate units of .01 inch. Range is 0-6500. Example: Location.Y=1500
Radius	Integer	Specified in units of .01 inch. Example: Radius=30
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
StartAngle	Real	Start angle can be any degree from 0 to 360. Example: StartAngle=33.333
EndAngle	Real	End angle can be any degree from 0 to 360. Example: EndAngle=100
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceArc

PlaceArray process

Description

Place an array with the last set array placement options.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units Example: Location.X=100
Location.Y	Integer	Specified in coordinate units Example: Location.Y=100

Example

Process: SCH:PlaceArray

Parameters : Location.X = 5000 | Location.Y = 5000

PlaceBezier process

Description

Place a bezier curve on the current document.

Parameters

Parameter	Value	Description
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.

Server Process Reference

Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).
		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceBezier

PlaceBus process

Description

Place a bus on the current schematic document.

Parameters

Parameter	Value	Description
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).
		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceBus

PlaceBusEntry process

Description

Place a bus entry on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).

		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceBusEntry

PlaceComponentFromLibraryEditor process

Description

Places a component part from library editor into the current schematic document.

Parameters

Parameter	Value	Description
LibraryName	String	Text string up to 255 characters in length. Specifies the full path and file name of the schematic library document.
PartType	String	Text string up to 255 characters in length. Specifies the component part name.

Example

Process: SCH:PlaceComponentFromLibraryEditor

PlaceEllipse process

Description

Place an elliptical shape on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
Radius	Integer	Specified in units Example: Radius=30
Secondary Radius	Integer	Specified in units Example: SecondaryRadius=30
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
IsSolid	True,False ,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
Selection	True,False	This parameter has three possible selection states for the object. True=Selected, False= Not

Server Process Reference

	,Toggle	selected and Toggle=switch selection state based on previous setting.
Transparent	True,False	

Example

Process: SCH:PlaceEllipse

PlaceEllipticalArc process

Description

Place an elliptical arc on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
Radius	Integer	Specified in units Example: Radius=30
Secondary Radius	Integer	Specified in units Example: SecondaryRadius=30
StartAngle	Real	Start angle can be any degree from 0 to 360. Example: StartAngle=33.333
EndAngle	Real	End angle can be any degree from 0 to 360. Example: EndAngle=100
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceEllipticalArc

PlaceGraphicImage process

Description

Place a graphical image on the current schematic document.

Parameters

Parameter	Value	Description
FileName	String	The full path and file name of the image file to be opened. Example: FileName=C:\MyBitmap.bmp
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units

Radius	Integer	Specified in units Example: Radius=30
Secondary Radius	Integer	Specified in units Example: SecondaryRadius=30
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
IsSolid	True,False ,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
KeepAspect	True,False ,Toggle	Set KeepAspect to true to maintain the aspect of the image.

Example

Process: SCH:PlaceGraphicImage

PlaceIEEESymbol process

Description

Places an IEEE symbol on a library schematic sheet.

Parameters

Symbol (Dot, RightLeftSignalFlow, CLock, ActiveLowInput, AnalogSignalIn, NotLogicConnection, PostponedOutput, OpenColelctor, Hiz, HighCurrent, Pulse, Delay, GroupLine, GroupBinary, ActiveLowOutput, PiSymbol, GreaterEqual, OpenCollectorPullup, OpenEmitter, OpenEmitterPullUp, DigitalSignalIn, Invertor, OrGate, InputOutput, AndGate, XorGate, ShiftLeft, LessEqual, Sigma, Schmitt, ShiftRight).

Example

Process: SCH:PlaceIEEESymbol

Parameters : Symbol = Schmitt

PlaceIntegratedComponent process

Description

The PlaceIntegratedComponent process places a component from the integrated library on a schematic sheet only.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value,

Server Process Reference

		R+256*(G+(256*B)).
		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
LibReference	String	Name of the part to be placed
Library	String	Component library filename where the part exists. By default if no library is specified the component cache is used, if the part is still not found then all .lib files are searched in the current working directory.
Designator	String	Specify the designator of the component.
SourceLibraryName	String	Specify the Source library name of the component.
PartID		Specifiy the part ID of the component.
ParameterNameX	String	Specify the parameter name X for the component. Where X denotes the number of parameters for the component. This is used in conjunction with the ParameterValueX
ParameterValueX	String	Specify the parameter value X for the component. Where X denotes the number of parameters for the component. This is used in conjunction with the ParameterName
ModelType	String	Specify the model type for the component.
ModelParameterNameX	String	Specify the model parameter name for this component's model.

Example

Process: SCH:PlaceIntegratedComponent

PlaceJunction process

Description

Place a junction on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units of .01 inch. Range is 0-6500. Example: Location.X=100
Location.Y	Integer	Specified in coordinate units of .01 inch. Range is 0-6500. Example: Location.Y=1500
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
Locked	True,False ,Toggle	Set Locked to true to prevent this junction from being edited graphically.

Example

Process: SCH:PlaceJunction

PlaceLine process

Description

Place a graphical line on the current schematic document.

Parameters

Parameter	Value	Description

Location.X	Integer	Specified in coordinate units. Example: Location.X=100
Location.Y	Integer	Specified in coordinate units. Example: Location.Y=1500
Corner.X	Integer	Specified in coordinate units. Example: Location.X=100
Corner.Y	Integer	Specified in coordinate units. Example: Location.X=100
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
StartAngle	Real	Start angle can be any degree from 0 to 360. Example: StartAngle=33.333
EndAngle	Real	End angle can be any degree from 0 to 360. Example: EndAngle=100
Color	Integer	Specifies RGB value converted from 6 digit hexadecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False ,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceLine

PlaceNetLabel process

Description

Place a net label on the current schematic document.

Parameters

Parameter	Value	Description
S	String	Text string up to 255 characters in length. Example: S=.date specifies the special string for displaying the current date.
Location.X	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Location.Y	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
FontID	String	The font is specified by a description string. The string contains seven fields each separated by a space. "<Size> <Rotation> <Underline> <Italic> <Bold> <StrikeOut> <FontName>". Size is an integer; Rotation cannot be set, always 0; Underline, Italic, Bold and Strikeout are boolean flags 0=False, 1=True; FontName is the actual font name. Examples: FontID=8 0 0 0 1 0 Helv - produces a Helvetica bold font with 8pt size. FontID=12 0 0 1 0 0 Times New Roman - produces a Times New Roman bold, italic font with 12pt size.
Selection	True,False , Toggle	This parameter has three possible states, True=on, False=off and Toggle=switch state based on previous setting.

Example

Process: SCH:PlaceNetLabel

Server Process Reference

PlaceNoErc process

Description

Place a NoErc to suppress no-connection error message for a net on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Location.Y	Integer	Specified in coordinate units of .01 inch. Range is 0-6500
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
FontID	String	The font is specified by a description string. The string contains seven fields each separated by a space. "<Size> <Rotation> <Underline> <Italic> <Bold> <StrikeOut> <FontName>". Size is an integer; Rotation cannot be set, always 0; Underline, Italic, Bold and Strikeout are boolean flags 0=False, 1=True; FontName is the actual font name. Examples: FontID=8 0 0 0 1 0 Helv - produces a Helvetica bold font with 8pt size. FontID=12 0 0 1 0 0 Times New Roman - produces a Times New Roman bold, italic font with 12pt size.
Selection	True,False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceNoErc

PlaceNote process

Description

The PlaceNote process places a memo like text container on the current schematic document. This note can be collapsed or in full view.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units. Example: Corner.X=200
Corner.Y	Integer	Specified in coordinate units. Example: Corner.Y=500
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black. Color=255 is red. Color=65280 is green. Color=16711680 is blue. Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125
TextColor	Integer	Specifies RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: TextColor=125

FontID	String	The font is specified by a description string. The string contains seven fields each separated by a space. "<Size> <Rotation> <Underline> <Italic> <Bold> <StrikeOut> <FontName>". Size is an integer; Rotation cannot be set, always 0; Underline, Italic, Bold and Strikeout are boolean flags 0=False, 1=True; FontName is the actual font name. Examples: FontID=8 0 0 0 1 0 Helv - produces a Helvetica bold font with 8pt size. FontID=12 0 0 1 0 0 Times New Roman - produces a Times New Roman bold, italic font with 12pt size.
Selection	True,False, Toggle	This Selection parameter has three possible states, True=on, False=off and Toggle=switch state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
ShowBorder	True,False,Toggle	Set ShowBorder to true to show the border around the note object. Use the Toggle value to toggle from True to False or False to True.
Alignment	True,False,Toggle	Set Alignment to true to align the text inside and around the note object. Use the Toggle value to toggle from True to False or False to True.
WordWrap	True,False,Toggle	Set WordWrap to true to wrap text around if text exceeds the length of the note object. Use the Toggle value to toggle from True to False or False to True.
ClipToRect	True,False,Toggle	Set ClipToRect to true to clip the text if it exceeds the length of the note object. Use the Toggle value to toggle from True to False or False to True.
Collapsed	True, False, Toggle	Set Collapsed to true to collapse the note object as a small triangle on the schematic. Use the Toggle value to toggle from True to False or False to True.
Author	String	Contains the text contents of the memo container.

Example

Process: SCH:PlaceNote

PlaceOffSheetConnector process

Description

Place a off sheet connector on a current schematic document.

Parameters

Parameter	Value	Description
OffSheetConnector	True, False	Set the OffSheetConnector to true or false.

Example

Process: SCH:PlaceOffSheetConnector

PlaceParameterSet process

Description

Places a parameter set object on the current schematic document.

Parameters

Parameter	Value	Description
ParameterSet	TestVectorIndex , Stimulus, PCBLayout	Set the ParameterSet to specify the parameter type.

Server Process Reference

Example

Process: SCH:PlaceParameterSet

PlacePart process

Description

Place a component part on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black. Color=255 is red. Color=65280 is green. Color=16711680 is blue. Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125
PinColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter
OverideColors	True, False, Toggle	Set the OverrideColors to true to override the global color setting. Use the Toggle value to toggle from True to False or False to True.
Selection	True, False, Toggle	This parameter has three possible states, True=on, False=off and Toggle=switch state based on previous setting.
CompDisplayMode	0..254	255 possible graphical mode representations of the same part object.
IsMirrored	True ,False, Toggle	If IsMirrored is set to true, the text string is flipped across the vertical axis. If IsMirrored is set to Toggle, the previous state is toggled, for example True to False or False to True.
PartId	String	Specifies the component part number.
ShowHiddenFields	True, False, Toggle	Set ShowHiddenFields to true to display hidden parameters for this component. Use the Toggle value to toggle from True to False or False to True.
ShowHiddenPins	True, False, Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
LibReference	String	Name of the part to be placed.
Library	String	Component library filename where the part exists. By default if no library is specified the component cache is used, if the part is still not found then all .lib files are searched in the current working directory.
FootPrint	String	Specify the Footprint name
Designator	String	Specify the designator name
PartType	String	Specify the part type
Description1 Description16	String	Specify Description 1 to 16 fields. Obsolete. Use Parameters instead.

SheetPartFileName		Specifies the full path and file name of the schematic document. Example: SheetPartFileName=C:ChildSheet.SCH
DisplayFieldNames	True, False, Toggle	Set DisplayFieldNames to true to display the field names. Use the Toggle value to toggle from True to False or False to True.
UseBrowser	True or False	By default this parameter is set to false if no value supplied. If UseBrowser = True, then a browse library dialog pops up allowing you to select a part to place on the schematic document. Otherwise a place part dialog appears instead with attributes fields to fill in.

By default, a place part dialog appears with the previous component attributes automatically assigned.

Example

Process: SCH:PlacePart

PlacePartFromLibrayEditor process

Description

Place a part from the library editor onto the schematic document.

Parameters

N/A

PlacePartFromSchEditor process

Description

Place a part from the schematic editor panel onto the schematic document.

Parameters

N/A

PlacePCBLayoutDirective process

Description

Add a directive to schematic nets for the routing of the associated PCB document.

Parameters

Parameter	Value	Description
RoutingTrackWidth	Integer	Specifies routing track width passed to PCB layout. Example: RoutingTrackWidth=8
RoutingViaWidth	Integer	Specifies routing via width passed to PCB layout. Example: RoutingViaWidth=40
NetTopology	0..6	Seven topology settings, 0=X-bias .. 6=Star Point. Example: NetTopology=3 sets net topology to Daisy Chain.
RoutingPriority	0..4	Five routing priorities, 0=Highest .. 4=Lowest. Example: RoutingPriority=2 sets priority to Medium.
Layer	0..21	Twenty two layer settings, 0=Undefined .. 21=Power Plane 4. Example: Layer=2 sets layer to Top, Layer=16 sets layer to Bottom.
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)).

Server Process Reference

		Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True,False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlacePCBLayoutDirective

PlacePieChart process

Description

Place a pie shape on the current document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Radius	Integer	Specified in units of .01 inch. Example: Radius=30
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
StartAngle	Real	Start angle can be any degree from 0 to 360. Example: StartAngle=33.333
EndAngle	Real	End angle can be any degree from 0 to 360. Example: EndAngle=100
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True,False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.

Example

Process: SCH:PlacePieChart

PlacePin process

Description

Place an electrical pin on a current schematic or library document.

Parameters

Parameter	Value	Description
Name	String	Text string up to 255 characters in length. Example: Name=Q1
Number	String	Text string up to 255 characters in length. Example: Number=10
Location.X	Integer	Specified in coordinate units

Location.Y	Integer	Specified in coordinate units
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True,False, Toggle	This parameter has three possible states, True=on, False=off and Toggle=switch state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
Dot	True, False, Toggle	Sets Dot Symbol option.
Clk	True, False, Toggle	Sets Clk Symbol option.
Electrical	0..7	() Eight Electrical Types, 0=Input .. 7=Power. Example: Electrical=4 sets pin electrical type to Passive.
IsHidden	True, False, Toggle	Set IsHidden to true to hide this pin. Use the Toggle value to toggle from True to False or False to True.
ShowName	True, False, Toggle	Set ShowName to true to display the name next to its associated pin. Use the Toggle value to toggle from True to False or False to True.
ShowNumber	True, False, Toggle	Set ShowNumber to true to display the number next to its associated number. Use the Toggle value to toggle from True to False or False to True.
PinLength	Integer	Specified in coordinate units of .01 inch.

Example

Process: SCH:PlacePin

PlacePolygon process

Description

Place a graphical polygon shape on the current schematic document.

Parameters

Parameter	Value	Description
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125

TR0124 (v1.5) August 4, 2008

Server Process Reference

Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True, False, Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
Location1.X..Location 50.X	Integer	Specified in coordinate units
Location1.Y..Location 50.Y	Integer	Specified in coordinate units

Example

Process: SCH:PlacePolygon

PlacePort process

Description

Place a port on the current schematic document.

Parameters

Parameter	Value	Description
Name	String	Text string up to 255 characters in length. Example: Name=D1
Style	0..3	Four styles, 0=None, 1=Left, 2=Right, 3=Left & Right. Example: Style=1 sets style to Left.
IOType	0..3	Four I/O types, 0=Unspecified, 1=Output, 2=Input, 3=Bidirectional. Example: IOType=3 sets I/O type to Bidirectional.
Alignment	0..2	Three alignment settings, 0=Center, 1=Left, 2=Right. Example: Alignment=0 aligns text in the center of the port.
Width	Integer	Specified in coordinate units
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: AreaColor=8224125
TextColor	Integer	RGB value converted from 6 digit hexadecimal number. See Color parameter.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True, False, Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units

Example

Process: SCH:PlacePort

PlacePowerPort process

Description

Place a specified power port on a schematic sheet.

Parameters

Parameter	Value	Description
S	String	Specifies the text for the power port object.
Style	0..6	0..6 = Circle, Arrow, Bar, Wave, Power Ground, Earth Ground, Earth respectively
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Orientation	0..3	Four orientation settings, 0=0 degrees, 1=90 degrees, 2=180 degrees 3=270 degrees. Example: Orientation=1 rotates annotation object vertically 90 degrees.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Repeat	True, False	If the Repeat parameter is assigned to true, every-time you place a power port, you have a floating power port ready to be placed again, like a rubber stamp. If the repeat parameter is not present, only one copy will be placed on the sheet.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlacePowerPort

Parameters : Color=128|Orientation=3|S=GNDBUS[..]|Style=4

PlaceProbe process

Description

The PlaceProbe process is used to place a simulation probe onto the current schematic document. A Probe is a special marker which is placed on the schematic document to identify nodes for digital simulation.

Parameters

Parameter	Value	Description
S	String	Specifies the text for the power port object.
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Server Process Reference

Example

Process: SCH:PlaceProbe

PlaceRectangle process

Description

Place a rectangle on the current schematic document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.

Example

Process:SCH:PlaceRectangle

PlaceRoundRectangle process

Description

Place a round rectangle on the current document.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
CornerXRadius	Integer	Specified in coordinate units, the arc in X direction
CornerYRadius	Integer	Specified in coordinate units, the arc in Y direction.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted

		decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.

Example

Process: SCH:PlaceRoundRectangle

Parameters : Location.X= 100 | Location.Y = 200 | CornerXRadius = 30 | CornerYRadius = 30 | Corner.X = 500 | Corner.Y = 500

PlaceSheetEntry process

Description

The PlaceSheetEntry process is used to add a sheet entry to a sheet symbol. A sheet entry is used to direct signals to another sheet in a hierarchical design. There are four types of Sheet Entry symbols, Input, Output, Bi-directional and Unspecified.

Parameters

N/A

Example

Process: SCH:PlaceSheetEntry

PlaceSheetSymbol process

Description

The PlaceSheetSymbol process is used to place a sheet symbol onto a schematic document. A sheet symbol represents another schematic document in a hierarchical design. Sheet symbols include sheet entry symbols, which provide a connection point for signals between the parent and child sheets.

Parameters

Parameter	Value	Description
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
XSize	Integer	Specified in coordinate units
YSize	Integer	Specified in coordinate units
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125

Server Process Reference

Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.
IsSolid	True,False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
ShowHiddenFields	True,False,Toggle	Set ShowHiddenFields to true to show the hidden fields for this sheet symbol. Use the Toggle value to toggle from True to False or False to True.
SheetFileName	String	Text string up to 255 characters in length. Specifies the full path and file name of the schematic document. Example: FileName=C:Child.SCH
SheetName	String	Text string up to 255 characters in length

Example

Process: SCH:PlaceSheetSymbol

Parameters : Location.X=100|Location.Y=100|XSize = 200|YSize=300|IsSolid=True|ShowHiddenFields=True|Color=8092|AreaColor=1100|LineWidth = 2

PlaceStimulus process

Description

The PlaceStimulus process is used to place a stimulus directive onto the schematic document.

Parameters

Parameter	Value	Description
S	String	Text string up to 255 characters in length. Example: S=BaudClk
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceStimulus

PlaceTestVectorIndex process

Description

The PlaceTestVectorIndex process is used to place a Test Vector directive onto the schematic document. Test Vectors are special symbols used to identify a node with a simulation test vector. The test vectors are referred to by a column number, which indicates the column of the test vector file to use when the simulation is run.

Parameters

Parameter	Value	Description
S	String	Text string up to 255 characters in length. Example: S=BaudClk
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceTestVectorIndex

Parameters : Location.X = 200 | Location = Y | S = Test Vector Index | Color = 65280

PlaceTextFrame process

Description

The PlaceTextFrame process is used to place detailed notes or descriptive text onto the schematic document.

Parameters

Parameter	Value	Description
LongString	String	Text up to 64000 characters in length.
Location.X	Integer	Specified in coordinate units
Location.Y	Integer	Specified in coordinate units
Corner.X	Integer	Specified in coordinate units
Corner.Y	Integer	Specified in coordinate units
CornerXRadius	Integer	Specified in coordinate units, the arc in X direction
CornerYRadius	Integer	Specified in coordinate units, the arc in Y direction.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
AreaColor	Integer	Specifies RGB value converted from 6 digit hexidecimal number. See Color parameter. Example: AreaColor=8224125
TextColor	Integer	Specifies RGB value converted from 6 digit hexadecimal number. See Color parameter. Example: TextColor=125
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
FontID	String	The font is specified by a description string. The string contains seven fields

Server Process Reference

		each separated by a space. "<Size> <Rotation> <Underline> <Italic> <Bold> <StrikeOut> <FontName>". Size is an integer; Rotation cannot be set, always 0; Underline, Italic, Bold and Strikeout are boolean flags 0=False, 1=True; FontName is the actual font name. Examples: FontID=8 0 0 0 1 0 Helv - produces a Helvetica bold font with 8pt size. FontID=12 0 0 1 0 0 Times New Roman - produces a Times New Roman bold, italic font with 12pt size.
IsSolid	True, False,Toggle	Set IsSolid to true to fill the background of this object with the area color. Use the Toggle value to toggle from True to False or False to True.
ShowBorder	True,False,Toggle	Set ShowBorder to true to show the border around the text frame object. Use the Toggle value to toggle from True to False or False to True.
Alignment	True,False,Toggle	Set Alignment to true to align the text inside and around the text frame object. Use the Toggle value to toggle from True to False or False to True.
WordWrap	True,False,Toggle	Set WordWrap to true to wrap text around if text exceeds the length of the text frame object. Use the Toggle value to toggle from True to False or False to True.
ClipToRect	True,False,Toggle	Set ClipToRect to true to clip the text if it exceeds the length of the text frame object. Use the Toggle value to toggle from True to False or False to True.
Selection	True, False,Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceTextFrame

Parameters : Location.X=500 | Location.Y = 500

PlaceWire process

Description

Place an electrical wire on the current schematic document.

Parameters

Parameter	Value	Description
LineWidth	0..3	Four line widths, 0=Smallest, 1=Small, 2=Medium, 3=Large. Example: LineWidth=2 sets line width to medium.
Color	Integer	Specifies RGB value converted from 6 digit hexidecimal number. For example the color blue would be RGB:0,0,255 and Hex:FF0000 therefore the converted decimal value would be 16711680. The following formula may be used to calculate the required value, R+256*(G+(256*B)). Examples: Color=0 is black Color=255 is red Color=65280 is green Color=16711680 is blue Color=16777215 is white.
Repeat	True, False	If the Repeat parameter is assigned to true, every-time you place a wire object, you have another floating wire ready to be placed again, like a rubber stamp. If the repeat parameter is not present, only one copy will be placed on the sheet.
Selection	True, False, Toggle	This parameter has three possible selection states for the object. True=Selected, False= Not selected and Toggle=switch selection state based on previous setting.

Example

Process: SCH:PlaceWire

Parameters: LineWidth = 3 | Color = 255 |

PreviousComponentLibraryEditor process

Description

The PreviousComponentLibraryEditor process is used to go to the previous component in a current library document.

Parameters

N/A

PreviousComponentPart process

Description

The PreviousComponentPart process is used to show the previous part of a multiple part component in the current library document.

Parameters

N/A

PrintDocument process

Description

N/A

Parameters

N/A

Redo process

Description

Redoes the previous operation.

Parameters

N/A

Example

Process: SCH:Redo

RemoveComponentPart process

Description

The RemoveComponentPart process is used to delete a part from a component within a current library document.

Parameters

N/A

RemoveDuplicateComponentNames

Description

The RemoveDuplicateComponentNames process is used to remove/delete duplicate components from the current library. The first component name in the library is saved and all other components with the same name are removed.

Parameters

N/A

Example

Process: SCH:RemoveDuplicateComponentNames

RemoveTemplate process

Description

Remove any template information from the current schematic document.

Parameters

N/A

Server Process Reference

Example

Process: SCH:RemoveTemplate

RenameObjectText process

Description

Changes the text field associated with an object from the Browser such as the designator of the current part.

Parameters

N/A

Example

Process: SCH:RenameObjectText

ReportComponent processs

Description

The ReportComponent process is used to retrieve general information about the current component in the schematic library.

Parameters

Parameter	Value	Description
FileName	String	Text string up to 255 characters in length. Specifies the full path and file name of the report document.
Show	True,False	Load and display report in text editor if True. Defaults to True

Example

Process: SCH:ReportComponent

Parameters : Filename = Report | Show = True

ReportComponentLibrary process

Description

The ReportComponentLibrary process is used to retrieve general information about the current schematic library.

Parameters

Parameter	Value	Description
FileName	String	Text string up to 255 characters in length. Specifies the full path and file name of the report document.
Show	True,False	Load and display report in text editor if True. Defaults to True

Example

Process: SCH:ReportComponentLibrary

Parameters : Filename = ReportLibrary.Txt | Show = True

ResetUniqueIds process

Description

The ResetUniqueIds process resets the Unique Ids of all components on the schematic document.

Parameters

N/A

Example

Process: SCH:ResetUniqueIds

Select process

Description

Select objects inside/outside an area, all objects or a connection on a current schematic sheet.

Parameters

Parameter	Value	Description
Action	(InsideArea,OutsideArea, All, Connection	Set Action to the scope required.

Example

Process: SCH:Select

Parameters : Action= All

See also

DeSelect process.

SelectionMemory process

Description

Perform one of the many selection memory processes including displaying the Selection Memory dialog.

Parameters

Parameter	Value	Description
Action	ShowDialog, Store, Recall, StorePlus, RecallPlus, Clear, Apply	Set Action to the required action.
Index	1..n	Set Index to one of the selection memories.

Notes

There are up to 9 Store and Recall memory states for the Selection Memory dialog.

Example

Process: SCH : SelectionMemory

Parameters : Action = ShowDialog | Index = 1

SendObjectToBack process

Description

The SendObjectToBack process is used to send objects on the schematic document to the back of all other objects.

Parameters

N/A

SendObjectToBackOf process

Description

The SendObjectToBackOf process is used to move an object on the schematic document behind another object. This process is similar to the SendObjectToBack process, except that you will be prompted to select the object that the original object will be placed behind.

Parameters

N/A

SetLocationMark process

Description

The SetLocationMark (1..10) processes are used to tag or mark specific locations on the schematic worksheet. The JumpLocationMark (1..10) processes are used to move the cursor to this set location.

Parameters

Server Process Reference

Parameter	Value	Description
LocationMark	1..10	Set location mark to one of one the 10 location marks on the schematic.
CurrentLocation	True, False	If CurrentLocation is false, the location mark is set at where you click on the schematic document.

SetupArrayPlacement process

Description

The SetupArrayPlacement process is used to define a multiple placement of the clipboard contents onto the current schematic document.

Parameters

Parameter	Value	Description
ItemCount	Integer	Specifies the item count value.
TextIncrement	Integer	Specifies the text increment value.
HorizontalSpace	Integer	Specifies the horizontal spacing value.
VerticalSpace	Integer	Specifies the vertical spacing value.

Example

Process: SCH:SetupArrayPlacement

Parameters : ItemCount = 4 | TextIncrement = 10 | HorizontalSpace = 10 | VerticalSpace = 10

SetupPreferences process

Description

The SetupPreferences process is used to define various Schematic and Library Editor settings, such as cursor shape, default template file, grid display type, selection color, auto-junction, and other options that apply to all Schematic and Library Editor documents.

Parameters

Parameter	Value	Description
Tab	Schematic, Graphical Editing, Compiler, AutoFocus, Break Wire, Default Primitives, Orcad (Tm) Options	If none supplied, the Preferences dialog with Schematic Tab is shown. Otherwise if Value is specified for the particular tab, then Preferences dialog will be displayed with that tab active. Take note of white space between characters.
SelectionColor	Integer	The SelectionColor value is composed of three Red, Green and Blue values according to this formula: RedVal + 256*(GreenVal + 256*BlueVal).
ResizeColor	Integer	The ResizeColor value is composed of three Red, Green and Blue values according to this formula: RedVal + 256*(GreenVal + 256*BlueVal).
TranslateRotateColor	Integer	The TranslateRotateColor value is composed of three Red, Green and Blue values according to this formula: RedVal + 256*(GreenVal + 256*BlueVal).
DocumentScope	0,1	0 = Current Document. 1= Open Documents.
LibraryScope	0,1	0 = Current Library. 1 = Open libraries.

ConfirmSelectionMemoryClear	Integer	0 False, 1 True. If true, you are prompted before you can clear the selection memory.
SnapToCenter	True,False,Toggle	Set SnapToCenter to true and when the object is being moved or dragged by its center (for objects which do not have a reference point such as a rectangle). Set it to false and the reference point is used for dragging or moving objects (such as library components or ports). Use the Toggle value to toggle from True to False or False to True.
UseOrcadPorthWidth	True,False,Toggle	Use True if you wish to have existing port widths be recalculated based on the number of characters in their name and they cannot be graphically resized. Use False to use the default port width. Use the Toggle value to toggle from True to False or False to True.
SelectionReference	True,False,Toggle	Set SelectionReference to true and then, when you select Edit » Copy or Edit » Cut from the menu, you will be asked to select a reference point. This is useful when copying a section of circuit which is to be pasted back into a schematic sheet. This reference point will be the point where the section of circuit will be held when pasting. Use the Toggle value to toggle from True to False or False to True.
UndoRedoStackSize	Integer	The UndoRedoStackSize parameter specifies the number of actions held in the Undo Buffer. The default value is 50. Enter a value for this parameter to set the Undo Buffer size. There is no limit to the size of the Undo Buffer, however, the larger the size, the more main memory is used to store undo information.
ConvertSpecialStrings	True,False,Toggle	Set ConvertSpecialStrings to true to see the contents of the special strings on the schematic document, as they appear on a printout. Use the Toggle value to toggle from True to False or False to True.
MaintainOrthogonal	True,False,Toggle	Set MaintainOrthogonal to True and when you drag components, any wiring that is dragged with the component is kept orthogonal (i.e. corners at 90 degrees). Set it to False and wiring dragged with a component will be repositioned obliquely. Use the Toggle value to toggle from True to False or False to True.
DisplayPrinterFonts	True,False,Toggle	Not all fonts are supported on all output devices (and Windows will automatically substitute). To see what the text is going to look like on the printout, Set the DisplayPrinterFonts to true.. Use the Toggle value to toggle from True to False or False to True.
HotSpotGridDistance	Integer	Set HotSpotGridDistance to setup the electrical hot spot grid.
SnapToHotspot	True,False,Toggle	Set SnapToHotspot to true and when the object is being moved or dragged it is snapped to by the nearest electrical hot spot (eg, the end of a pin). Use the Toggle value to toggle from True to False or False to True.
AutoZoom	True,False,Toggle	Set AutoZoom to true and the schematic sheet is automatically zoomed when jumping to a component. Set it to false and the Zoom level remains as it was. Use the Toggle value to toggle from True to False or False to True.
AutoJunction	True,False,Toggle	Set AutoJunction to True and the system-generated junctions are displayed for bus objects on a schematic document. Use the Toggle value to toggle from True to False or False to True.
OptimizePolylines	True,False,Toggle	Set OptimizePolylines to True to prevent extra wires, poly-lines or

		buses overlapping on top of each other and the overlapping wires, poly-lines or busses are removed automatically.
		Note: You need to set this to True to have the ability to automatically cut a wire and terminate onto any two pins of this component when this component is dropped onto this wire.
		Use the Toggle value to toggle from True to False or False to True.
ComponentsCutWires	True,False,Toggle	Set ComponentsCutWires to True so you can drop a component onto a schematic wire and then the wire is cut into two segments and the segments are terminated onto any two hot pins of this component automatically. You need to have the OptimizePolylines value set to true as well.
		Use the Toggle value to toggle from True to False or False to True.
AddTemplateToClipboard	True,False,Toggle	Set AddTemplateToClipboard to true, so the current sheet template is also copied to the clipboard when you copy or cut from the current schematic sheet.
		Use the Toggle value to toggle from True to False or False to True.
DefaultTemplateFilename	String	Enter the filename to set the default user template file that will be used to create new schematic sheets. Enter the full path and file name of a schematic template file.
		If this value is empty, a default blank schematic is created when you open a new schematic sheet.
AutoPanJumpDistance	Integer	Set the AutoPanJumpDistance to set the size of each auto-panning step. The step size determines how fast the document pans when auto-panning is enabled. The smaller the value, the slower or finer the auto-panning movement.
AutoPanShiftJumpDistance	Integer	Set the AutoPanShiftJumpDistance to set the size of each step when the SHIFT key is held during auto-panning. The shift step size determines how fast the document pans when auto-panning is enabled and the SHIFT key is pressed. The smaller the value, the slower or finer the auto-panning movement.
PinNameMargin	Integer	Normally, component pin names are displayed inside the body of the component, adjacent to the corresponding pin. This value controls the placement of component pin names. It specifies the distance (in hundredths of an inch) from the component outline to the start of the pin name text. The default is 5.
PinNumberMargin	Integer	Normally, component pin numbers are displayed outside the body of the component, directly above the corresponding pin line. This value controls the placement of the pin numbers. It specifies the distance (in hundredths of an inch) from the component outline to the start of the pin number text. The default is 8.
ShowPinDirection	True,False,Toggle	Set the ShowPinDirection to True to display the direction of pins of components on a schematic document. The pin direction is indicated by the orientation of a triangle symbol.
		Use the Toggle value to toggle from True to False or False to True.
ShowPortDirection	True,False,Toggle	Set the ShowPortDirection to True and ports' style can be determined by the I/O type attribute of corresponding ports.
		Note, if this value is enabled, the setting of the style in the **Change Port** properties dialog is overridden by the I/O type option if the **Show** Port **Direction** option is enabled.
		Use the Toggle value to toggle from True to False or False to True.

UnconnectedLeft2Right	True,False,Toggle	Set the UnconnectedLeft2Right value to True and those unconnected ports on a schematic document is displayed in a Left To Right direction (as a Right style).
		Use the Toggle value to toggle from True to False or False to True.
ShowSheetEntryDirection	True,False,Toggle	Set the ShowSheetEntryDirection to True and the sheet entries' style can be determined by the I/O type attribute of corresponding sheet entries.
		If this value is enabled, the setting of the style in the **Change Sheet Entry** properties dialog is overridden by the I/O type option if the **Show Sheet Entry Direction** option is enabled.
		Use the Toggle value to toggle from True to False or False to True.
DefaultPrimsPermanent	True,False,Toggle	Set the DefaultPrimsPermanent value to true and the default values for all primitives are locked and cannot be changed when you are placing primitives on the schematic.
		If this value is false (default), when you edit the properties of an object while placing it (by pressing the TAB key while the object is floating on the cursor before placement), any changes you make to its properties are subsequently used as the defaults when you next place a new object of the same type.
		Use the Toggle value to toggle from True to False or False to True.
IgnoreSelection	True,False,Toggle	By default, selecting an object or a group of objects is considered an action which may be undone. Complex selections can quickly fill the Undo Buffer's **Stack Size**. Set IgnoreSelection to True to stop each selection action from going to the Undo Buffer, thus reducing the demands on the computer system's memory.
		Use the Toggle value to toggle from True to False or False to True.
ClickClearsSelection	True,False,Toggle	Set ClickClearsSelection to True if you want to deselect all design objects by clicking any where on the schematic workspace.
		Set it to False if you do not want to have this click anywhere to deselect all ability and the selection is cumulative.
		Note: regardless of the setting, you can deselect a selected design object by clicking on it.
		Use the Toggle value to toggle from True to False or False to True.
DoubleClickRunsInspector	True,False,Toggle	Set DoubleClickRunsInspector to True to bring up the **Inspector** dialog instead of the design object's properties dialog when you double click on a design object.
		Set DoubleClickRunsInspector to False if you want to see the design object's properties dialog when you double click on a design object.
		Use the Toggle value to toggle from True to False or False to True.
Sensitivity	Integer	Set Sensitivity to set the auto panning speed. If 0, the slowest or finest the auto-panning movement or 100 fastest or coarsest the movement.
SingleSlashNegation	True,False,Toggle	Set SingleSlashNegation to true and a net name can be negated by typing a backslash character before the first letter in the net name. This applies to ports, net labels and sheet entries.
		Use the Toggle value to toggle from True to False or False to True.
RunInPlaceEditing	True,False,Toggle	Set RunInPlaceEditing to True, then the focused text field may be directly edited within the Schematic Editor, rather than in a dialog box. After focusing the field you wish to modify, clicking upon it

		again or pressing the **F2** shortcut key will open the field for editing.
		Set it to false and you cannot edit the text directly and you have to edit it from the **Parameter Properties** dialog. You can just graphically move this text field.
		Use the Toggle value to toggle from True to False or False to True.
DefaultPowerGndName	String	When placing a Power Ground style power port in a schematic, its net name will default to this value.
		If the value is empty, then the last valid value will apply to any new ports of this style. The default name for Power Ground is GND.
DefaultSignalGndName	String	Set the DefaultSignalGndName with a name and when placing a Signal Ground style power port in a schematic, its net name will default to this value.
		If the value is empty, then the last valid value will apply to any new ports of this style. The default name for Signal Ground is SGND.
DefaultEarthName	String	Set the DefaultEarthName with a name and when placing an Earth power port in a schematic, its net name will default to this value.
		If the value is empty, then the last valid value will apply to any new ports of this style. The default name for Earth is EARTH.
StringIncA	String	Set StringIncA a value to auto-increment on pin designators of a component when you are placing pins for a component. This is used for building components in the Library editor. Normally you would use a positive increment value for pin designators and negative increment value for pin names. Eg 1, 2,3 for pin designators and D8, D7, D6 for pin names. Thus StringIncA (**Primary)** = 1 and StringIncB (**Secondary)** = -1 and set **Display Name** to D8 and **Designator** to 1 in the **Pin Properties** dialog before you place the first pin.
StringIncB	String	Set StringIncB a value to auto-increment on pin names of a component when you are placing pins for a component. This can be used for building components in the Library editor. Normally you would use a positive increment value for pin designators and negative increment value for pin names. Eg 1, 2,3 for pin designators and D8, D7, D6 for pin names. Thus StringIncA (**Primary)** = 1 and StringIncB (**Secondary)** = -1 and set **Display Name** to D8 and **Designator** to 1 in the **Pin Properties** dialog before you place the first pin.
MarkManualParameters	True,False,Toggle	Set the MarkManualParaemters to true and Parameters are displayed with a dot which denotes that auto-positioning has been turned off and that parameters are moved or rotated with its parent object (component for example). To hide the dots, Set this MarkManualParameters to False.
		Use the Toggle value to toggle from True to False or False to True.
CtrlDbleClickGoesDown	True,False,Toggle	Set CtrlDbleClickGoesDown to True to open the sub-sheet of its associated sheet symbol by double clicking on this sheet symbol.
		Set it to false and when you double-click on a sheet symbol, the change properties dialog is displayed instead.
		Use the Toggle value to toggle from True to False or False to True.
CutterGridSizeMultiple	2-10	Set the CutterGridSizeMultiple value to snap to a section of the polyline that is the same size as the specified **Snap Grid Size Multiple** option from the Preferences page prior to cutting the polyline. Multiplicity range from 2 to 10.

CutterFixedLength	Integer	Set the CutterFixedLength value so that a fixed length of the section of a polyline can be snapped prior to cutting this polyline.
MultiPartNamingMethod	Integer	If 0, alpha suffix is used for naming of components. if 1, numeric suffix is used for naming of componentsl
BufferedPainting	True,False,Toggle	Set BufferedPainting to True to buffer the redraw process so the graphical display looks less jerky when you zoom in or out or drag the workspace. This option increases the demands on the computer system. Disable this option if you want faster redraw processes but there will be flicker.
Metafile_NoERCMarkers	True,False,Toggle	Set Metafile_NoERCMarkers to True to include **No ERC Marker** design objects when copying to the clipboard or when printing.
Metafile_ParameterSets	True,False,Toggle	Set Metafile_ParameterSets to True to include **Parameter Sets** design objects when copying to the clipboard or when printing.
MustHoldShiftToSelect	True,False,Toggle	Set MustHoldShiftToSelect to True if you wish to use the SHIFT key to select specific primitives as specified by the **Primitives** list. This list is accessed by clicking the **Primitives** button. Disable this option and you can select a primitive normally.
VisibleGridStyle	0,1	0 = Dot Grid, 1 = Line Grid
GraphicsCursorStyle	0,1,2	0 = Large Cursor 90, 1= Small Cursor 90, 2 = Small Cursor 45
OrcadFootPrint	0..8	0 ..7 = Part Fields 1.. 8, 8 Ignore
AutoPanStyle	0,1,2	AutoPan Off, Auto Pan Fixed Jump, AutoPan Recenter
PolylineCutterMode	0,1,2	0 = Cutter Snap to Segment, 1 = Grid Size, 2 = Fixed Length
ShowCutterBoxMode	0,1,2	0 = Never, 1 = Always, 2 = On Polyline
ShowCutterMarksMode	0,1,2	0 = Never, 1 = Always, 2 = On Polyline
AF_PlacementDim	True, False, Toggle	Set AF_PlacementDim to true if you want to have the ability to dim unconnected objects on the schematic object when you are editing this connected object. Use the Toggle value to toggle from True to False or False to True.
AF_PlacementZoom	True, False, Toggle	Set AF_PlacementZoom to true if you want to have the ability to zoom in the connected object that is being edited. Use the Toggle value to toggle from True to False or False to True.
AF_PlacementThicken	True, False, Toggle	Set AF_PlacementThicken to true if you want to have the ability to thicken the surrounding connected objects when you are moving a object connected to a network of connected objects on the schematic sheet. Use the Toggle value to toggle from True to False or False to True.
AF_EditLocationDim	True, False, Toggle	Set AF_EditLocationDim to true if you want to have the ability to dim all unconnected objects when you are moving a object connected to a network of connected objects on the schematic sheet. Use the Toggle value to toggle from True to False or False to True.
AF_EditLocationZoom	True, False, Toggle	Set AF_EditLocationZoom to true if you want to have the ability to zoom in the surrounding connected objects when you are moving a object connected to a network of these connected objects on the schematic sheet. Use the Toggle value to toggle from True to False or False to True.
AF_EditLocationThicken	True, False, Toggle	Set AF_EditLocationThicken to true if you want to have the ability to

Server Process Reference

		thicken the surrounding connected objects when you are moving a object connected to a network of connected objects on the schematic sheet. Use the Toggle value to toggle from True to False or False to True.
AF_EditPropertyDim	True, False, Toggle	Set AF_EditPropertyDim to True, if you want to have the ability to dim all unconnected objects on the schematic sheet when you are re-sizing a connected object. Use the Toggle value to toggle from True to False or False to True.
AF_EditPropertyZoom	True, False, Toggle	Set AF_EditPropertyZoom to true if you want to have the ability to zoom in all the connected objects of a network on the schematic sheet when you are re-sizing a connected object of this network. Use the Toggle value to toggle from True to False or False to True.
AF_EditPropertyThicken	True, False, Toggle	Set AF_EditPropertyThicken to true, if you want to have the ability to thicken all the connected objects of a network on the schematic sheet when you are re-sizing a connected object part of this network. Use the Toggle value to toggle from True to False or False to True.
AF_EditInPlaceDim	True, False, Toggle	Set AF_EditInPlaceDim to true if you want to have the ability to dim all unconnected objects when you are placing a new object on the schematic sheet. You can toggle the Electrical Grid in the **Preferences** dialog to help you with the ease of placement of electrically aware objects. Use the Toggle value to toggle from True to False or False to True.
AF_EditInPlaceZoomText	True, False, Toggle	Set AF_EditInPlaceZoomText to true if you want to have the ability to zoom in the text string of a connected object (for example the port) that is being edited. Use the Toggle value to toggle from True to False or False to True.
AF_EditInPlaceZoomConnection	True, False, Toggle	Set AF_EditInPlaceZoomConnection to true if you want to have the ability to zoom in the connected object that is being edited. Set Use the Toggle value to toggle from True to False or False to True.
AF_DimLevel	Integer	Set AF_DimLevel of the dimming level. A value of 0, objects are dimmed to the point of being almost transparent, and a value of 100, objects are slightly dimmed.
AF_ThickenDelay	Integer	Set AF_ThickenDelay of the thickening delay. A value of 0, the delay to thicken connected objects are minimal, a value of 100 the delay to thicken is longer.
ConvertCrossJunctions	True, False, Toggle	Set ConvertCrossJunctions to True and when the addition of a wire would create a four-way junction, this is converted into two adjacent three-way junctions. Set it to False and when a four way junction is created, the two wires crossing at the intersection are not joined electrically and if the **DisplayCrossOvers** is True a cross over is shown on this intersection. Use the Toggle value to toggle from True to False or False to True.
DisplayCrossOvers	True, False, Toggle	Set DisplayCrossOvers to True and the wiring cross-overs will be displayed with little bridges on the currently focussed schematic sheet. Use the Toggle value to toggle from True to False or False to True.
ShowHints	True, False, Toggle	Set ShowHints to True and you have ability to see compile hints on the schematic sheet and hover over the underlined object (not the squiggle itself) for details. Use the Toggle value to toggle from True

		to False or False to True.
DefaultSheetStyle	0..17	0 = A4,1 =A3, 2= A2,3=A1, 4=A0, 5 = A, 6= B, 7 = C, 8 = D, 9 = E, 10 = Letter, 11 = Legal, 12 = Tabloid, 13 = OrcadA, 14 = OrcadB, 15 = OrcadC, 16 = OrcadD, 17 = OrcadE

Example

Process: SCH:SetUpPreferences

Parameters : ConvertSpecialStrings = Toggle | ShowHints = True | DisplayCrossOvers = True

SetupPrinter process

Description

N/A

Parameters

N/A

SynchronizeHierarchy process

Description

Updates ports and sheet entries in the project, current document or the chosen sheet symbol.

Parameters

Parameter	Value	Description
Action	WholeProject, WholeDocument, SingleSheetSymbol	Set Action to Whole Project to synchronize across the whole project, set to Document to synchronize across the document or set to SingleSheetSymbol to update the ports and sheet entries for this Sheet Symbol only.

Example

Process: SCH:SynchronizeHierarchy

Parameters : Action = WholeDocument

ToggleComponentModeDisplay process

Description

The ToggleComponentModeDisplay process toggles a component into one of its alternate component modes if any. The default mode for the component is Mode0 and the next modes are Mode1..Mode 254 if any exist.

Parameters

Parameter	Value	Description
Action	Previous,Next, Add, Remove, Moden	Previous value goes to the previous component mode. Next goes to the next component mode. Add creates a new component mode and you can define a new component in the library. Remove deletes the current component mode, and Moden goes to the specified component mode, where n is 0 to 254.

Example

Process: SCH:ToggleComponentModeDisplay

Parameters : Action = Mode0

ToggleElectricalGrid process

Description

The ToggleElectricalGrid process is used to turn the Electrical grid on or off. The Electrical grid defines an array of points in the workspace that restricts cursor movement and the placement of objects. Turning the Electrical grid off allows you to place objects at any location on the schematic document without any restrictions.

Server Process Reference

Parameters

N/A

ToggleHiddenPins process

Description

The ToggleHiddenPins process is used to show or hide all the pins that have the attribute of Hidden. This process does not change the Hidden property of pins.

Parameters

N/A

ToggleSelection process

Description

Toggles the selection state of selected objects on a schematic document.

Parameters

N/A

Example

Process: SCH:ToggleSelection

ToggleSingleObjectSelection process

Description

The ToggleSingleObjectSelection process is used to select and de-select objects on a schematic by moving the cursor over the object and clicking SHIFT + LEFT MOUSE.

Parameters

N/A

ToggleSnapGrid process

Description

The ToggleSnapGrid process is used to turn the snap grid on or off. The snap grid defines an array of points in the workspace which restrict cursor movement and the placement of primitives. Turning the snap grid off allows you to place primitives at any location on the schematic document without any restrictions.

Parameters

N/A

ToggleVisibleGrid process

Description

The ToggleVisibleGrid process is used to turn the visible grid on or off in the current document window. The visible grid provides visual reference as you move around the schematic and can be displayed as dots or straight lines.

Parameters

N/A

Undo process

Description

Undoes the current operation.

Parameters

N/A

UpdateComponentsFromLibraryEditor process

Description

The UpdateComponentFromLibraryEditor process is used to automatically update/replace matching parts in opened schematic documents with the component in the current documet.

Parameters

Parameter	Value	Description
Filename	String	The filename of the library
LibRef	String	The name of the part to be updated.

Example

Process: SCH:UpdateComponentsFromLibraryEditor

UpdateCurrentTemplate process

Description

The UpdateCurrentTemplate process is used to update the current sheets template information from its template file.

Parameters

N/A

UpdateLibraryMask process

Description

N/A

Parameters

N/A

UpdatePartDatabaseLinks process

Description

N/A

Parameters

N/A

UpdatePartFromLibraryEditor process

Description

The UpdatePartFromLibraryEditor process is used to update all parts in every schematic that is open from the library editor.

Parameters

Parameter	Value	Description
Filename	String	The filename of the library.
LibRef	String	The name of the part to be updated.

UpdatePartsFromLibraryList process

Description

The UpdatePartsFromLibraryList process is used to update all parts in every single sheet that is open, from libraries listed in the Change Library File List dialog box.

Parameters

N/A

Zoom process

Description

The Zoom process is used to set the zoom level of the current Schematic document. Depending upon the parameters, a number of zoom actions can be performed from refreshing the screen to displaying a specified region of the schematic document.

Parameters

Parameter	Value	Description

Server Process Reference

ZoomLevel	Real	Prompts for a zoom value if not specified
Action	ZoomIn,ZoomOut,Pan, Redraw, All	Use ZoomIn to zoom in a document, ZoomOut to zoom out of a document, pan to pan across a document, Redraw to refresh the document or All to resize the document in full view.
Object	Window, Point, Selected	Choose which point of reference to display the specified region (Window, Point or Selected) of the schematic document.

Example

Process: SCH:Zoom

Parameters : ZoomLevel = 0.5

Work Space Manager Processes

This section covers the Work Space Manager processes and their parameters (if any).

ClipBoardAction process

Description

Execute a specified action for the clipboard, to cut, copy, paste or clear the contents from the Workspace manager clipboard. For example you are able to copy, cut, paste and clear different output jobs for each project.

Parameters

Parameter	Value	Description
Action	Cut, Copy, Paste, Clear	
ObjectKind	OutputSingle	

Example

Process: WorkSpaceManager:ClipBoardAction

Parameters : Action = Copy | ObjectKind = OutputSingle

ChangeObject process

Description

Change Output Generator properties for the Output jobs.

Parameters

Parameter	Value	Description
Action	Cut, Copy, Paste, Clear	
ObjectKind	OutputSingle	

Example

Process: WorkSpaceManager:ClipBoardAction

Parameters : Action = Copy | ObjectKind = OutputSingle

CloseObject process

Description

Close a document, focussed document, project, all documents in a project or all documents in the Altium Designer workspace.

Parameters

Parameter	Value	Description
ObjectKind	All, Document, FocusedDocument, FocusedCategory, ProjectAndDocuments, FocusedProjectAndDocument, FocusedProjectTree, ProjectDocuments, FocusedProjectDocuments, WorkspaceDocuments	

Example

Process: WorkSpaceManager:CloseObject

Parameters : ObjectKind = All

Example 2: Close everything in Altium Designer in a script

Server Process Reference

```
Procedure CloseEverything;
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','All');
    RunProcess('WorkspaceManager:CloseObject');
End;
```

Example 3 Close all the documents within the focussed project tree in a script

```
Procedure CloseProjectTree(ProjectName : String);
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','FocusedProjectTree');
    RunProcess('WorkspaceManager:CloseObject');
End;
```

Example 4 Close a Project

```
AddStringParameter('ObjectKind','Project');
AddStringParameter('FileName', ProjectName);
RunProcess('WorkspaceManager:CloseObject');
```

Compile process

Description

The Compile process is used to compile a document, project or open projects to refresh all the linking information and net information.

Parameters

Parameter	Value	Description
Action	Compile, Recompile	Set Action to Compile to compile the document, project etc only. Otherwise set it to Recompile to refresh all the linking information and net information.
ObjectKind	Document, Project, All, FocusedDocument, FocusedProject	Set ObjectKind to the scope required for the compiling process to act on.

Example

Process: WorkSpaceManager : Compile

Parameters: ObjectKind = Project

Compare process

Description

The Compare process is used to update or import source documents of a PCB project with respect to a reference project, such as comparing an active document against a parent project.

Parameters

Parameter	Value	Description
ObjectKind	Project,FocusedProject	Set ObjectKind to Project or FocusedProject for the compare process.
Action	UpdateOther, UpdateMe	UpdateOther updates other documents with respect to the reference source document, and UpdateMe updates the actual source document.
Index	1..9	Compare to the indexed project.

Example

Process: WorkSpaceManager : Compile

Parameter: Action = UpdateOther

DocumentOptions process

Description

Execute a specified configuration process to update a document or project such as updating search paths, difference level for the ERC, or configure the output jobs facility.

Parameters

Parameter	Value	Description
Action	ViewChannels, ComponentLinking, VariantManagement, OutputJobManagement, Options,ViolationLevels, ViolationMatrix, DifferenceLevels, ChangeOrderLevels, SearchPaths, Parameters, Annotate, Order, AnnotateReset, AnnotateQuiet, AnnotateAll, BackAnnotate, LibrarySynch, DatabaseUpdate	
ObjectKind	Project, FocusedProject, FocusedDocument, Document, OutputSingle, Workspace, Databaselink	Set the ObjectKind
Mode	EditConnection	

Notes

If ObjectKind=DatabaseLink, and you need to change the database connection, assign mode=EditConnection.

Example

Process: WorkSpaceManager:DocumentOptions

Parameters : Action = Project | ObjectKind = AnnotateAll

DeleteObject process

Description

Execute a specified delete action.

Parameters

Parameter	Value	Description
ObjectKind	FocusedDocument, FocusedCategory, Document, Variant, OutputSingle	The FocusedCategory value is to remove all documents in a specified category from a parent project. The Document value is to remove the active document from the parent project. The OutputSingle value removes the specified output generator.

Example

Process: WorkSpaceManager:DeleteObject

Parameters : ObjectKind = FocusedDocument

ExploreTo process

Description

Opens Windows Explorer in the focussed documents, projects or design workspace folder.

Parameters

Parameter	Value	Description
ObjectKind	FocusedDocument,	Set the ObjectKind to the scope for the Explorer to explore in.

Server Process Reference

	FocusedProject, Workspace	

Example

Process: WorkSpaceManager:Explore

Parameters : ObjectKind = Workspace

EmbeddedScanDependencies process

Description

Scan the dependencies for the entire project in Altium Designer.

Parameters

Parameter	Value	Description
ObjectKind	All	

Example

Process: WorkSpaceManager:EmbeddedScanDependencies

Parameters : ObjectKind = All

FPGAPins process

Description

Runs the FPGA signal manager, configure the FPGA component for pin swapping, create a PCB project from a FPGA project or generate a FPGA project from a FPGA component.

Parameters

Parameter	Value	Description
Action	RunManager, RunPinSwapManager, GenerateFromFPGA, GenerateFromPCB	

Example

Process: WorkSpaceManager:FGPAPins

Parameters : Action = RunManager

EmbeddedSelectToolChain process

Description

Select a new tool chain for your FPGA/Embedded project.

Parameters

N/A

Example

Process: WorkSpaceManager:EmbeddedSelectToolChain

EmbeddedOptions process

Description

Edit the embedded option settings for the active or focussed project or document.

Parameters

Parameter	Value	Description
ObjectKind	Project, FocusedProject,	

	FocusedDocument	

Example

Process: WorkSpaceManager:EmbeddedOptions

Parameters : ObjectKind = Project

FPGAWorkspace process

Description

Runs the FPGA workspace map.

Parameters

N/A

GenerateReport process

Description

Generate a specified report in Altium Designer.

Parameters

Parameter	Value	Description
Action	Run,Load	
ObjectKind	Report,Netlist, OutputSingle,OutputBatch,OutputSelected,Fabrication, Assembly,Simulator	To generate a group of reports, specify the value (OutputSingle, OutputBatch or OutputSelected) for the ObjectKind parameter.
Kind	SimpleBOM, BOM, BOM_PartType, Violations, Hierarchy, Designators, Parameters, CrossReference, AdvSimNetlist	
Target	Document	
DoEditProperties	True,False	
DefaultCaption	True,False	
DoGenerate	True,False	
Index	1..n	Choose one of the indexed generators to generate a report.

Notes

- Using the XSPICE simulator, assign the values to the following parameters:

To setup the simulator, ObjectKind=Simulator | Kind = AdvSimNetlist |DoEditProperties = True | DoGenerate = True | DefaultCaption = True.

To run the simulator, ObjectKind=Simulator | Kind = AdvSimNetlist | DoEditProperties = False | Action = Run | DefaultCaption = True.

To generate a XSPICE netlist, ObjectKind=Simulator | Kind = AdvSimNetlist | DoEditProperties = False | Action = Load | DefaultCaption = True.

Example

Process: WorkSpaceManager:GenerateReport

Parameters : Action = Run | ObjectKind = OutputSelected

GenerateSymbol process

Description

Server Process Reference

Generate a schematic symbol in a library from the current document.

Parameters

N/A

See also

WorkSpaceManager processes

HideObject process

Description

Execute this process to hide current document, focused document, project or all documents in Altium Designer workspace.

Parameters

Parameter	Value	Description
ObjectKind	Document,FocusedDocument,Foc usedCategory,FocusedProject, All	

Example

Process: WorkSpaceManager:HideObject

Parameters : ObjectKind = FocusedDocument

InsertObject process

Description

The InsertObject process creates projects from a specified path, add a document to a project or add a free document to a focused project for example.

Parameters

Parameter	Value	Description
ObjectKind	CreateProjectsFromPath, Project, FocusedProject, FocusedDocument, Variant, Fabrication, Documentation, OtherOutput, Assembly, Netlist, Report, OtherOutputOutput, OutputSingle, OutputDuplicate	Fabrication, Documentation, OtherOutput, Assembly, Netlist, Report, OtherOutputOutput values for the ObjectKind parameter, you need to specify the Index value too.
Index	1..20	

Example

Process: WorkspaceManager:InsertObject

Parameter: ObjectKind = Fabrication | Index = 4

Print process

Description

Execute a specified process to invoke the page setup for printing, preview documents, or setup the printer.

Parameters

Parameter	Value	Description
Action	PageSetup, Preview, PrintWithDialog, PrintDocument,	

	PrintSelection, PrinterSetup	
ObjectKind	OutputSingle,FocusedDocument	

Example

Process: WorkSpaceManager:Print

Parameters : Action = Preview

OpenObject process

Description

Open a specific object in Altium Designer such as a document or a project.

Parameters

Parameter	Value	Description
ObjectKind	NewAnything, DocumentList, RecentDocument, RecentProject, RecentProjectGroup, Document, FocusedDocument, FocusedCategory, Variant, Project, FocusedProject, ProjectAndDocuments, FocusedProjectAndDocuments, ProjectDocuments, FocusedProjectDocuments, ProjectGroupDocuments, ProjectGroup, LastProjectGroup, PrimaryDocument, MessageView, MessageViewSelection, CreateProjectsFromPath, All,Assembly, Fabrication, Report, Netlist, Documentation, ,OtherOutput, OutputSingle, OutputBatch, OutputSelected, OutputDuplicate, DatabaseLink, OutputPreference, ControlPanel, FreeDocument, WorkSpaceDocuments, Workspace	If ObjectKind = NewAnything, you need to specify the Kind parameter as well.
OpenMode	Standard, NewFromExisting, NewFromTemplate, Project	
Ext	*.PCBDOC, *.SCHDOC	
DocumentType	PCB, SCH	
Filename	String	
Index	Integer	The position in the most recently used documents list.
Kind	EDIF, EDIFLIB, ProtelNetlist, PCB, SCH, SCHLIB, PCBLIB, PCB3DLIB, C, TEXT, CPP, CUPL, MDL, NSX, CKT, SIMDATA, EditScript, OutputJob, DatabaseLink, VHDL,VHDLIB, VHDMDL, VHDTST VHDL,PcbProject, CoreProject, FpgaProject, EmbeddedProject, IntegratedLibrary, ProjectGroup, CAM, WAVE, IntLibrary, ScriptProject, EditScriptUnit, EditScriptForm	

Example

Process: WorkspaceManager:OpenObject

Parameter: ObjectKind = Project

Example 2: Open a project using the script

```
Procedure OpenProject(ProjectName : String);
```

Server Process Reference

```
Begin
    Result := True;
    If Not FileExists(ProjectName) Then Result := False;
    ResetParameters;
    AddStringParameter('ObjectKind','Project');
    AddStringParameter('FileName', ProjectName);
    RunProcess('WorkspaceManager:OpenObject');
End;
```

Example 3: Open a Devices page in Altium Designer

```
Procedure OpenDevicesPage;
Begin
    ResetParameters;
    AddStringParameter('ObjectKind','ControlPanel');
    AddStringParameter('Index', '1');
    RunProcess('WorkspaceManager:OpenObject');
End;
```

See also

CloseObject process

SaveObject process

SaveObject process

Description

Execute a specified save action to save a document as another document with a different filename, focussed document, all documents in the workspace, or documents of a project for example.

Parameters

Parameter	Value	Description
ObjectKind	Document, FocusedDocument, FocusedCategory, FocusedProject, ProjectDocuments, WorkspaceDocuments, Workspace, All	You can specify the scope of the save action, whether you wish to save a document, documents in a project, or all documents currently open in Altium Designer.
SaveMode	Standard, SaveAs, SaveCopyAs,	When SaveMode is set to Standard, the single document is saved normally. When SaveAs is set, the current document is saved and the filename for this document is replaced with the new filename in the associated project. When SaveCopyAs is set, the document is saved and you are prompted to save a copy of this document with a new filename. This copy is produced in the folder where the original document is.
ModifiedOnly	True, False	If ModifiedOnly parameter is set to true, then only modified documents are saved. If this parameter is set to anything else (empty etc) then all documents etc (depending on the ObjectKind parameter) are saved.
FileFormat	String	

Example 1: Save a document as a different document

Process: WorkSpaceManager:SaveObject

Parameters : ObjectKind = Document | SaveMode = SaveAs

Example 2 : Save all documents from a DelphiScript script

```
Procedure SaveAll;

Begin

    ResetParameters;

    AddStringParameter('ObjectKind','All');

    RunProcess('WorkspaceManager:SaveObject');

End;
```

Example 3 in DelphiScript script

```
Procedure SaveDocument(      Document   : IServerDocument;

                       const SaveFormat : TDynamicString);

Begin

    If Client = Nil Then

       Exit;

    Document.Modified := True;

    Client.ShowDocument(Document);

    Document.Focus;


    ResetParameters;

    AddStringParameter('SaveMode', 'Standard');

    AddStringParameter('ObjectKind', 'Document');

    AddStringParameter('FileFormat', SaveFormat);

    RunProcess('WorkspaceManager:SaveObject');

End;
```

SetupPreferences process

Description

This SetUpPreferences process invokes the Preferences dialog for either Output Job or Database Link documents.

Parameters

Parameter	Value	Description
ObjectKind	OutputPreferences, DatabaseLink	When ObjectKind is set to OutputPreferences, the Output Job Options dialog is invoked for the Outjob File document. A Outjob File has to be open first in Altium Designer for this process to work.

Example

Process: WorkSpaceManager:SetupPreferences

SetCurrentProject process

Description

Make the current project the active project in Altium Designer.

Parameters

N/A

Example

Process: WorkSpaceManager:SetCurrentProject

SetSubProject process

Description

Set the embedded project for the nexus core, or the sub project for the FPGA component.

Parameters

Server Process Reference

Parameter	Value	Description
Target	Core, Component	Specify the Target parameter for the SubProject- Core or FPGA Component.

Example

Process: WorkSpaceManager:SetSubProject

Parameters : Target = Core

ShowHiddenDocument process

Description

Show an indexed hidden document in Altium Designer

Parameters

Parameter	Value	Description
Index	1..16	Show the indexed document in Altium Designer.

Example

Process: WorkSpaceManager:ShowHiddenDocument

Parameters : Index = 1

VersionControl process

Description

Execute a specified action for the version control software application. You can add/remove documents, projects to/from the version control, check in or out documents and projects, check out the history, differences and properties of the version control system.

Parameters

Parameter	Value	Description
Action	Get,Checkout, Checkin, UndoCheckout, History, Differences, Properties, Refresh, Run, Add, Remove	Specify the Action parameter to the value as required.
ObjectKind	FocusedProject, FocusedDocument	Set ObjectKind parameter to FocusedProject or FocusedDocument in Altium Designer with the specified Action parameter that defines the course of action on this object kind.

Example

Process: WorkSpaceManager:VersionControl

Parameters : ObjectKind = FocusedDocument | Action = Refersh

View process

Description

The View process can bring a Panel into view in Altium Designer. Navigator panel (two modes – flattened project or physical hierarchy), Projects panel, Messages panel, Differences panel and Browser panel for example.

Parameters

Parameter	Value	Description
ObjectKind	NewAnything, DocumentList, RecentDocument, RecentProject, RecentProjectGroup, Document, FocusedDocument, Variant, Project, FocusedProject, ProjectAndDocuments, FocusedProjectAndDocuments,	

	ProjectDocuments, FocusedProjectDocuments, ProjectGroupDocuments, ProjectGroup, LastProjectGroup, PrimaryDocument, MessageView, MessageViewSelection, CreateProjectsFromPath, All, Assembly, Fabrication, Report, Netlist, Documentation, OtherOutput, OutputSingle, OutputBatch,OutputSelected, OutputDuplicate, DatabaseLink, OutputPreferences	
ShowError	True, False	Locates the violation in the project.
ShowDifference	True, False	Locates the difference object in the project.
Action	ShowPhysicalHierarchy, ShowLogicalHierarchy, ShowDocuments, ShowLIbraries, ShowFlattened, ShowErrors, ShowDifferences, ToggleBrowser, ToggleProperties	

Server Process Reference

Revision History

Date	Version No.	Revision
01-Dec-2004	1.0	New product release
26-Apr-2005	1.1	Altium Designer
15-Dec-2005	1.2	Updated for Altium Designer 6
21-Apr-2006	1.3	Incorrect parameters updated.
21-Apr-2008	1.4	Updated Page Size to A4
4-Aug-2008	1.5	Updated PCB Document Preferences and Setup processes and added PCB ManageLayerSets Process. Cleaned up many processes. Updated Schematic SetupPreferences. Updated IntegratedLibrary Processes.