

Altium[®]

AltiumLive 2017: ANNUAL PCB DESIGN SUMMIT

Martin Schuster

Boost your Altium Designer
Efficiency with Templates and
Shortcuts

Munich,
25.10.2017



Agenda

- 1 Introduction**
- 2 Shortcuts
- 3 Templates
- 4 Linking 3D Models
- 5 Questions



Martin Schuster
Wolfsburg, Germany



University of Manchester
MEng Electronics System Engineering

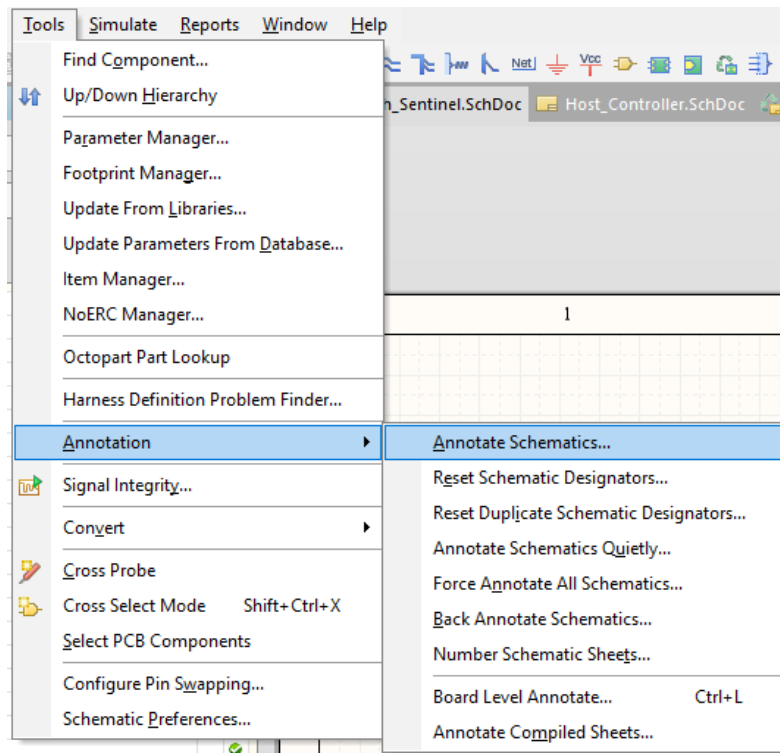


Noa Technologies GmbH
Electronics Engineer

Agenda

- 1 Introduction
- 2 Shortcuts**
- 3 Templates
- 4 Linking 3D Models
- 5 Questions

Why do this....



If you could do this...

T-A-A

D-R

C-O

D-O

T-G-A

P-W

My favorite keyboard shortcuts...

DEMO IN ALTIUM DESIGNER

Best way to learn and remember Altium shortcuts...

- Shortcut reference guide is available in the Altium online documentation or as PDF
- Shortcut menu built-in into Altium Designer as a panel on the bottom right
- Press **F1** or ~ key during an active command to see shortcut options

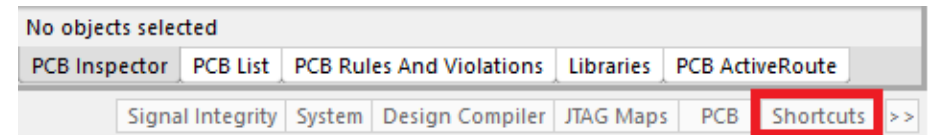
Altium Shortcut Keys

Summary
This guide provides a list of shortcut keys available for use within Altium Designer, ranging from those used at the environment level, through to those associated with particular document editors.

Guide
GU0104 (v2.2) May 22, 2008

Environment Shortcuts

F1	Access Documentation Library (in context with object under cursor)
CTRL + O	Access Choose Document to Open dialog
CTRL + F4	Close active document
CTRL + S	Save current document
CTRL + P	Print current document
ALT + F4	Close Altium Designer
CTRL + TAB	Cycle through open documents (to the right)
SHIFT + CTRL + TAB	Cycle through open documents (to the left)
Drag & drop from Windows Explorer into Altium Designer	Open document as a free document
F4	Hide/display all floating panels
SHIFT + F4	Tile open documents
SHIFT + F5	Toggle between active panel and workspace
SHIFT + F1	Access Natural Language Search facility (Smart Search)
Hold CTRL while moving a panel	Prevent automatic docking, grouping or snapping



Graphical Editing Hot Key List

Shortcut	Action
F1	Help
Tab	Edit Trace Properties
Escape	Suspend
Return	Commit
Backspace	Undo Commit
Ctrl+Click	Autocomplete Segments To Target (Ctrl-Cli
1	Toggle Elbow Side
Space	Look Ahead Mode
Shift+Space	Cycle Corner Style
Shift-R	Toggle Routing Mode
5	Toggle Follow Mouse Trail Mode
Shift+D	Toggle Loop Removal
Ctrl+W	Show Clearance Boundaries
Shift+W	Choose Favorite Width
Shift+V	Choose Favorite Via Size
3	Cycle Track-Width Source
4	Cycle Via-Size Source
Plus	Next Layer
Multiply	Next Layer
Minus	Previous Layer
L	Switch Layer For Current Trace
/	Add Fanout Via and Suspend
2	Add Via (No Layer Change)
6	Change routing Via Start/End layers
7	Next Routing Target
9	Swap To Opposite Route Point
Shift+A	Add Accordions
Shift+G	Toggle Length Gauge

Current Process Help | Close

Agenda

- 1 Introduction
- 2 Shortcuts
- 3 Templates**
- 4 Linking 3D Models
- 5 Questions

You don't want to do the same steps over and over again when setting up a new project?

Tired of duplicate work to make adjustments across multiple sheets and designs?

Templates

Schematic

Draftsman

BOM

Parameters

Component

Project

Document

How to make best use of Templates and Parameters...

DEMO IN ALTIUM DESIGNER

Best way to get started with templates...

- Use the provided standard templates as a starting point:
\Users\Public\Documents\Altium\<AD Version>\Templates
- Add your custom templates to the template folder or create your own template folder
- Share the templates with your colleagues to get a uniform design environment and documentation

A.PCBDOC	17.06.2015 14:00	Altium PCB Document
A.SchDot	03.12.2014 15:52	Altium Schematic Template
A0.PCBDOC	17.06.2015 14:00	Altium PCB Document
A0.SchDot	03.12.2014 15:52	Altium Schematic Template
A0_portrait.SchDot	03.12.2014 15:52	Altium Schematic Template
A1.PCBDOC	17.06.2015 14:00	Altium PCB Document
A1.SchDot	03.12.2014 15:52	Altium Schematic Template
A1_portrait.SchDot	03.12.2014 15:52	Altium Schematic Template
A2.PCBDOC	17.06.2015 14:00	Altium PCB Document
A2.SchDot	03.12.2014 15:52	Altium Schematic Template
A2_portrait.SchDot	03.12.2014 15:52	Altium Schematic Template
A3.PCBDOC	17.06.2015 14:00	Altium PCB Document
A3.SchDot	03.12.2014 15:52	Altium Schematic Template
A3_portrait.SchDot	03.12.2014 15:52	Altium Schematic Template
A4.PCBDOC	17.06.2015 14:00	Altium PCB Document
A4.SchDot	03.12.2014 15:52	Altium Schematic Template
A4_portrait.SchDot	03.12.2014 15:52	Altium Schematic Template

Data Management – Templates

Document Templates

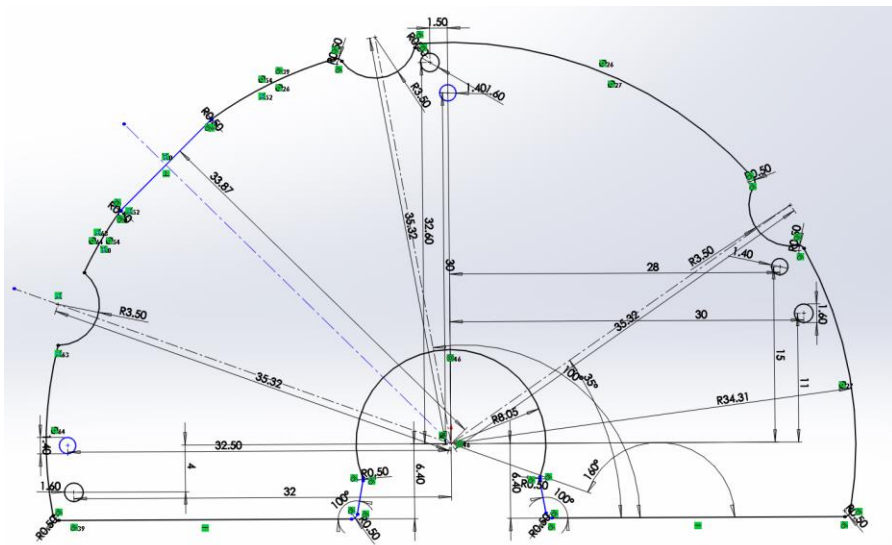
Set the path to the directory containing your document templates. This location is a new project of a particular type.

Template location

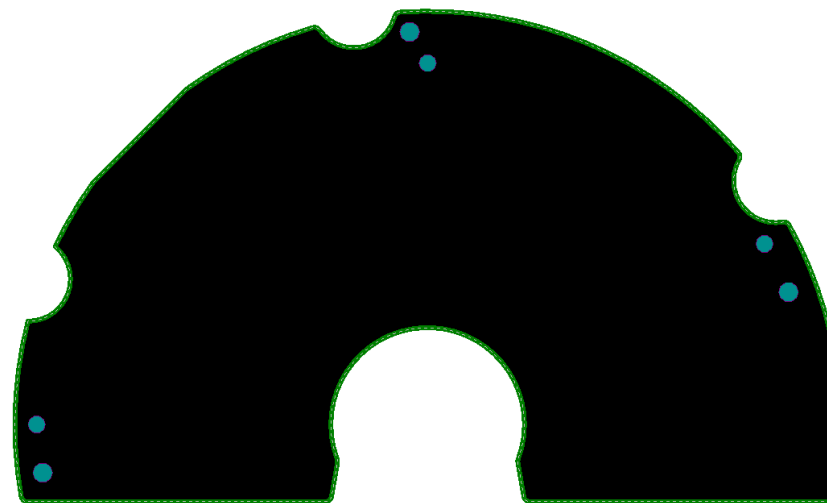
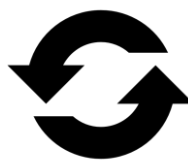
Agenda

- 1 Introduction
- 2 Shortcuts
- 3 Templates
- 4 Linking 3D Models**
- 5 Questions

How do I make sure that my ECAD board shape is matching my MCAD PCB model?



MCAD



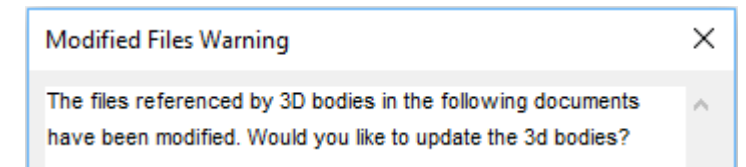
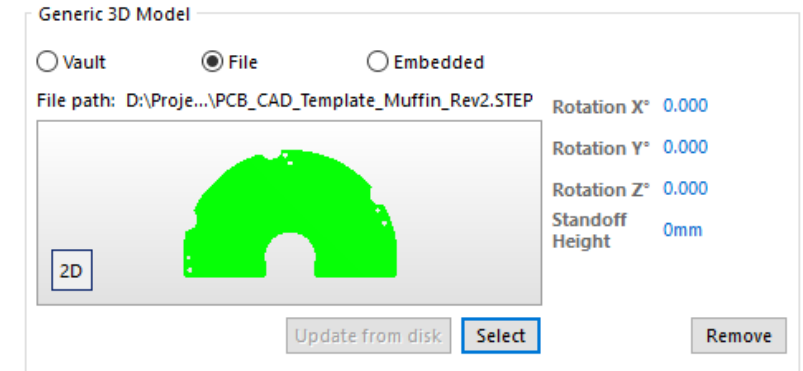
ECAD

Keeping MCAD and ECAD in sync without ECAD/MCAD Collaboration Tool...

DEMO IN ALTIUM DESIGNER

Best way to keep you ECAD board shape in sync with your MCAD PCB model...

- Export MCAD PCB model as STEP file into your PCB step model directory on your computer
- Add PCB step model from 'File' to your PCB document and define board shape from 3D body (D-S-D)
- When changing MCAD model, overwrite the existing step file and Altium automatically detects the change and asks you to update the board shape based on the new step model



Thanks for your Attention!
Questions?