

RF circuits design

Grzegorz Beziuk

Examples of CAE software

Choose yourself and new technologies



Project co-financed from the EU European Social Fund



Wrocław University of Technology

Master programmes in English
at Wrocław University of Technology



References

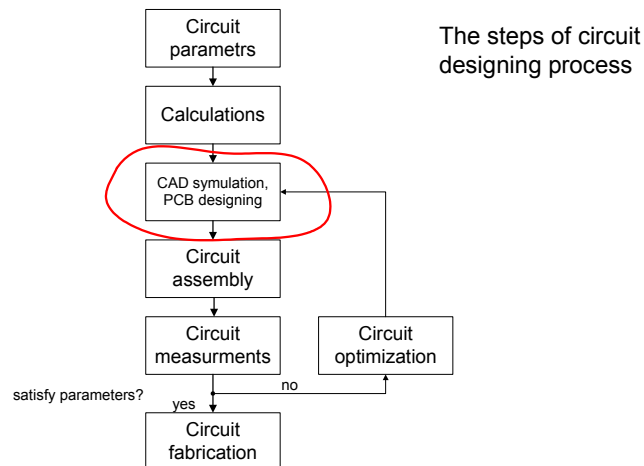
- [1] Ansys Incorporation, [Ansoft High Frequency Structure Simulator - tutorials](http://www.ansoft.com), Ansys, (www.ansoft.com)
- [2] Kraus G., [Ansoft designer SV 2.0 – tutorial fo begginers](http://www.elektronikschule.de/~krausg/). Open source document, 2005, (<http://www.elektronikschule.de/~krausg/>)
- [3] Altium limited, [Altium designer tutorial – getting started with PCB design](http://wiki.altium.com/display/ADOH/Getting+Started+with+Altium+Designer), Altium limited, (<http://wiki.altium.com/display/ADOH/Getting+Started+with+Altium+Designer>)
- [4] Altium limited, [Denining and running simulation analises](http://wiki.altium.com/display/ADOH/Getting+Started+with+Altium+Designer), Altium limited, (<http://wiki.altium.com/display/ADOH/Getting+Started+with+Altium+Designer>)



Project co-financed from the EU European Social Fund



Introduction



Introduction

Circuits symulation software:

Pspice, Orcad, Multisim (free AD), Altium Designer, Tina (free TI), SmartSpice, Hspice, T-Spice, Spectre (RF), Eldo (RF), UltraSim, LT Spice (free LT), NanoSim, Nspice, Hsim, B2Spice, ICAD/4, EDSpice, WINecad, TopSpice, Spice Opus, SiMetrix, Micro-cap

Circuits and EM simulation (RF and microwave): Microwave Office (RF), Ansoft Designer (RF), Sonnet Lite (RF, EMC), Agilent ADS





Introduction

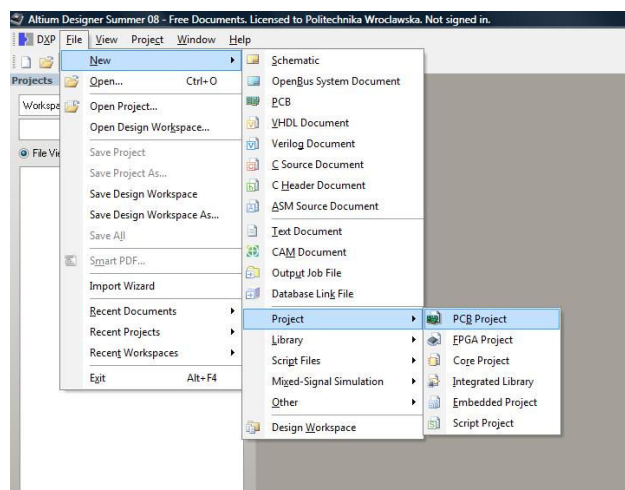
PCB designing software:

Altium Designer (Easytrax, Autotrax, Protel), Eagle, Spectra i Allegro (autorouting), CadStar, Orcad, Tina, Circuit Maker, P-Cad, PCB Elegance, EDWin, VisualPC, BPECS32, Expert PCB, CirCAD, Layout, McCAD, EPD (RF, hybrid), gEDA (free – linux), ZenitPCB (free), PCB (free), KiCAD (free)

Some software contains circuit simulation module and PCB designing module, for instance Altium Designer.



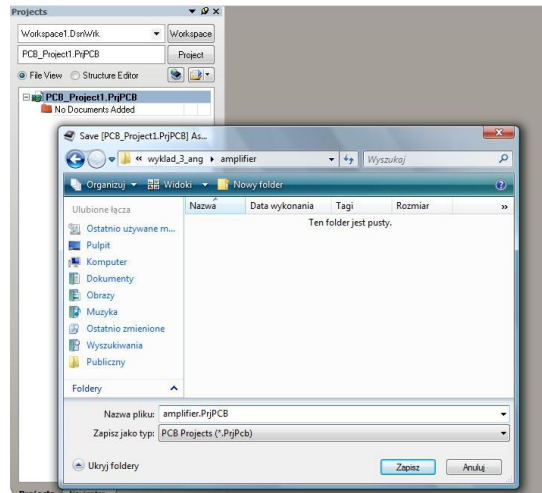
Altium designer – circuits symulation



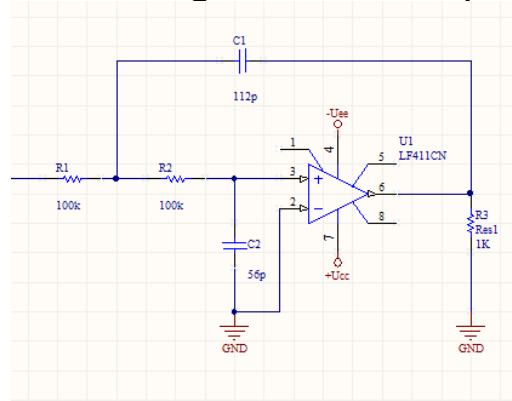
Create on your
PC folder „....”



Altium designer – circuits symulation



Altium designer – circuits symulation



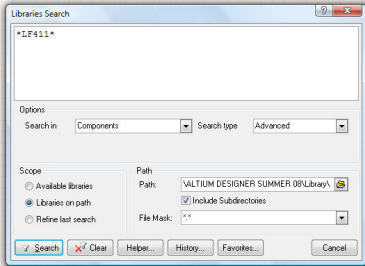


Wrocław University of Technology

Master programmes in English
at Wrocław University of Technology



Altium designer – circuits simulation



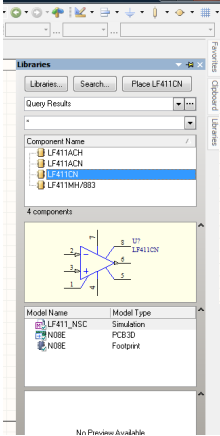
Libraries Search dialog box showing search criteria for "LF411".

Options: Search in: Components, Search type: Advanced

Scope: Available libraries, Libraries on path, Refine last search

Path: Path: ALTUM DESIGNER SUMMER 08/Library, Include Subdirectories, File Mask: *

Search, Clear, Help, History, Favorites, Cancel



Libraries panel showing search results for "LF411".

Libraries: Search, Place LF411CN

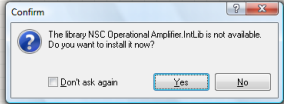
Query Results: LF411ACH, LF411ACN, LF411CN, LF411MU883

4 components

Model Name: LF411_NSC, N08E, N08E, N08E

Model Type: Simulation, PCB 3D, Footprint

No Preview Available



Confirm dialog box: The library NSC Operational Amplifier IntLib is not available. Do you want to install it now?

Don't ask again, Yes, No



Project co-financed from the EU European Social Fund

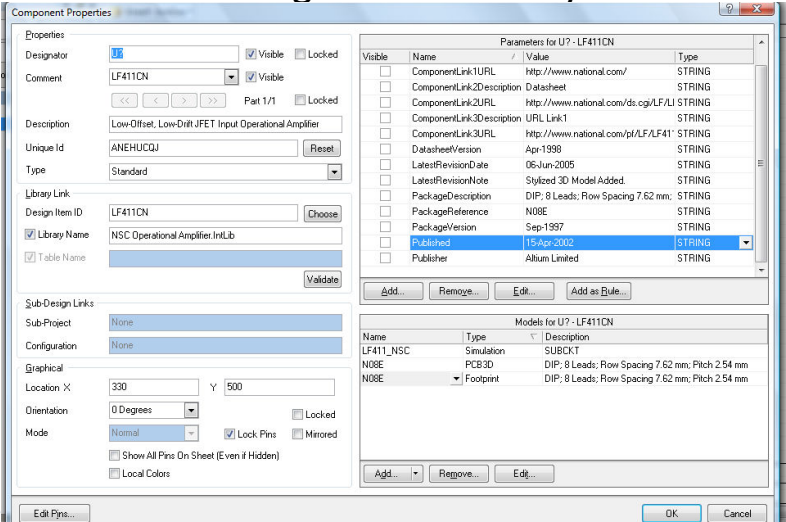


Wrocław University of Technology

Master programmes in English
at Wrocław University of Technology



Altium designer – circuits simulation



Component Properties dialog box for LF411CN.

Properties: Designator: LF411CN, Comment: LF411CN, Description: Low-Offset, Low-Drift JFET Input Operational Amplifier, Unique Id: ANEHUCQJ, Type: Standard

Library Link: Design Item ID: LF411CN, Library Name: NSC Operational Amplifier IntLib, Table Name: [empty]

Sub-Design Links: Sub-Project: None, Configuration: None

Graphical: Location: X: 330, Y: 500, Orientation: 0 Degrees, Mode: Normal, Lock Pins: [checked], Show All Pins On Sheet (Even if Hidden): [checked], Local Colors: [checked]

Parameters for U7 - LF411CN

Visible	Name	Value	Type
<input type="checkbox"/>	ComponentLink1URL	http://www.national.com/	STRING
<input type="checkbox"/>	ComponentLink2Description	Datasheet	STRING
<input type="checkbox"/>	ComponentLink2URL	http://www.national.com/ds/cg/LF/LF411	STRING
<input type="checkbox"/>	ComponentLink3Description	URL Link1	STRING
<input type="checkbox"/>	ComponentLink3URL	http://www.national.com/pt/LF/LF411	STRING
<input type="checkbox"/>	DatasheetVersion	Apr-1998	STRING
<input type="checkbox"/>	LatestRevisionDate	06-Jun-2005	STRING
<input type="checkbox"/>	LatestRevisionNote	Stylized 3D Model Added	STRING
<input type="checkbox"/>	PackageDescription	DIP: 8 Leads; Row Spacing 7.62 mm;	STRING
<input type="checkbox"/>	PackageReference	N08E	STRING
<input type="checkbox"/>	PackageVersion	Sep-1997	STRING
<input type="checkbox"/>	Published	15-Apr-2002	STRING
<input type="checkbox"/>	Publisher	Altium Limited	STRING

Models for U7 - LF411CN

Name	Type	Description
LF411_NSC	Simulation	SUBCKT
N08E	PCB 3D	DIP: 8 Leads; Row Spacing 7.62 mm; Pitch 2.54 mm
N08E	Footprint	DIP: 8 Leads; Row Spacing 7.62 mm; Pitch 2.54 mm

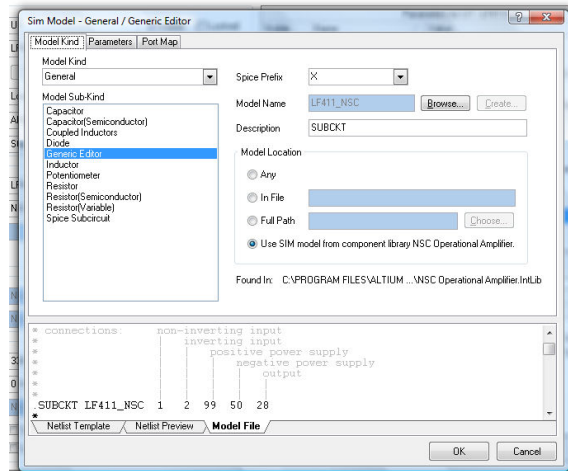
Edit Pins..., OK, Cancel



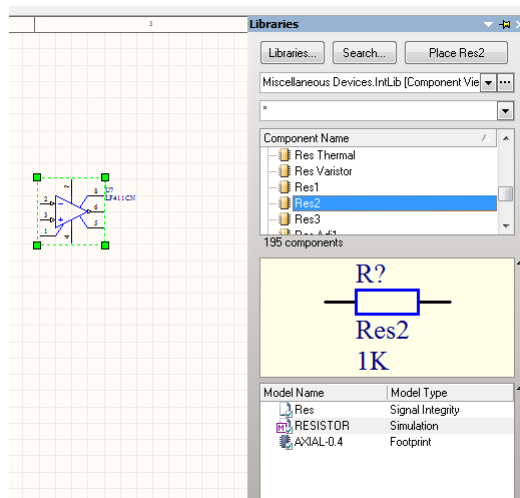
Project co-financed from the EU European Social Fund



Altium designer – circuits symulation

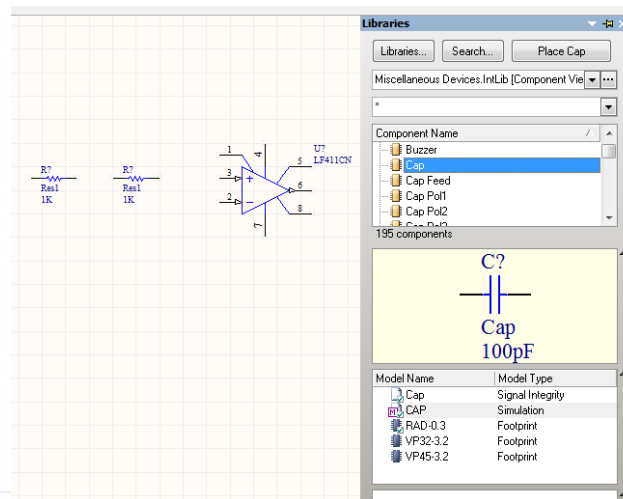


Altium designer – circuits symulation

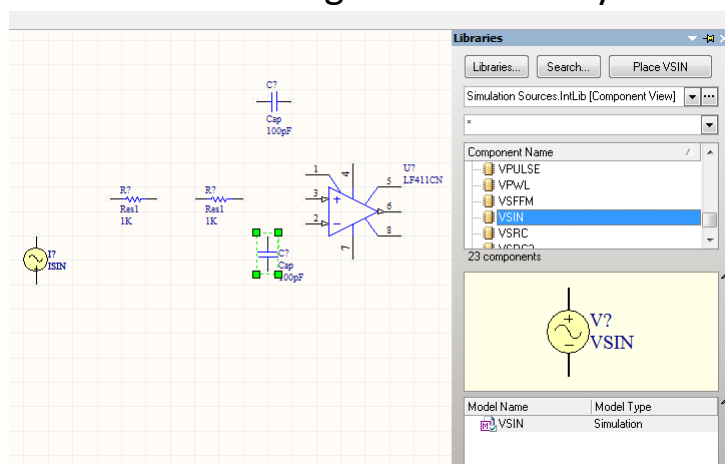




Altium designer – circuits symulation

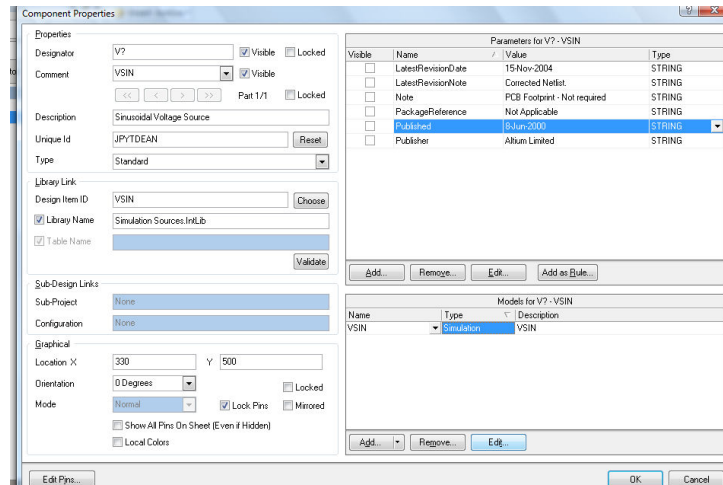


Altium designer – circuits symulation

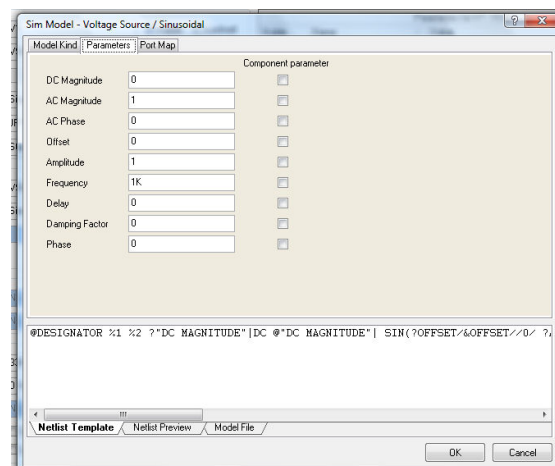




Altium designer – circuits symulation



Altium designer – circuits symulation



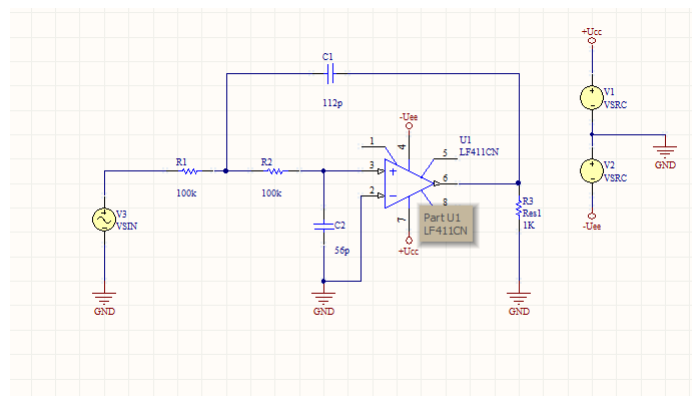


Altium designer – circuits symulation

- add power sources VSRC, set up volatge values required for +Ucc, -Uee
- add ports (from menu – Vcc power ports) : GND, +Ucc, - Uee
- add load resistance (R3)
- wiring up the circuits
- annotate parts
- compiling the project

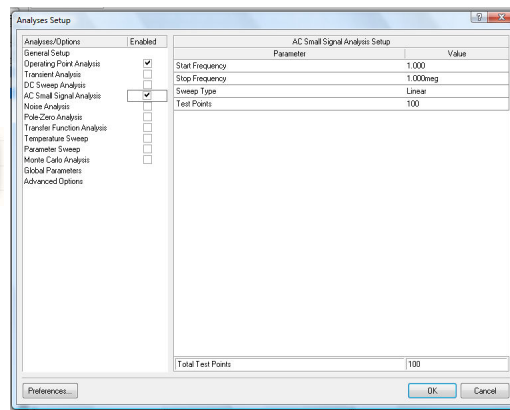
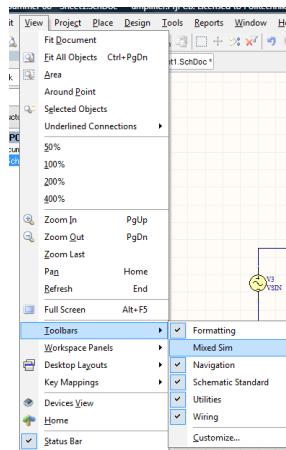


Altium designer – circuits symulation

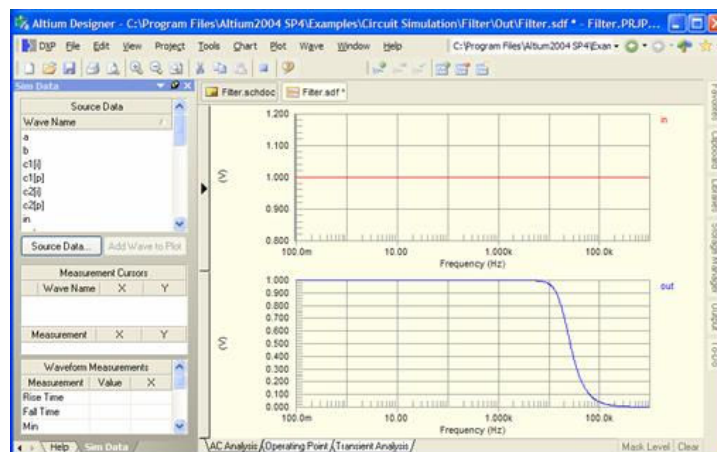




Altium designer – circuits simulation



Altium designer – circuits simulation



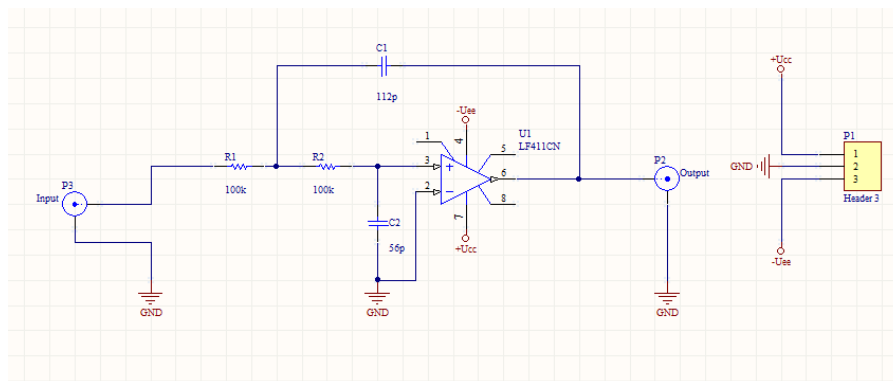


Altium designer – PCB designing

- add to existing project PCB
- save PCB as „.....”
- replace in the circuit all sources by appropriate ports,
- once again annotate schematic (function – update changings list)
- check whether all parts have defined footprints
- then use function – „Design/Update PCB document
***.PcbDoc”
- define outline of Pcb (by means of Keep out layer) and set origin

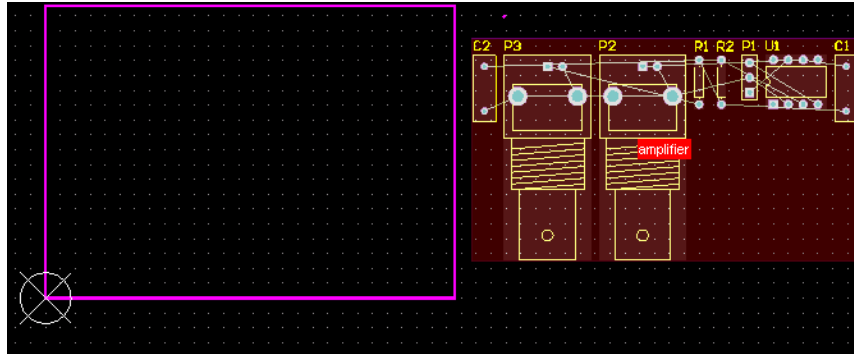


Altium designer – PCB designing

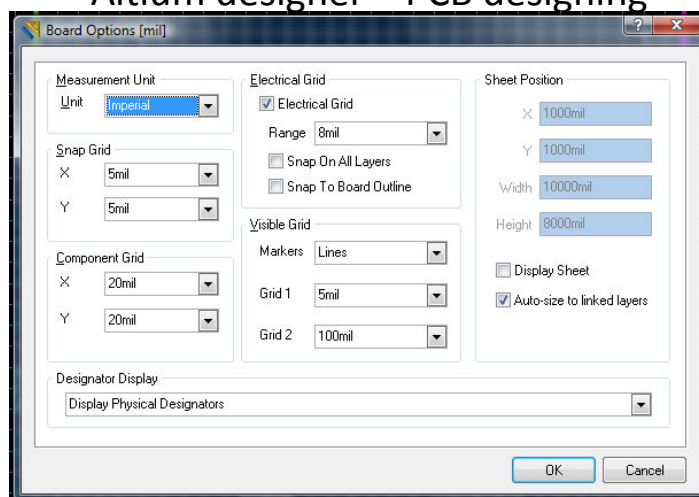




Altium designer – PCB designing

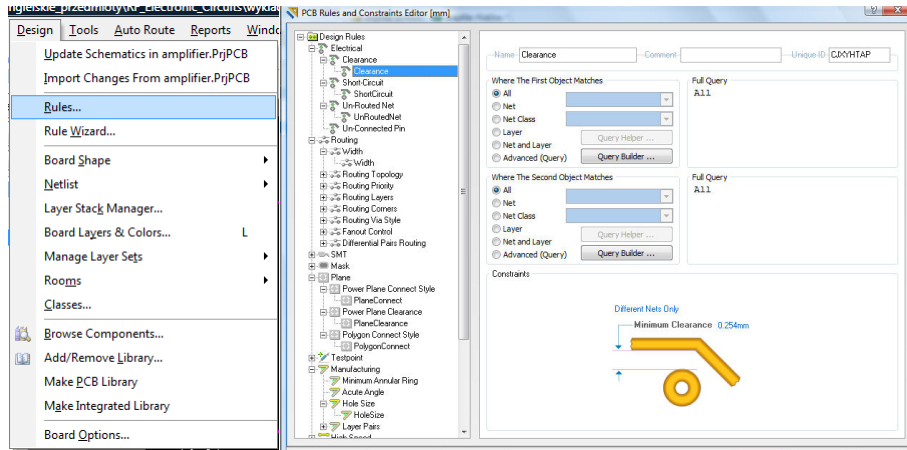


Altium designer – PCB designing

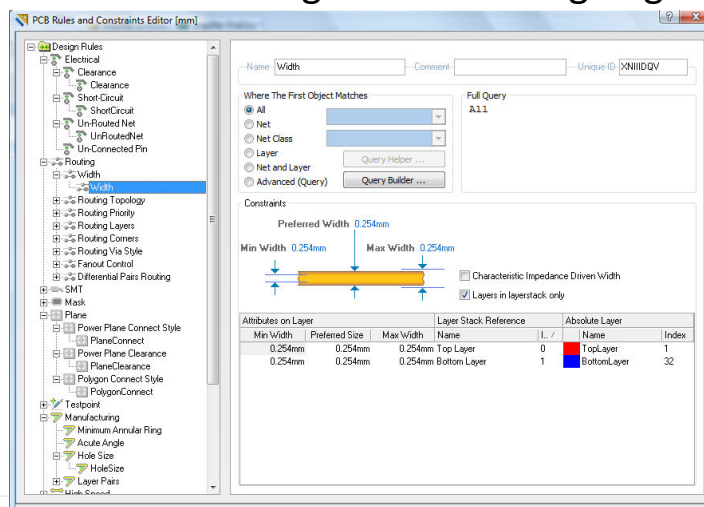




Altium designer – PCB designing



Altium designer – PCB designing



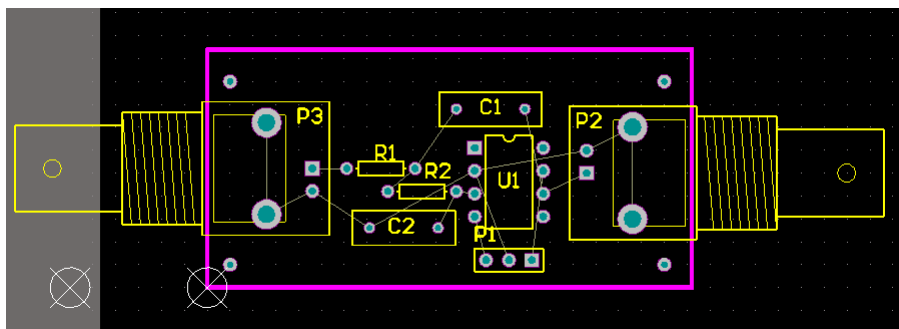


Altium designer – PCB designing

- set up all rules
- place all components
- define new size of Pcb outline
- define Pcb clamping points
- connect components (one layer board – bottom/top layer, two layers board top and bottom layer)

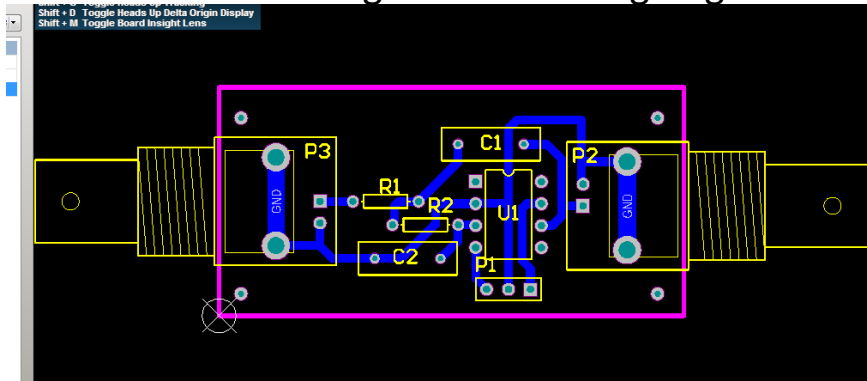


Altium designer – PCB designing

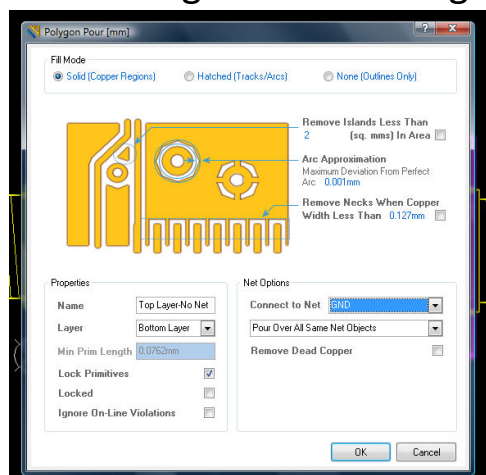




Altium designer – PCB designing

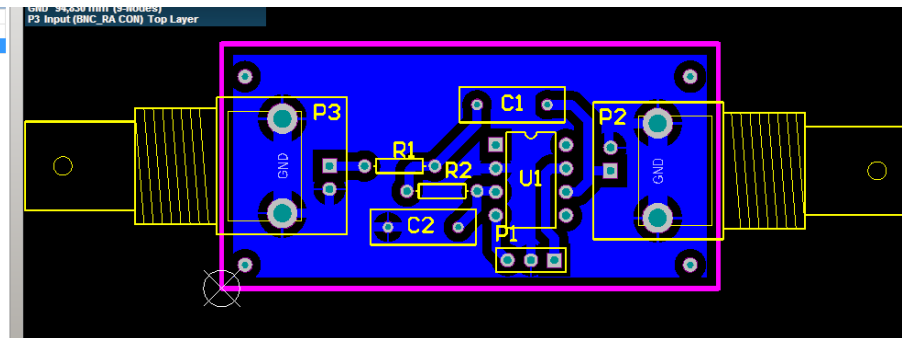


Altium designer – PCB designing

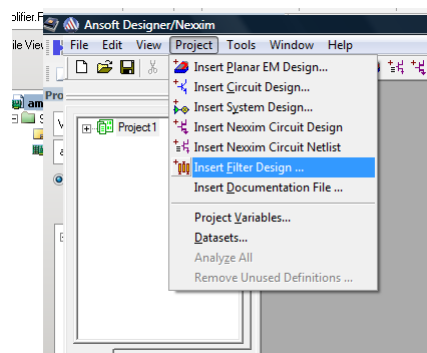
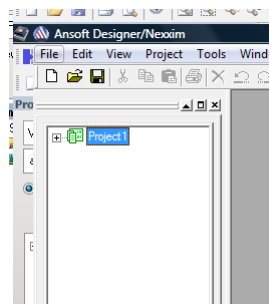




Altium designer – PCB designing

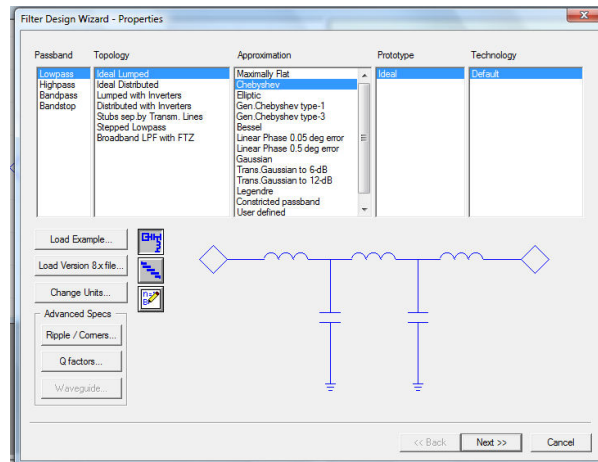


Ansoft Designer – filter project

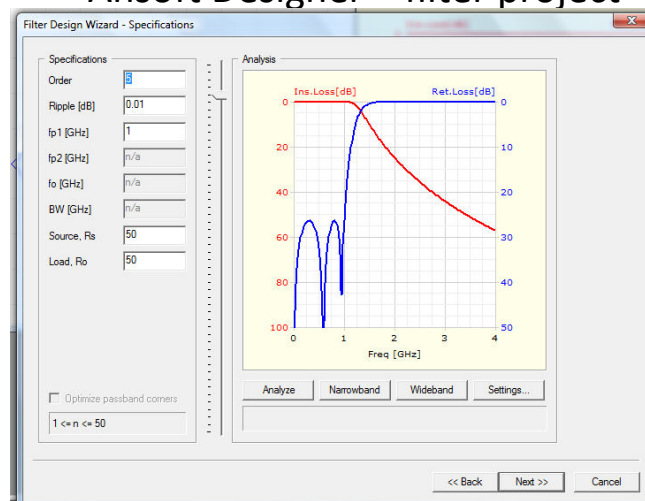




Ansoft Designer – filter project

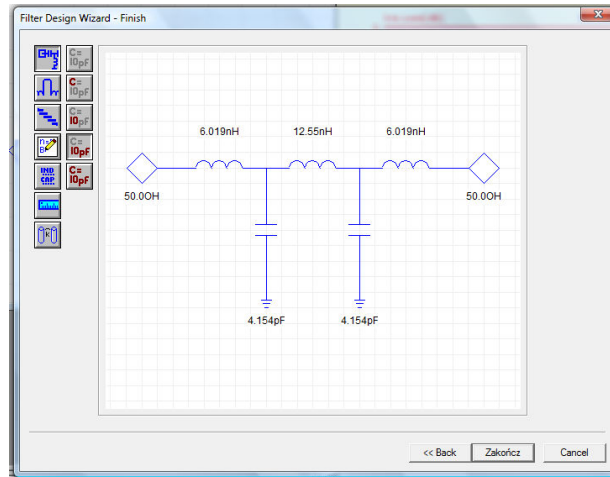


Ansoft Designer – filter project

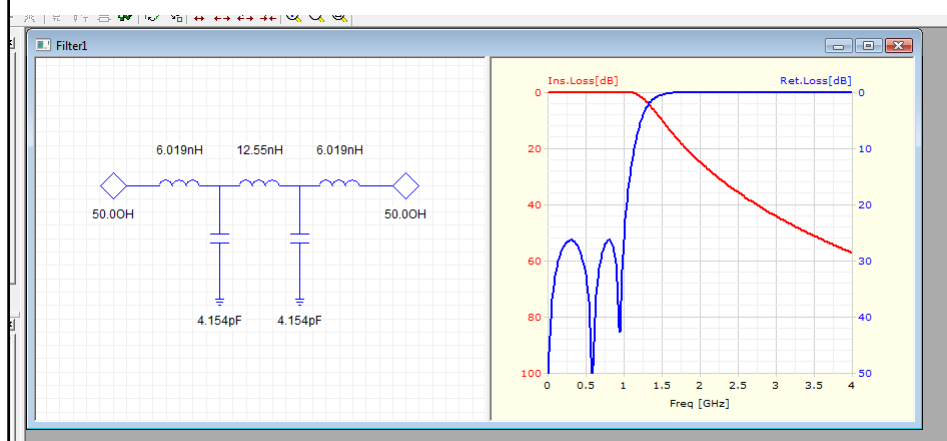




Ansoft Designer – filter project

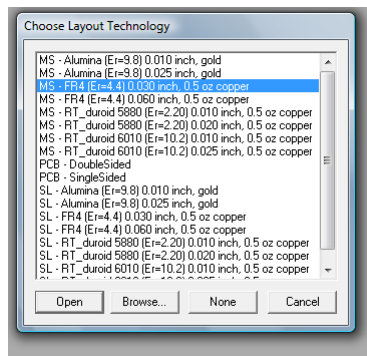
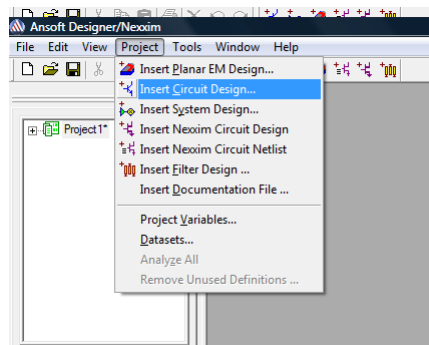


Ansoft Designer – filter project

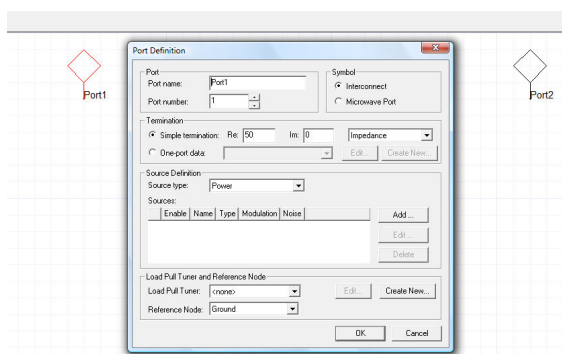
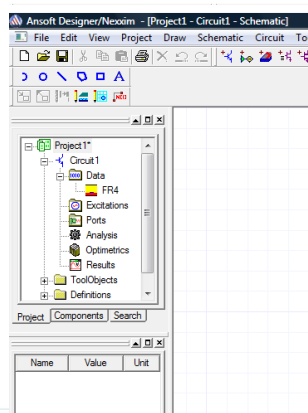




Ansoft Designer – Impedance Matching using a $1/4$ –Microstrip-Line (on a FR4-Board)

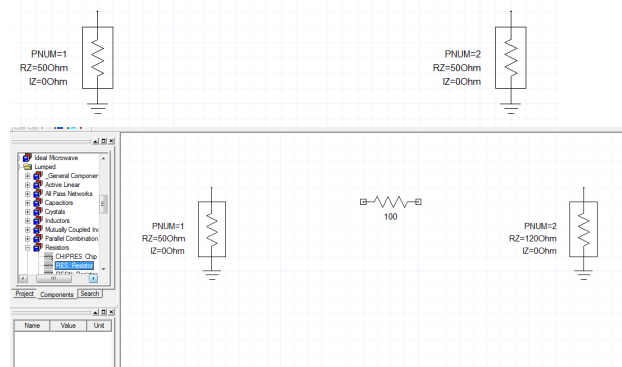


Ansoft Designer – Impedance Matching using a $1/4$ –Microstrip-Line (on a FR4-Board)

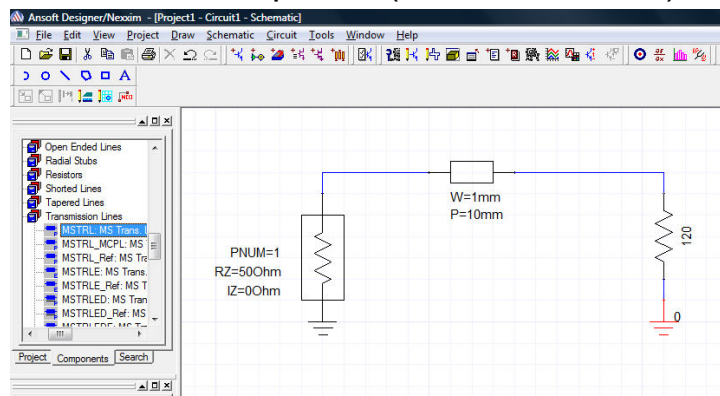




Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)

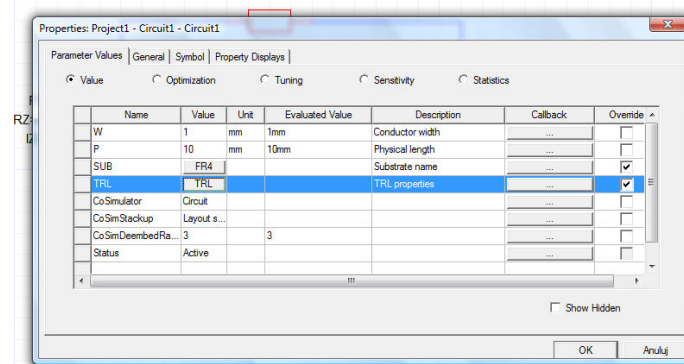


Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)

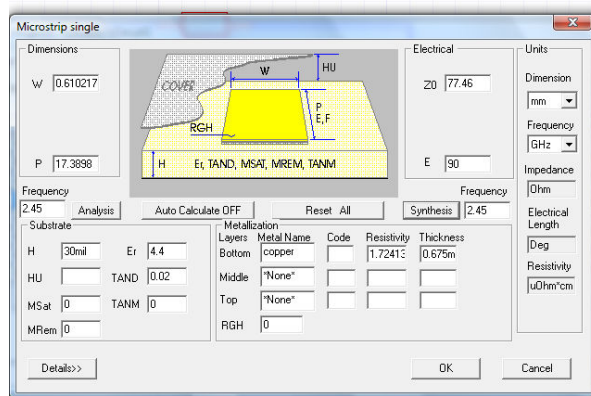




Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)



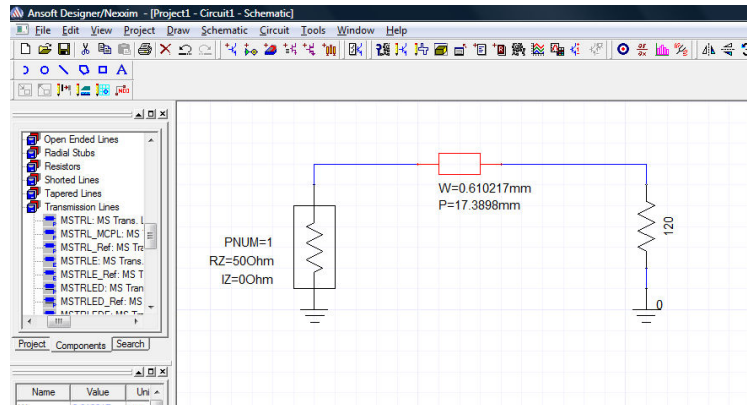
Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)



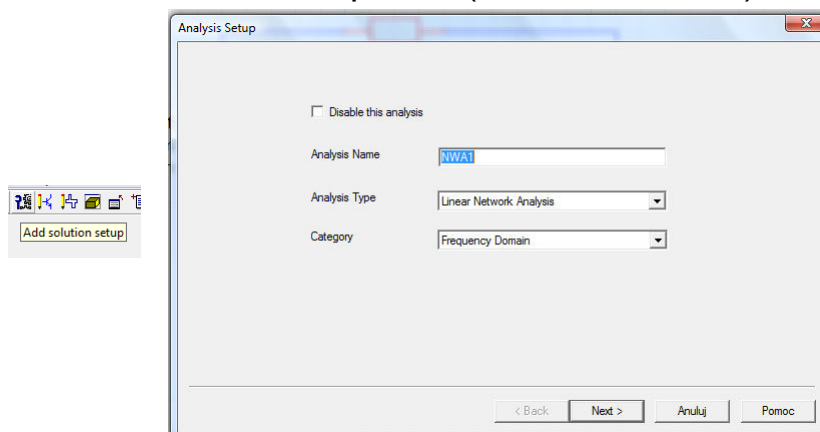
$$Z_0 = \sqrt{50 \cdot 120} = 77.46 \Omega$$



Ansoft Designer – Impedance Matching using a $\frac{1}{4}$ –Microstrip-Line (on a FR4-Board)

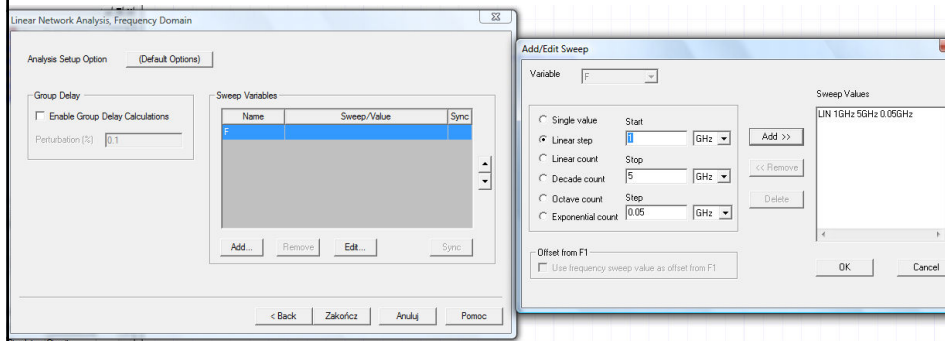


Ansoft Designer – Impedance Matching using a $\frac{1}{4}$ –Microstrip-Line (on a FR4-Board)

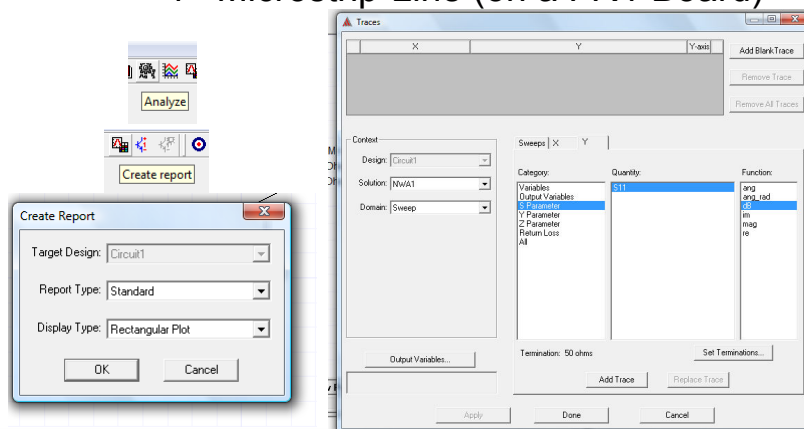




Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)

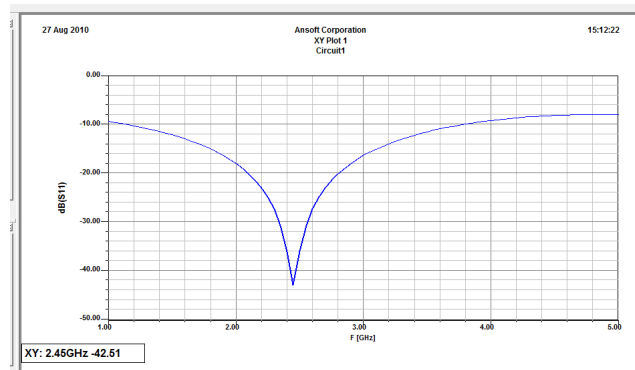


Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)





Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)



Ansoft Designer – Impedance Matching using a 1 / 4 –Microstrip-Line (on a FR4-Board)

