Advanced Design System (ADS) Tutorial:

ADS is a simulator like spice, cadence. But it focuses on the RF and microwave design, so most of its devices on the library are microwave devices.

Circuit Simulation:

Here are some of ADS analysis:

–DC analysis: is used for determining the bias point of the circuit.

—Transient analysis: runs the time domain analysis on the circuits and considers the nonlinearity of the elements.

–AC analysis: runs the small signal analysis and use the linear model of elements on their bias point. So the nonlinear elements like transistor are replaced by a linear model (small signal circuit) which includes resistors, capacitors, inductors and voltage & current sources.

–S_parameters analysis: calculates the Scattering parameters of the components, and shows the variation of the S_parameters over different frequencies. It is also used for calculating noise figure and group delay.

Starting ADS:

All design work must be done in a project directory. Working in project directories enables you to organize related files within a predetermined file structure. This predetermined file structure consists of a set of subdirectories. These subdirectories are used in the following manner:

- *networks* contains schematic and layout information, as well as information needed for simulating
- *data* is the default directory location for input and output data files used or generated by the simulator
- *mom_dsn* contains designs created with the Agilent EEsof planar electromagnetic simulator, Momentum
- synthesis contains designs created with DSP filter and synthesis tools
- *verification* contains files generated by the Design Rule Checker (DRC), used with Layout

🖬 Advanced Design System 2009 Update 1 (Main) 📃 🗖							
<u>F</u> ile <u>V</u> iew <u>T</u> ools <u>W</u> indow Design <u>K</u> it DesignGuide <u>H</u> elp							
] 🗂 🔗 📅 🛥 🗞 🌠 🔽 🔝 🖹 📟 🐜 🌆							
File View Project View							
File Browser	Project Hierarchy						
Example 1_prj 1							

Creating a new project:

-Choose **File**> **New Project.** A dialog box appears and asking about the work directory that by default

is set to your start up directory.

-Provide a name and then press OK. ADS add **_prj** at the end of project name.

-A schematic window is displayed automatically.

Opening an existing project:

Choose **File**> **Open Project.** After a project is opened its name and path are displayed. In the **Project View** tab you see that the schematics files have an extension of **.dsn** and the data or simulation result files have an extension of **.dds** . If double click on each of these files, it will be opened.

If you want to open a new schematic, choose **Windows>New Schematic** or click on **New Schematic** icon.

Advanced Design System 2009 Update 1 (Main)	
<u>F</u> ile <u>V</u> iew <u>T</u> ools <u>W</u> indow Design <u>K</u> it DesignGuide <u>H</u> elp	
] 🗂 🔗 📅 🔽 💼 🔚 🐂 🐂 👘	
File View Project Vie New Schematic Window	1
🖻 🃅 //EX1_prj	
🛄 DA_MLine_untitled1.dsn	
Pl.dsn	
w wo Fixed Freq Osc dds	
vto Fixed Freq Osc. dsn	
w you kee required be ability	
	//

After opening a **New Schematic** window, a dialog window appears that provides help with circuit or simulation settings, you can also choose ''**No help needed**''.

Creating Schematic:

Before starting you can set the units from **Options>Preferences>Units/Scale.** Components can be selected either form **Component Palette** or **Component library**.

Palette list:

The Palette list is on the left side of the schematic window. In the Palette list you see all the libraries with their components. Select one component and move the cursor in the drawing area, you will see a ghost image. Place it wherever you like and press **Esc** to end it.

Component library:

For selecting components from Component library choose **Insert>Component>Component Library**, then a dialog box appears which list the libraries with their components. After selecting a component, drag it into the schematic window then a ghost image moves along the cursor. Place the component wherever you want, press **Esc** to end the command.

Component Library						
Analog/RF Libraries	Search	Search				
⊕ Analog Parts Library (No Layout)	Component /	Description —				
Data Items	BIT_NPN	Nonlinear Bipolar Transistor, NPN				
Devices-BJT Devices-Diodes	BJT_PNP	Nonlinear Bipolar Transistor, PNP				
- Devices-GaAs	BJT4_NPN	Nonlinear Bipolar Transistor w/ Substrate Te				
Devices-JFET	BJT4_PNP	Nonlinear Bipolar Transistor w/ Substrate Te				
Devices-Linear	STRIT Model	ST B IT Model				
Eqn Based-Linear	EE_BJT2_Model	EE Bipolar Transistor Model				
- Eqn Based-Nonlinear	VBIC_NPN	VBIC Nonlinear Bipolar Transistor, NPN				
Filters-Bandstop	VBIC_PNP	VBIC Nonlinear Bipolar Transistor, PNP				
Filters-Highpass	VBIC5_NPN	VBIC Nonlinear Bipolar Transistor w/ Therm				
Download Libraries	VBIC5_PNP	VBIC Nonlinear Bipolar Transistor w/ Therm ▼ ▶				

For rotating a component use **Ctrl+r** or use **Edit>Rotate**

For copying a component use Ctrl+**c** or use **Edit>copy**

For pasting a component use Ctrl+**p** or use **Edit>paste**

For editing a component double click on it, then a window appears with all the parameters of the component.

For connecting the components click on Insert Wire icon.

For labeling the wires click on **Name** icon. After choosing a desired name click on the desired wire to label it.



Simulation/Example_1:

Draw the circuit which is shown in the following figure:



-From Palett choose:

-Lumped_Componenet library, then select **R**.

-Sources_Time Domain library, then select Sine source

-Tlines_Microstrip, then select MLIN and MSUB components.

MSUB defines the properties of the microstrip lines, for this simulation we use RO4350B .

RO4000[®] Series High Frequency Circuit Materials are glass reinforced hydrocarbon/ceramic laminates (Not PTFE) designed for performance sensitive, high volume commercial applications.

Double click on **MSUB** and set the following parameters:

– H	1.52	mm	Substrate thickness
– Er	3.48		Relative dielectric constant
-Cond		5.8 e+07 S/m	Conductor conductivity. (Copper)
– T	0.06	mm	Conductor thickness
–TanD	0.004		Dielectric loss tangent

-Simulation_DC, select DC

-Simulation_Transient, select Trans

-Edit the components according to given parameters in pervious figure.

-For calculating the 'W' and 'L' of the MLIN, use transmission line calculator (LineCalc).

Choose **Tools>LineCalc>StartLineCalc** then a dialog box appears.

-Fill out the Substrate parameters as given above.

-Set Z0 is the characteristic impedance of the transmission line. E_Eff is Effective electrical length of line in degree. **0.083*lamda** is 30 degrees and **0.166*lamda** is 60 degrees.

-Set Freq to 5 GHz

-After setting all the parameters press 'Synthesize' to calculate the 'W' and 'L'

] LineCalc/untitled						
<u>S</u> imulation <u>O</u> ptions	<u>H</u> elp					
) 🗁 🗯 🎒 🛛						
omponent						
/pe MLIN	ID MLIN:	MLIN_DEFAULT	-			
Substrate Parameters-					<u>∧</u> 2	
ID MSUB_DEFAUL	LT					
Er	3.480	N/A. 👻				
Mur	1.000	N/A. 👻	- Physical-			
н	1.520	mm 💌	w	1.626970 mm 🔻		
Hu	3.9e+34	mil 💌	L	3.095610 mm 🔻	Calculated Results	
т	0.060	mm 💌		N/A 🔻	A_DB = 0.009	
Cond	5.8e7	N/A 👻		N/A 🔻	SkinDepth = 0.036	
TanD	0.004	N/A.	Synthesize	Analyze		
Rough	0.000	mil 💌				
DielectricLossModel	1.000	N/A. 👻	- Electrical-			
FreqForEpsrTanD	1.0e9	N/A.	ZO	75.000 Ohm 🔻		
LowFreqForTanD	1.0e3	N/A.	E_Eff	30.000 deg 🔻		
HighFreqForTanD	1.0e12	N/A.		N/A 🔽		
		N/A. 👻		N/A 🗸		
Component Parameter	c	'.		N/A 👻		
Freq.	- 000	GHz V		, , _		
walli	5.000					
wallz						
lues are consistent						,

-Save the design and click on gear icon to simulate the circuit.

🖹 [Example1_prj] TransmissionLine * (Schematic):14 (on everlybrothers.ifi.uio.no)					
<u>F</u> ile <u>E</u> dit <u>S</u> elect <u>V</u> iew Insert <u>O</u> ptions <u>T</u> ools <u>L</u> ayout Si <u>m</u> ulate <u>W</u> indow DynamicLink DesignGuide <u>H</u> elp					
🗅 🗁 🏩 🖕 🚧 🛍 🌐 📩 🛫 🖳 🔹 💠 🏛 🕰 🏪 🍘 🎇 🕱 🌂					
Simulation-Transient 💽 GROUND 💽 🖓 🗄 🕅 🖓 👘 🏠 🔤 🛄					
Palette 🗗				Sim	ulate
🕲 🖻 🗕 🗄 🔆 Msub 📜 🗄	Node12	1 Node2	1 2 Nod	e3 <u>1 2</u> Sim	ulates the current design.
Itans Uptions · <th·< th=""> · <th·< td=""><td>VtSine R2 SRC1 R2</td><td></td><td>MLIN TL1</td><td>MLIN TL2</td><td>R1 R=100 Ohm</td></th·<></th·<>	VtSine R2 SRC1 R2		MLIN TL1	MLIN TL2	R1 R=100 Ohm
Plan PrmSwp E=3.48	Vdc=1 V Amplitude=0.5 V	••••••	Subst="MSub1" W=1.62 mm	Subst="MSub1". W=3.42 mm	
Int C IntCd	Delay=0 nsec		L-3 min		₩
Image: NdSet	Phase=0	· · · · · ·	· · · · · · · · ·	· · · · · · · · · · · ·	· · · · · · · · · · · ·
Name Rough=0 mi	· · · · · · · · · ·	· · · · · ·	· · · · · · · · ·	· · · · · · · · · · · ·	
		· · · · ·	· · · · · · · · ·	· · · · · · · · · · · ·	· · · · · · · · · · · ·
		· · · · ·	· · · · · · · · ·	· · · · · · · · · · · ·	· · · · · · · · · · · ·
Train 1	DC1			· · · · · · · · · · · ·	
IfoTran IspecTm				· · · · · · · · · · · · ·	· · · · · · · · · · · · · · · · · · ·
					· · · · · · · · · · · · · · · · · · ·
Select: Enter the starting point	0 items	wire	8.0000, 2.1250	1.5000, 4.3750 in	A/RF SimSchem

After simulation finishes, the **Data Display** window opens. From palette in **Data Display** window you can choose different charts to display simulation results.

-From Palette choose the **Rectangular Plot.** A **Plot Traces & Attributes** window pops up, from **plot**

type tab select the desired parameters to be plotted and **Add** them, then press **OK**.



Here is a plot of simulated results. As can be seen the phase difference between voltage of node1 and voltage of node4 is 30+60 =90 degrees.



In the schematic window, if you choose **Simulate>Annotate DC Solution** then DC current of all branches and DC voltage of all nodes are displayed on the circuit, if choose **Simulate>Clear DC Annotation** they will be erased.

Here you can find Advanced Design System 2009 Documentation:

http://edocs.soco.agilent.com/display/ads2009/Home