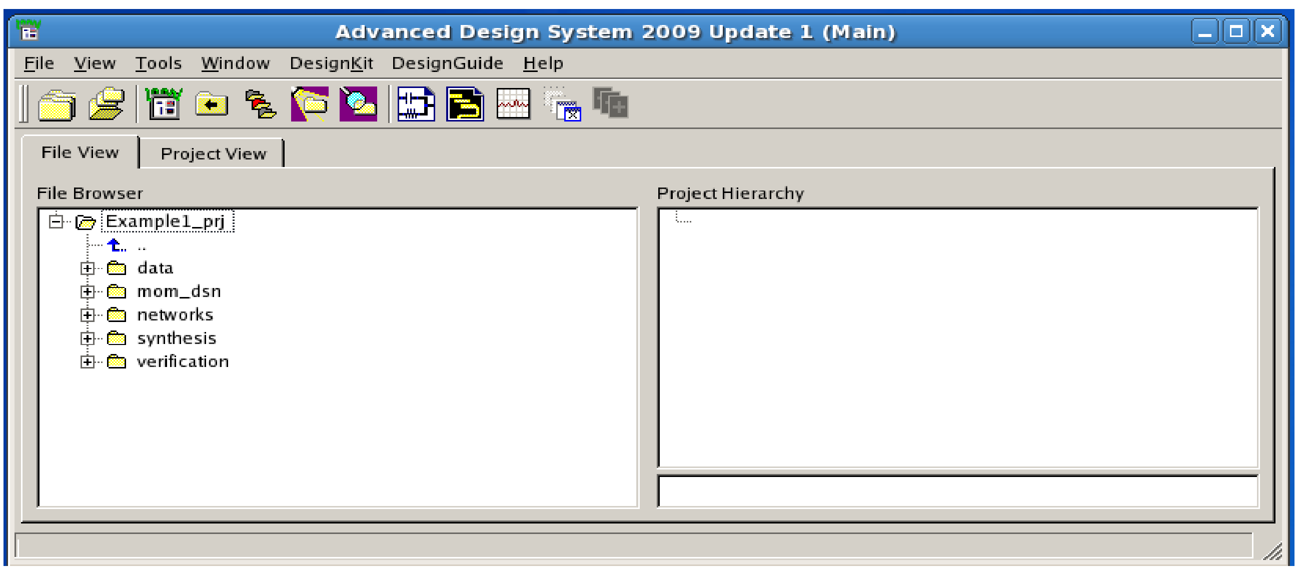
Advanced Design System (ADS) Tutorial:
ADS is a simulator like spice, cadence. But it focuses on the RF and microwave design, so most of its devices on the library are microwave devices.

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



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.

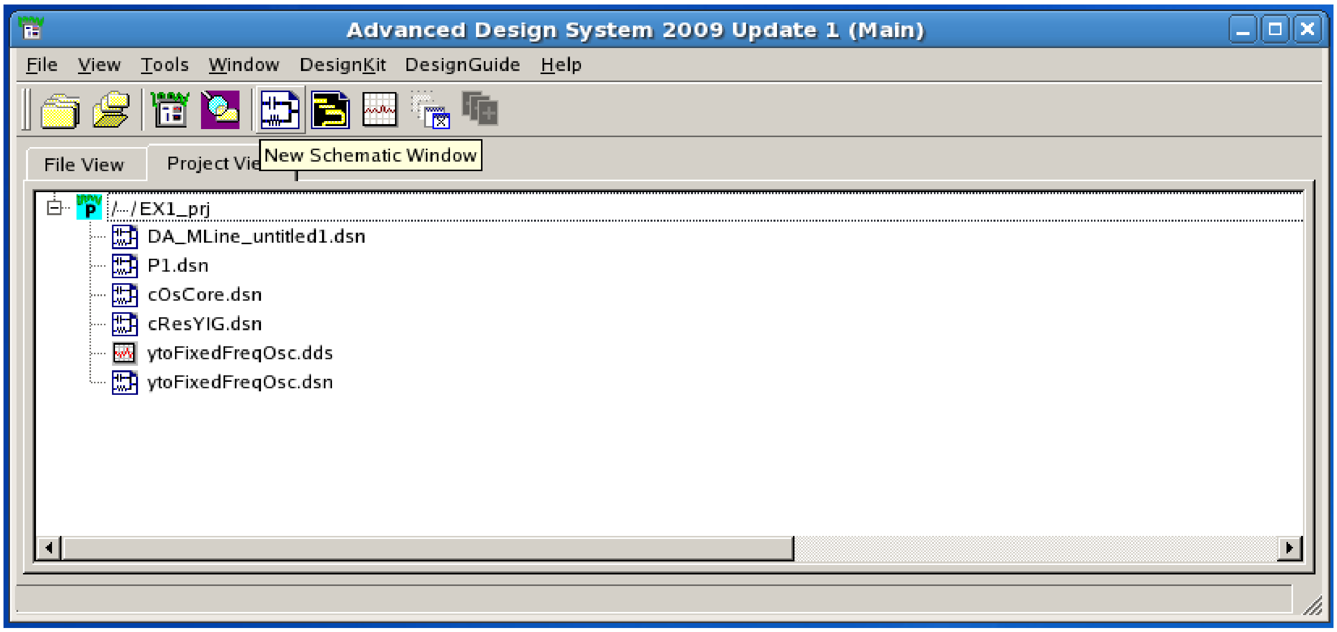
–Provide a name and then press OK. ADS add **_prj** at the end of project name.

–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.



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:

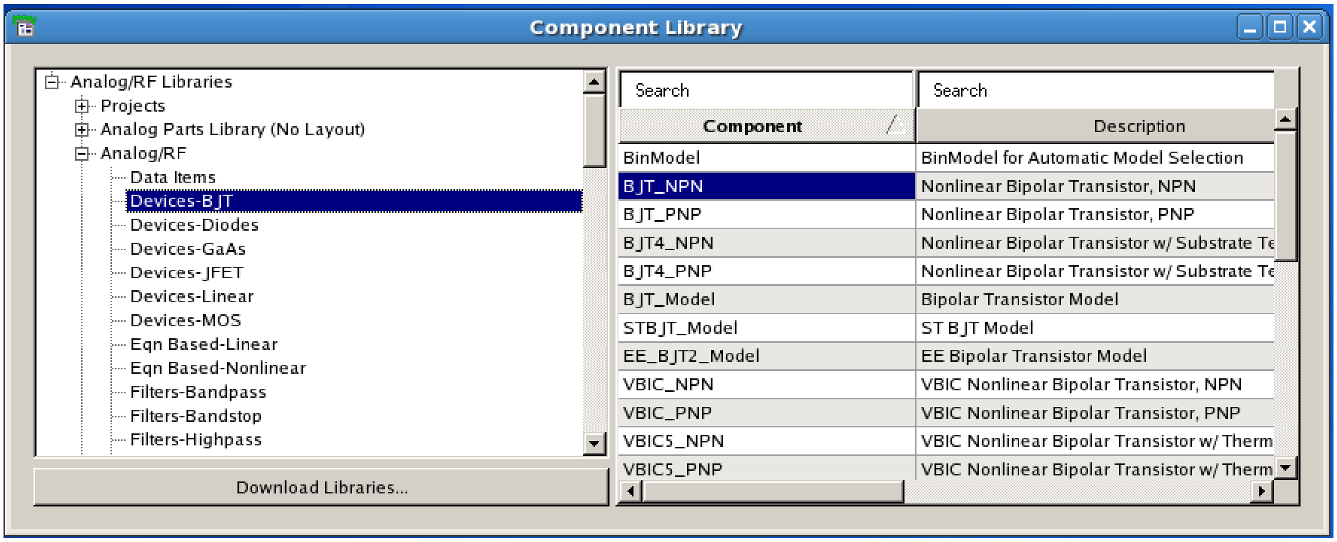
Before starting you can set the units from **Options>Preferences>Units/Scale.** Components can be selected either form **Component Palette** or **Component library**.

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**.**



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

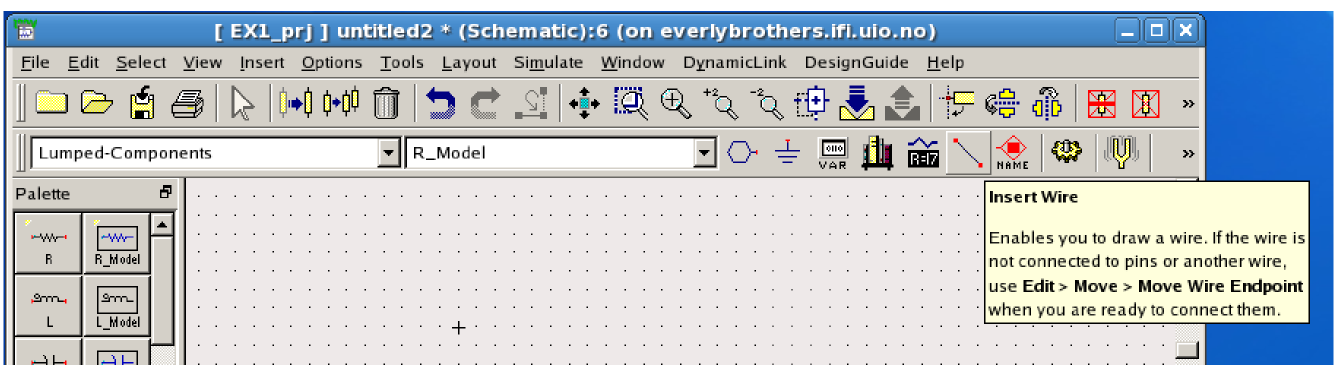
For copying a component use Ctrl+**c** or use **Edit>copy**

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.

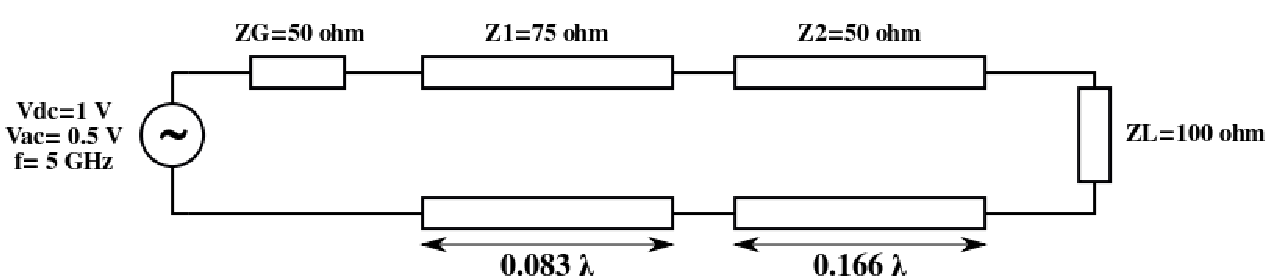
For connecting the components click on I**nsert Wire** icon.

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).

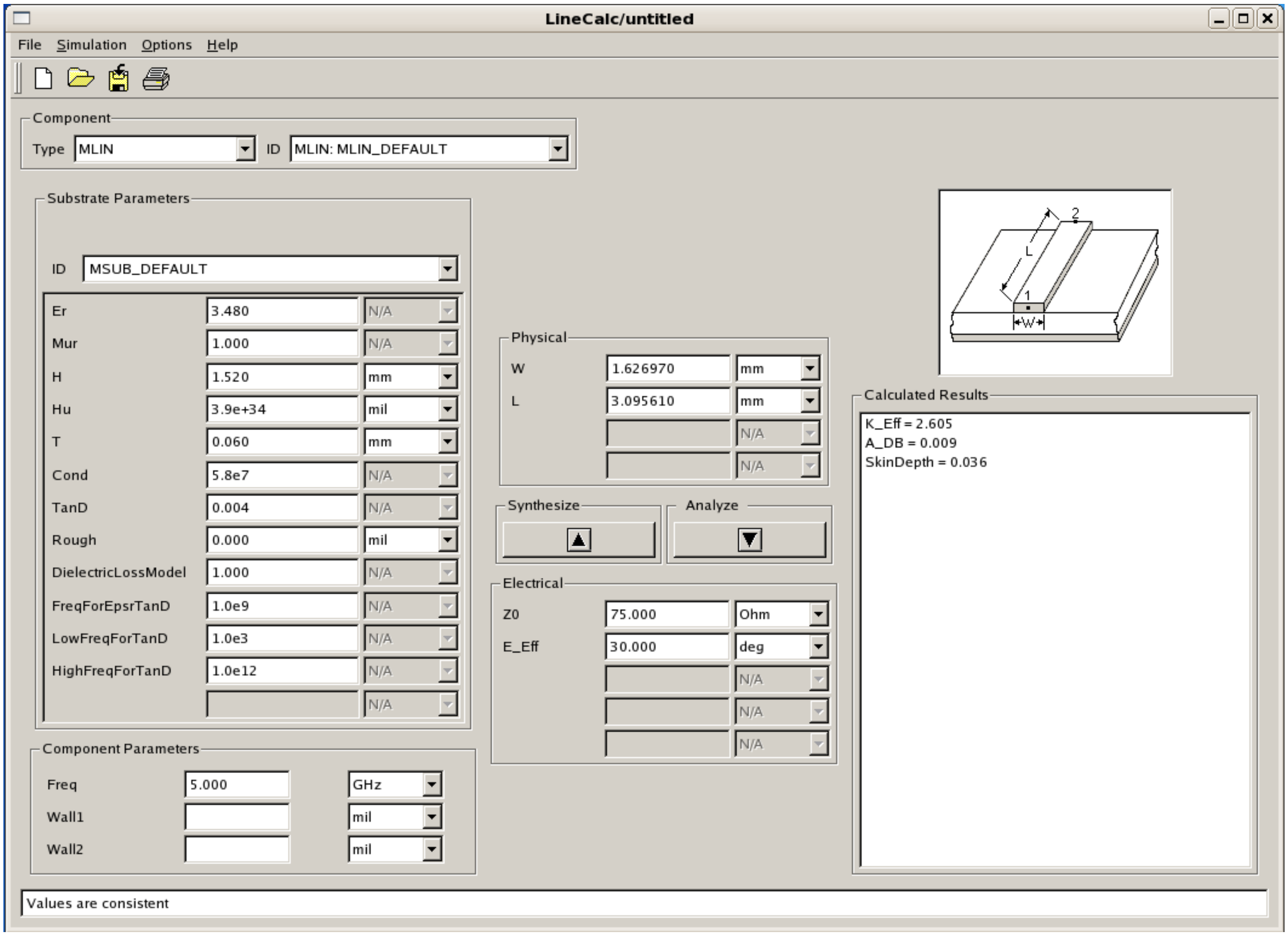
Choose **Tools>LineCalc>StartLineCalc** then a dialog box appears.

–Fill out the Substrate parameters as given above.

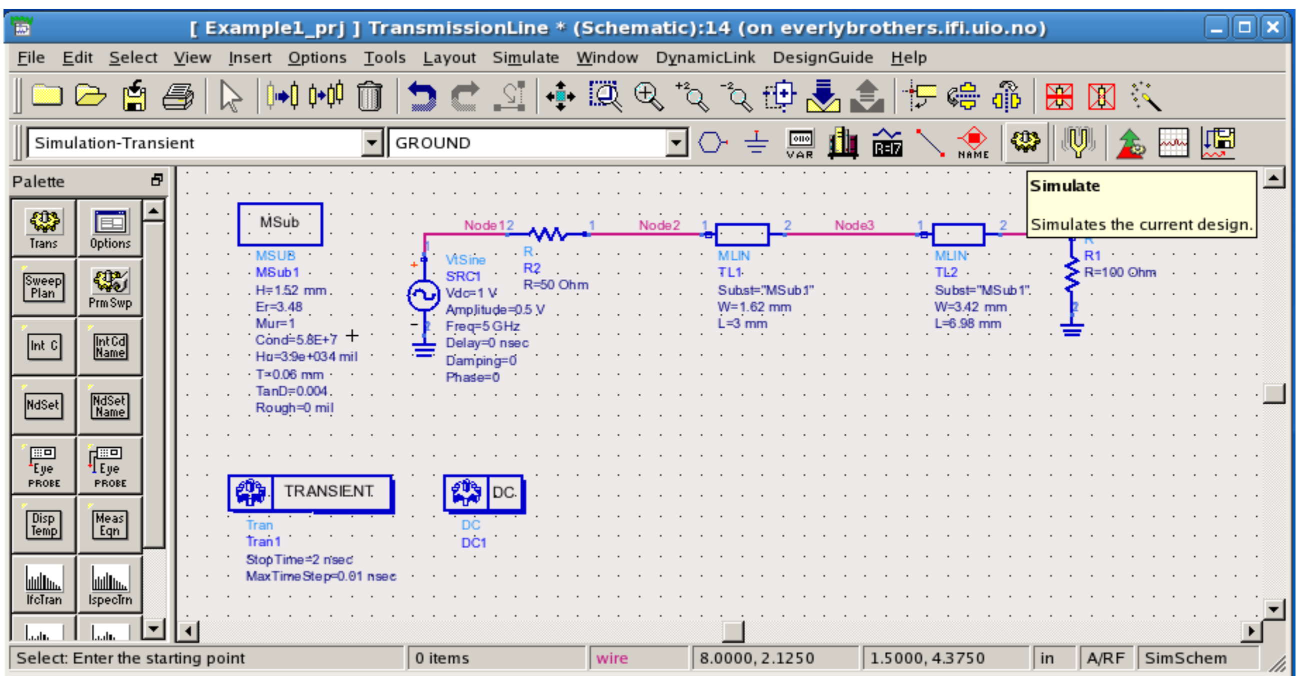
–Set Z0 is the characteristic impedance of the transmission line. E_Eff is Effective electrical length of line in degree. **0.083*lamda** is 30 degrees and **0.166*lamda** is 60 degrees.

–Set **Freq** to 5 GHz

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

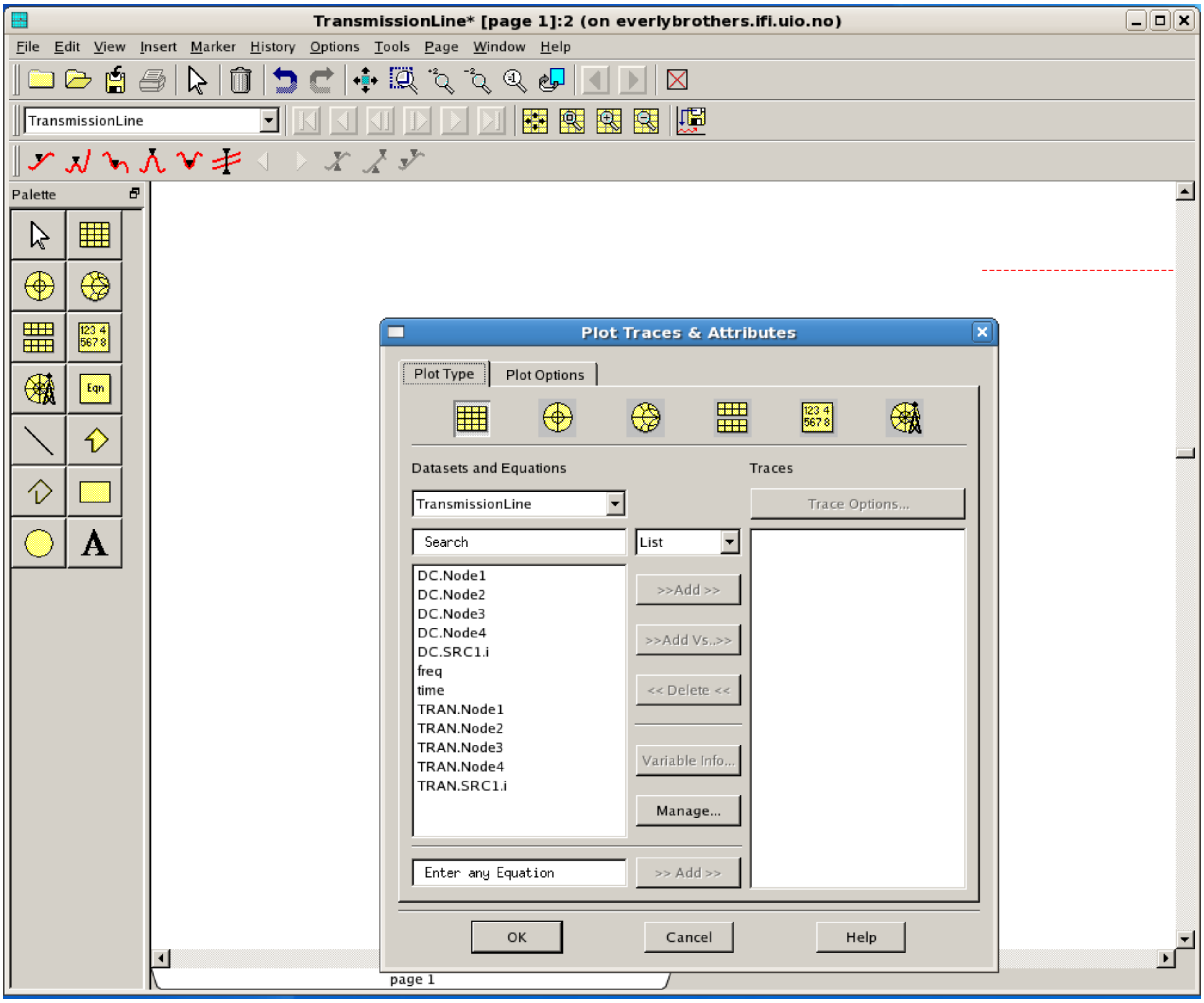


-Save the design and click on gear icon to simulate the circuit.

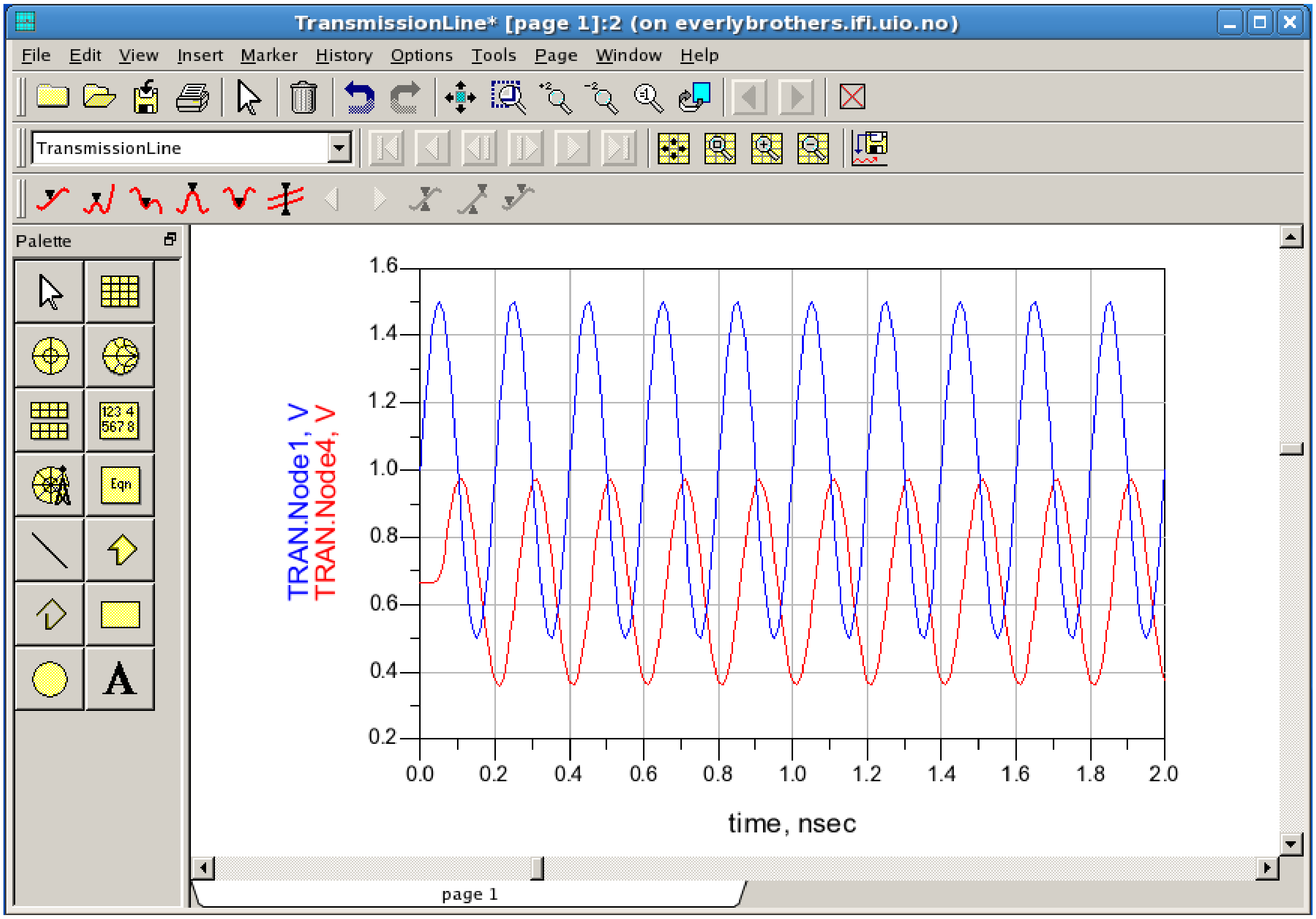


After simulation finishes, the **Data Display** window opens. From palette in **Data Display** window you can choose different charts to display simulation results.
-From Palette choose the **Rectangular Plot.** A **Plot Traces & Attributes** window pops up, from **plot**

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