Aalto University
School of Electrical Engineering
Department of Radio Science and Engineering

S-26.3120 Radio Engineering, laboratory course

Spring 2014

Introduction to

Microwave transistor amplifier design with ADS

Instruction translated and updated by:	Clemens Icheln/ Sathya Venkatasubramanian
Instruction updated:	13.01.2014

ADS simulation computer lab

Once the preliminary design is completed, the circuit is simulated, i.e. optimized with ADS. Also the circuit layout is done in ADS. The CAD software ADS is installed in computer class F402 on Windows-based computers, and software can be freely used within the computer class hours. You should start ADS simulations only after the preliminary exercise has been approved by the assistant.

Note that F402-class PCs are used by many students, so always save your project data in a safe place, e.g. your own USB stick or a network drive.

ADS Software:

Advanced Design System (ADS) is a high-frequency circuit design CAD tool. It supports the development of all types of RF designs, from simple to the most complex, from RF/microwave/MM-wave modules to integrated MMICs. With a complete set of simulation technologies ranging from frequency and time-domain circuit simulation to electromagnetic field simulation, ADS lets designers fully characterize and optimize designs. There is a good selection of help topics inside the software. More information can be found from the ADS on-line help at http://eesof.tm.agilent.com/applications/. The following guide covers the design basics with ADS. Refer to the ADS cookbook available at http://cp.literature.agilent.com/litweb/pdf/5991-1516EN.pdf for basic instructions on ADS.

Starting the program:

ADS can be found in a computer class F402. From the Windows start menu, choose the item

```
All Programs - Advanced Design System 2011.05 - Advanced Design
System 2013.06
```

When you start ADS it is possible to choose what you plan to do with help of the "start-up wizard". This point can now be bypassed and you then start directly with an empty new project. The main ADS window appears (Fig. 1).

If you are experiencing some problems with the software, please contact the assistant.
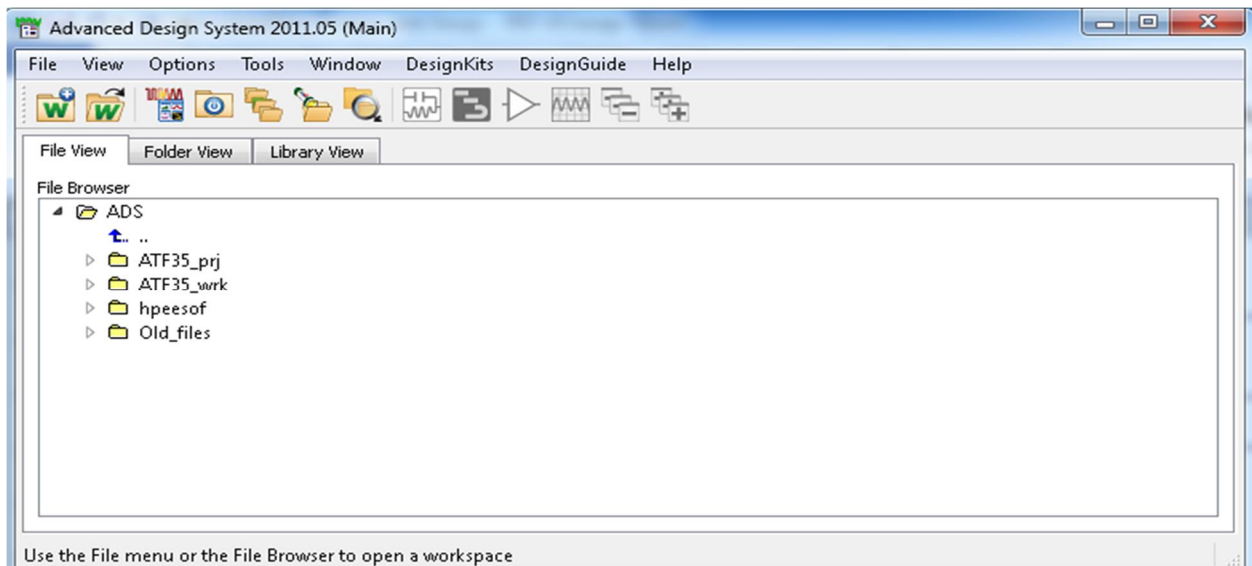
Fig. 1: The ADS start-up window

Creating your own project:

Make your own amplifier project folder, click

<div align="center">

File-New Work Space

</div>

And give it a name, such as

<div align="center">

Amplifierlab

</div>

Select the default options for included libraries and choose *millimetre* layout resolution. Now, you can create a new schematic window. An empty schematic window is shown in Fig. 2.
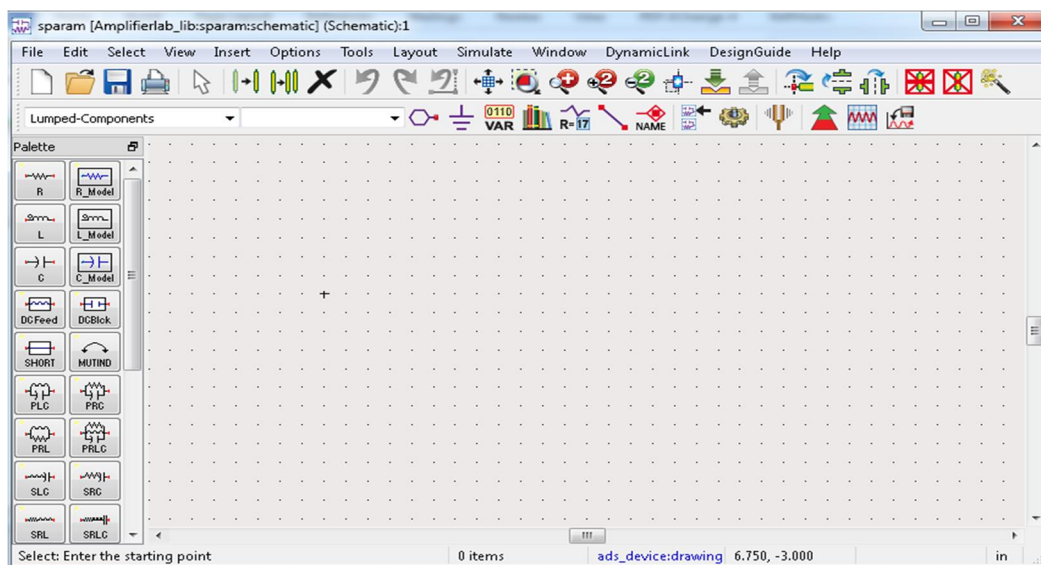

Fig. 2: Schematic window

ADS has now created a workspace *Amplifierlab.wrk*, in which all files created in this workspace can be found. *Although the files appear in Windows, do not copy or delete them from there, but do all file processing through ADS!*

Importing the transistor to the workspace(for the lab)

In the main window, select File -> Unarchive and then select the ZAP(.zap) file of the LNA which is available for download in Noppa. Proceed with the onscreen instructions to create a workspace with the transistor model. Once, the workspace is created, go to the Options-> Technology -> Technology setup and then select the unit as *millimetres* for the design.

Building a circuit

The amplifier circuit is created in this window (schematic). Click the left side of the window for selecting components (or insert them from the *Component* pull-down menu). Components are grouped by component palettes, different component are obtained by changing component palettes.

Possible desired components are, i.e. microstrip line, microstrip substrate, T-joint, radial stub, capacitor, resistor etc. Please note that if you did not change the unit length of the new project when you created it, the default unit length of *mils* is used, which is a thousandth of an inch. Changing the unit can be done under options

<div align="center">

`Options-Preferences-Units/Scale-Length-mm`

</div>

Component parameters are displayed by double clicking on the component and they can be edited from there. Explanation of the parameters may be found by clocking the help button of the component windows or from the PDF-manual. Figure 3 presents an example of the capacitor parameters.
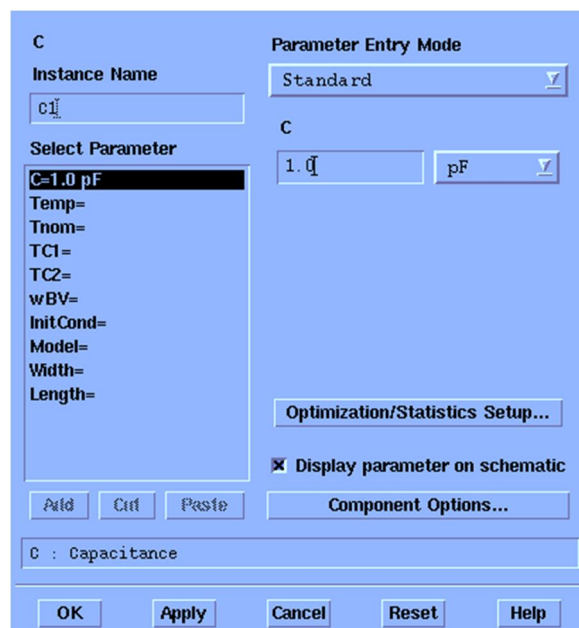

Fig. 3: Capacitor parameters. Explanation of the parameters can be displayed by pressing the help button.

The FET transistor that is used in this work has available measured S-parameter files from the manufacturer, which contain the S-parameters of the desired operating point. Please note that only the small-signal simulation (S-parameter simulations) can be done using these files. Place the 2-port component from the Component palette

<p style="color:red; text-align:center;">Data Items</p>

When double-clicking the component you get the information window. Select a file from

<p style="color:red; text-align:center;">File Name - Browse</p>

And choose e.g. the file

<p style="color:red; text-align:center;">c:\your_user_directory\bfptransistor.s2p</p>

Check again that the file type is Touchstone.

DC blocking capacitors from the manufacturer MURATA/ATC are used. A suitable capacitor can be selected by the student on discussion with the instructor. For designing purpose, a capacitor can be modeled by a capacitor, a resistor and an inductor in a series connection. The resistor is introduced for the losses while the inductor is used for parasitic. The Resistor, capacitor and inductor can be found in Lumped-Components menu. However, one can also use the measured S-parameter file provided by the vendor.

The amplifier circuit is typically unstable at very low frequencies. Stability of the designed circuit can be improved by connecting a resistance to improve the stability (standard procedure). A suitable spacing is required to produce a realizable layout.

Components are connected to each other via a wire, which has no significance in the simulation and can be assumed to have little practical significance in the layout either.

<p style="color:red; text-align:center;">Insert-Wire or Ctrl+W</p>

Ground node can be select from

<p style="color:red; text-align:center;">Insert-Ground</p>

The necessary extensions / generators can be found from the component palette

<p style="color:red; text-align:center;">Simulation-S_Param</p>

For the S-parameter simulation only terminations (*Term*) are needed at the ports. Note that the numbering of the ports is very important (the input terminal port is Num = 1 and output terminal port is Num = 2), in order to generate standard S-parameters. The use of equations for variables is very useful, for example, when you want to vary common input parameters of several components. In this case, the component is a variable input parameters (for example, the width "*w1*") and placed in the equation for the variable value. The equation can be placed in the circuit description of the file, by selecting

<p style="color:red; text-align:center;">Insert-Var</p>

For the variable we can write e.g. "*w1* = 3 mm", where *w1* may be the width of one of the microstrip lines (e.g., a 50 Ω line). One VAR component can contain several variables. Variables can be added by double clicking the VAR-component, giving a new name for the variable, for example, *w2*, and the value and the clicking *Add*.

Components and functions can also be chosen from the drop down menu at the top of the window using the icons.

Simulation

Circuit design should be done in small steps so that the initial simulation only includes, e.g. the bias circuit or the input matching circuit. Initially, the DC-block can be omitted from the matching circuit simulations. In smaller parts it is easier to find errors in the simulations and it can be checked first if the own pencil-and-paper circuits are working fine. These parts should be saved in your own circuit description files so that they can be simulated if necessary again later. (When simulating the matching circuits take into account the transistor dimensions, i.e. layout of the gate and the drain, see according Appendix).

There are also ready templates for performing different type of simulations, e.g. S-parameters for the simulation can choose the new file box when creating a description of district Insert- Templates click S_params.

Set the SP-component somewhere in the circuit schematic. You can set the simulation frequency here as a single point, upper limit and lower limit with particular frequency step. Simulation start menu

<div align="center" style="color:red">Simulate-Simulate</div>

Simulation status window appears after the initiation of the simulation. If you lose the simulation window, you can get it again from the window menu

<div align="center" style="color:red">Window-Simulation Status</div>

The simulator writes the results of the simulation in data file with the default name, which is same as the circuit schematic name. If you want to simulate several different circuit configurations and to compare them with the result window, or to compare the same circuit simulation results for different input parameters, it is possible to make more data files (should be done!) by placing them in the window control menu and naming them as you wish.

<div align="center" style="color:red">Simulate-Simulation Setup</div>

If the simulation is started, or ADS, an error occurs, the cause is an error in the circuit description file, or component parameter values that are not allowed to enter the area. Errors in the simulation box (Fig. 4).
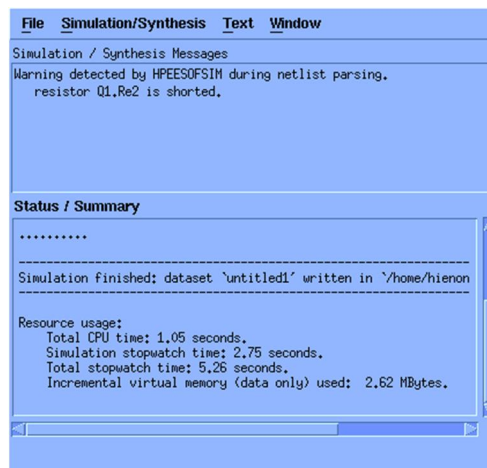
Fig. 4: Simulation window

Simulation results window

When simulation is completed a data window will open automatically for the graphical and numerical presentation of the result. This result or data window can also be opened later by selecting from main window.

<div align="center" style="color:red">

`Window-New Data Display`

</div>

Result/data window is shown in Fig.5. Save the results window will ask a name i.e. LNA_1. Select the desired coordinate system (linear scale, Smith chart, etc.) from the left side of the window and place it in the desired location. Afterwards, ADS will open a window where you can select the data set or the desired S-parameters generated after circuit simulation. In the result/data window, you can add the coordinates as much as you want. If you want to add data on the existing plot later, just double-click it and add curves.
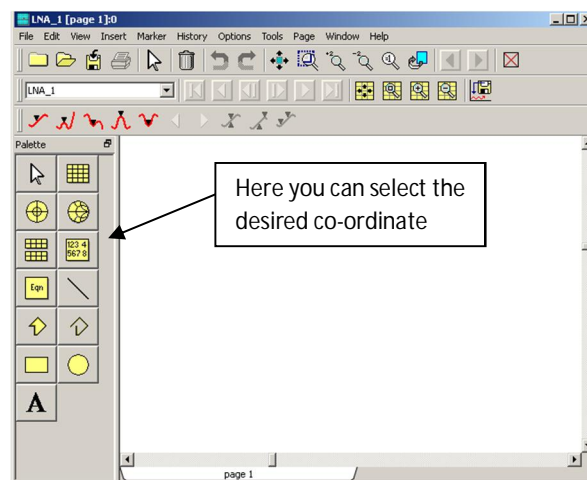

Fig. 5: Result/data window

ADS include built in functions, which can be dealt with simulation data. With the obtained S-parameter, stability factor such as coefficient K can easily be find out (the stability of the entire amplifier circuit should be checked across the frequency range close to DC), select

<div align="center">

Insert-Equation

</div>

and then enter the equation

<div align="center">

K=stab_fact(S)

</div>

Figure 6 shows the equation window. K: can be plot in the same way as the S-parameters. Once Equations is selected from the data set then K is in the variable list. Functions help can be find by pressing the Function help from equation window.
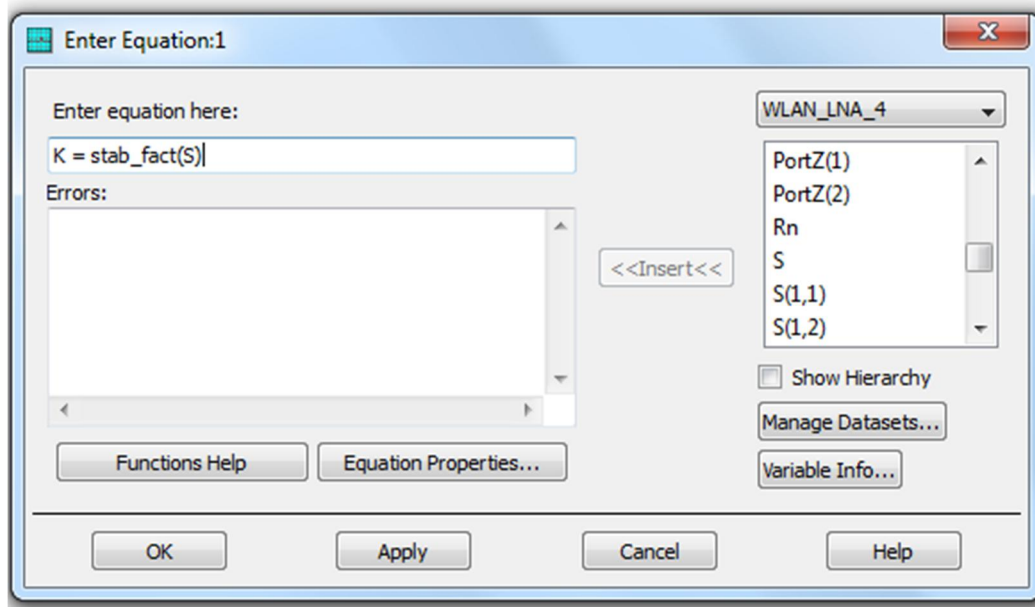

<div align="center">

Fig. 6 Entering an equation

</div>

Linecalc - Utility

For microstrip line input parameters are the physical dimensions and properties of the substrate. Substrate definition (MSUB) and microstrip line (MLIN) can be found under TLINES-MICROSTRIP component menu. From here you can also find models of various discontinuities, such as the various bends and joints. If you want to determine the electrical and physical length of the microstrip line, it can be done by LINECALC utility. It also allows you to calculate the width of the microstrip line from its specific characteristic impedance. The program can thus be calculated by hand to check the accuracy of measurements (should be done!). LINECALC starts by clicking

<div align="center">

Tools-LineCalc-Start LineCalc

</div>

from circuit schematic. Check the properties of the substrate as well as other parameters such as frequency. Length can be calculated by means of electronic length E_EFF. For example, a quarter wave case, is 90°.

Sub-circuit

In order to simplify the schematic sometimes it is good practice to create sub-circuit and which is very straight forward in ADS to create sub-circuits. Sub-circuit creation is done as follows:

<p style="text-align:center; color:red; font-family:monospace">File-New design</p>

When a new schematic file is opened, draw a new circuit or copy a circuit from other schematic to this new file. In this schematic, do not use any simulator (S-parameter, AC, DC, etc.). At the interface of the projected sub-circuit add so called port which can be found by selecting

<p style="text-align:center; color:red; font-family:monospace">Insert-Port</p>

When the connectors have been added and numbered, creating sub-circuits own higher-level circuit description appears in the display. Select

<p style="text-align:center; color:red; font-family:monospace">File-Design/Parameters</p>

Symbol Name and sub-circuit's suitable symbol can be set from here. For example, SYM_2Port is default for two port sub-circuit. Sub-circuit is ready to use in any schematic and can be found by selecting

<p style="text-align:center; color:red; font-family:monospace">Insert-Component-Component Library</p>

You can edit the sub-circuits normally by open it.

<p style="text-align:center; color:red; font-family:monospace">File-Open</p>

Optimization

Once the initial design has been done, or fine-tuned with help of the simulator close to the desired circuit, the circuit can be optimized with respect to the desired circuit characteristics, such as gain or matching, etc. (there may be more than one goal). The optimization process should not be started with the *brute-force* method, i.e. too early and far off the desired performance, because one could end up in the wrong minimum or maximum. This procedure also would not develop understanding of the underlying design principles, and certainly not help understanding the potential failure causes of the final circuit. It is also worth considering carefully which parameters are useful as the optimization variables. Also, all parameters must remain within reasonable limits. Note in particular that the microstrip line width may vary due to manufacturing tolerances. Therefore, it is advisable not to use very narrow lines (<0.4 mm), i.e. avoid very high impedance lines, since the relative effect of width uncertainties is then greater.

The input parameters used in the optimization may be defined through the variables in the circuit window (see above). For example, defining the length *l1* to be the parameter to be optimized is done as follows. In the circuit window add the variable component *Var*. Double-click it. In the window you now name the variable and click `Tune/Opt/Stat/DOE Setup`. Then choose `Optimization Status-Enabled`. Hereby the variable is chosen as the parameter to be optimized. Choose a reasonable default value as well as minimum and maximum values. Remember defining units! Finally close the `Tune/Opt/Stat/DOE Setup`- and `Variables and equations`-windows by clicking `OK`. Now in the circuit window the following is visible.

<p style="text-align:center">9</p>

`l1=2.0 mm opt{ 1.0 mm to 3.0 mm }`

where the variable is defined by the starting value, allowed range, and accuracy, which depends on the given amount of decimals. The optimization will modify all components of the circuit that contain the variable `l1`, i.e. only these components are part of the optimisation. Parameters of the optimization can be several but keeping an as small number as possible is advisable to keep the calculation time reasobale and the circuit diagram fairly close to the initial manual one.

The optimization requires in addition to the already definde S-parameters (`SP`) the optimization target (`Goal`) and the control (`Optim`) components, which are found in the `Optim/Stat/Yield/DOE`-component palette. First place the target component into the circuit diagram and double-click it. Name the component e.g. `OptimGoal1`. The `Expr` parameter should then be set e.g. to dB(S(2,1)), in which case the gain is optimized. Choose as `SimInstanceName`-parameter under `Analysis Components` the S-parameter simulation "SP1", which is available if the circuit description contains an `SP` component. Then define minimum and maximum target values, for example, 8 dB and 10 dB. The `Weight` parameter allows focusing on different targets relative to each other (the default value is 1). Specify the frequency range, in which the target is to be optimized. This is done by setting the `RangeVar`-parameter to `freq` and the lower and upper frequency limit with the `RangeMin`- and `RangeMax` parameters. Accept the target by clicking `Apply` and `OK`. Possible additional parameters that should be optimized, e.g. desired gain and matching, are similarly added as target components.

Next, double click the control component(s) and select from the `Setup` menu an optimization method. There are several methods, of which *gradient optimization* is used here. Familiarize with the different methods in the user manual of the software. As the target (`OptGoal`) select from the `Edit` menu the previously defined target `OptimGoal1`. If individual targets are not defined here, the optimization uses all available targets found in the circuit. The number of iterations should be small (default: 25) at first and increased later. Now click `Apply` and `OK`. All settings needed for the optimization are now complete. Additional options are worth exploring in the user manual.

To start the optimization, select `Simulate`. If you want to perform a simple S-parameter simulation, click on the `Optim` component with the **right** mouse button. From the pop-up menu, select `Component-Deactivate` which deactivates the optimization control component. For the optimization you need to activate the component again. The parameters changed due to the optimization are updated in the circuit description file by selecting

`Simulate-Update Optimization Values`

If you are not satisfied with the results of the optimization or experience convergence problems, check that the final values of the variables have not ended up at the given boundary, and that the target was reasonable and correct.

Archive/Unarchive project

In order to work with ADS model for different component, it is important to unarchive the project since most of the time the model are available as an archived project (.zap format).

However, it is recommanded to archive your project everytime after you finish your daily work and save in a safe place (since the lab PC's are public).
Your project can be archived from the ADS main menu by

<p align="center" style="color:red">File-Archive Project</p>

In the same manner any archived project can be unarchieved from the ADS main menu

<p align="center" style="color:red">File-Unarchive Project</p>

It is also possible to include an unarchived project into another project. In this work you need to unarchive the transistor model project available from the manufacturer and then include that into your project.
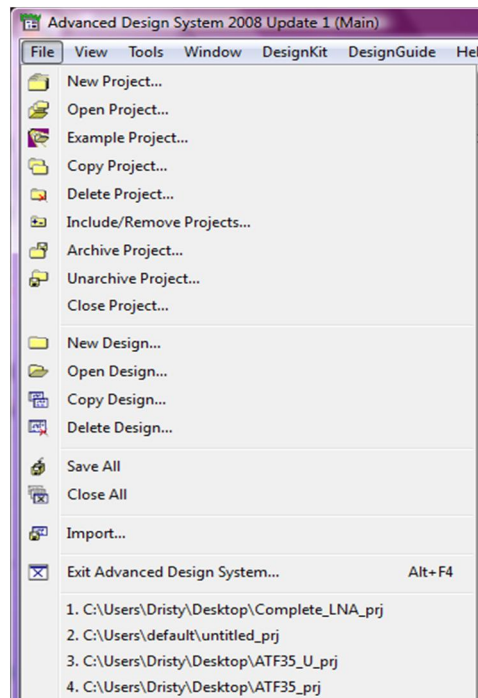
Fig. 7: Archive/Unarchive project

Design Guide

There are number of design guides in ADS. In this particular problem design guide can be used to find out the I-V curves of the transistor. Choose the appropiate transistor technology. Figure 10 shows where to find out the design guide for I-V curves. Remove the transistor in the design guide and insert the transistor that the I-V curves should be plotted for.
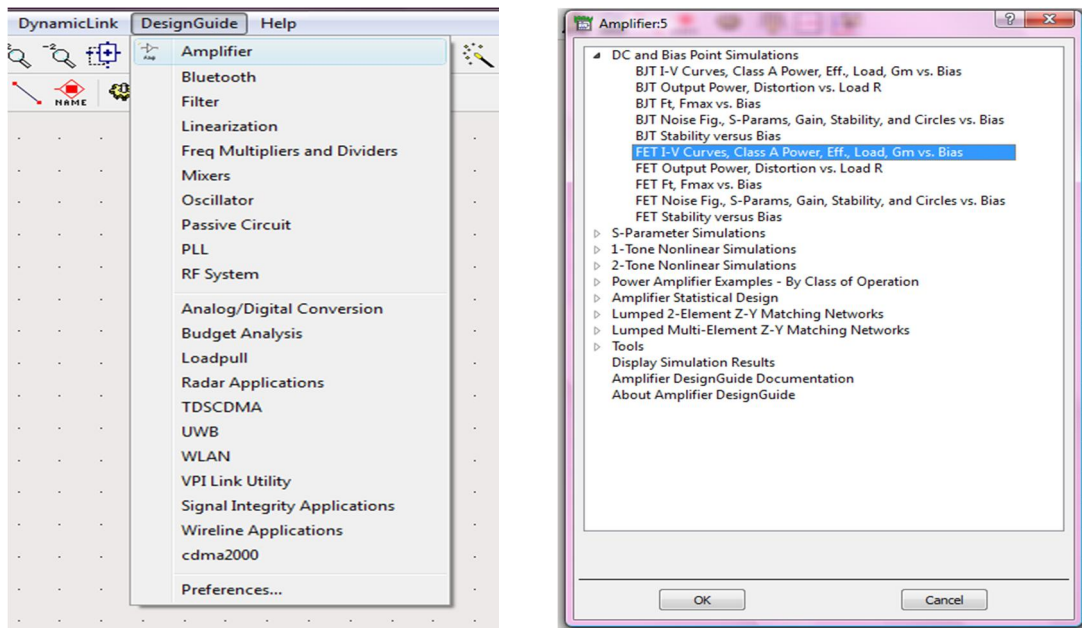
Fig. 8: Design guide for finding I-V curve

Layout

Creating the layout of more complex circuits is usually always a parallel process with the circuit simulation, because the layout sets many restrictions, which are difficult to predict in the circuit design. In this work, however, the layout can be done in a fairly straight-forward manner after the simulation, since a simple circuit layout can already be derived from the circuit diagram. After a successful simulation, you can create a layout file by selecting

<div style="text-align: center; color: red;">Layout-Generate/Update Layout</div>

Check that the unit is set to millimeters! Therefore, set

<div style="text-align: center; color: red;">Options-Preferences-Units/Scale-Length-mm</div>

and

<div style="text-align: center; color: red;">Options-Preferences-Layout Units-mm</div>

The principal PCB layout with all dimensions is shown in Figure 9. Adjust the input and output line lengths so that the amplifier with a size 60 x 85 mm^2 fits nicely onto the PCB for the given ground plate specifications. The larger measure is in the signal flow direction, and it is also the spacing between the two RF connectors. Also, plan the biasing circuits so that they fit onto the PCB. The PCB will be attached to a metal base with four 3-mm nylon screws near the RF connectors. Drilling of these holes is facilitated by placing patches on the board. Such patches can be set by selecting

<div style="text-align: center; color: red;">Insert-Circle</div>

and then

<div style="text-align: center; color: red;">Insert-Coordinate entry</div>

with help of which the spots can be located precisely. See Figure 11 for the locations of the eight holes. The intention is that the circuit board can be attached securely and easily to the metal base.
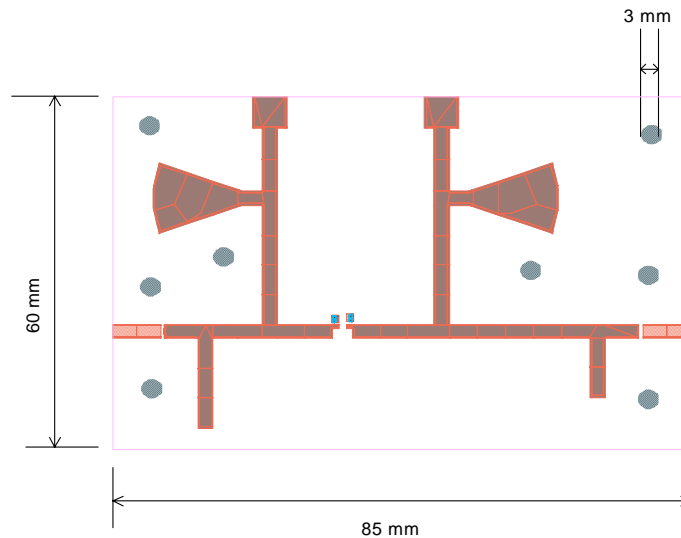

Fig. 9: Principle PCB layout of the amplifier circuit.

For the transistor foot print has to be made according to the transistor data sheet. Put the origin to the left bottom corner of the PCB (Edit-Modify-Set Origin). For the source grounding two vias are needed. For the via use microstrip line component VIAFC with a hole diameter D = 0.7 mm and for the width of the patch Dpad1 a value larger than the hole diameter. So with the patch we realize a metallization around the hole. Set the same value to Dpad2. Define the layer Cond1Layer to be Cond, but insert the layer HoleLayer = hole. Thus the holes are placed in a different layer.

For the DC biasing resistors sufficiently large soldering patches are required. Triangles in the corners of the PCB facilitate cutting the PCB to the right size. Group number and student names are inserted on the board in an unused area. Mark the input and output side (with "IN" and "OUT "). The text entries, rectangles, lines, etc., can be found under the Insert menu. Finally, eliminate unnecessary patches and lines, so that only the desired metallization picture remains (with help of the layer and preference settings).

Printing

Printing is done by selecting first the active window to be printed. Then click

<div align="center">File-Print or File-Print Area</div>

which opens a control window to start printing. You can select the printer from menu

<div align="center">File-Print Setup-Options</div>

or from the printing control box Options. Selecting File (Generic Only) directs the print file to your project directory. To add an image to e.g. a Word document, print the image in EPS format (default). Note that Word displays only a placeholder for the EPS graphic and that you need post-script printer to print the file. You can also save the simulation results in an ASCII file using the commands in the result window

File-Export-Write selected item to tab-delimited ASCII

Tools-Data File Tool…

The first one prints data from the selected item whereas the second line uses the entire data file.

## *Harmonic Balance Simulation*

Place a Harmonic Balance setup on the schematic and deactivate the S-Parameter simulator. Open the setup for harmonic balance and define the fundamental frequency.
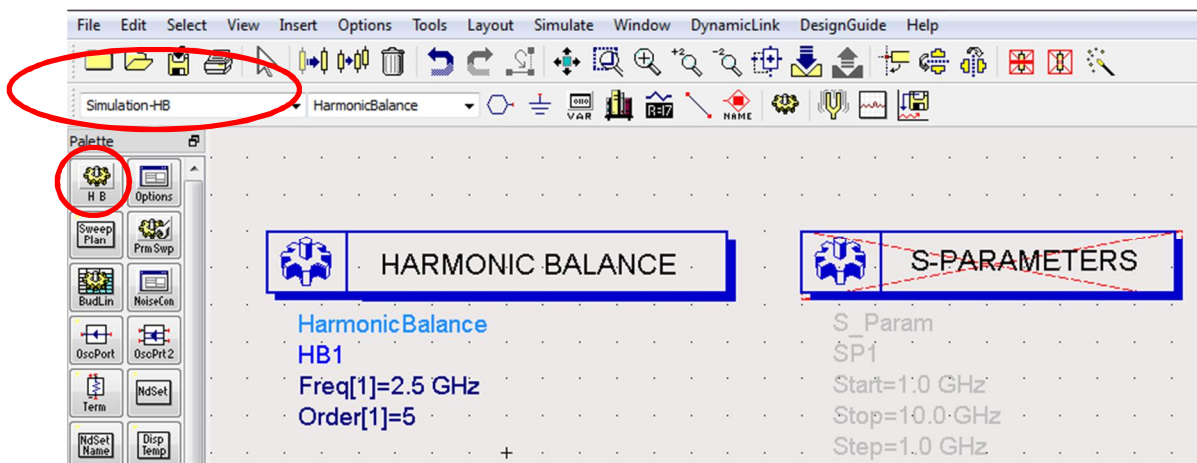

Remove or deactivate the input Term. And place a P_nTone on the workspace. Setup the P_nTone same frequency as on the harmonic balance and desired power in dBm.
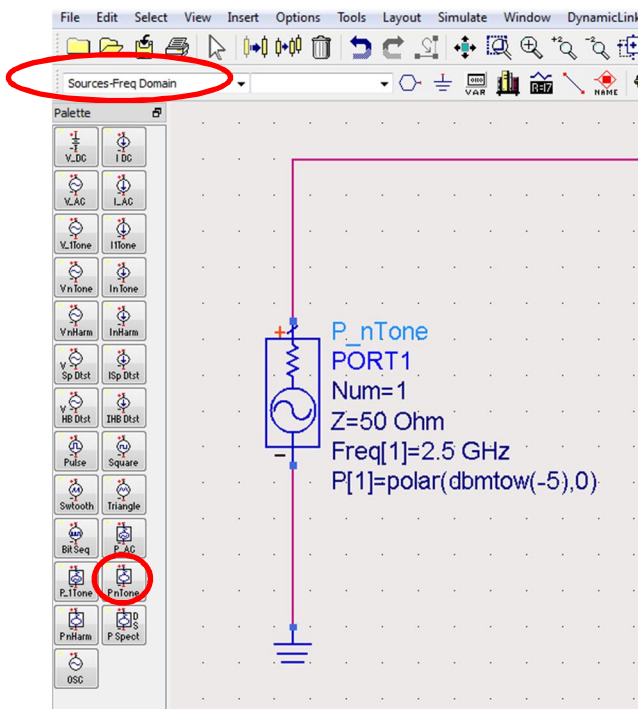
Name the output wire on the design and place the current probe in the scematic

14
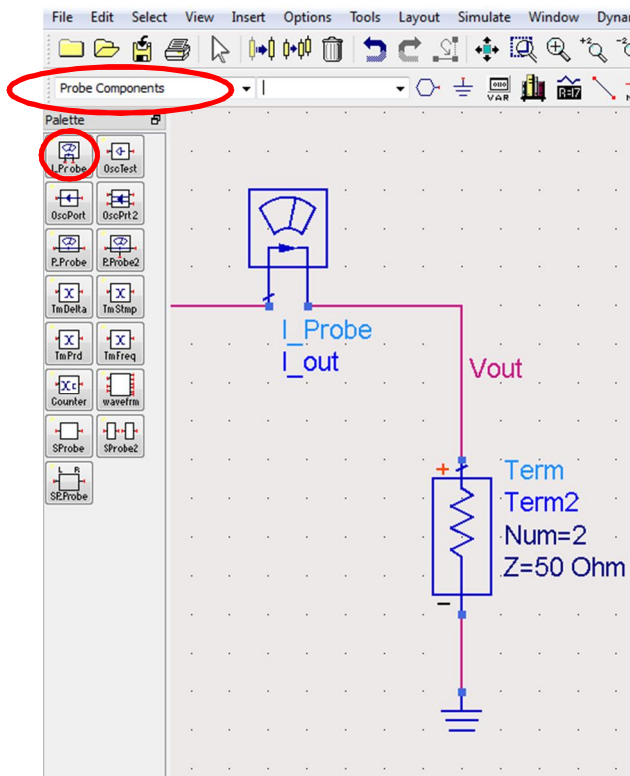
Run the simulation. Place a rectangular plot in the result workspace and choose the named wire as the trace. It is possible to observe the output both in frequency domain and time domain.
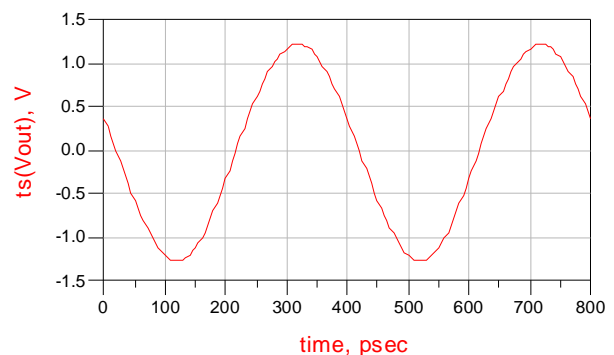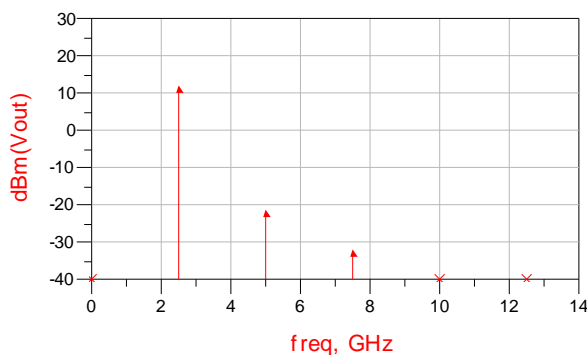
1 – dB compress point

Place a variable on the schematic for input power. Name it and open the harmonic balance setup. Under the tab Sweep, insert the variable name in Parameter to sweep and put the start and stop value. Run the simulation.
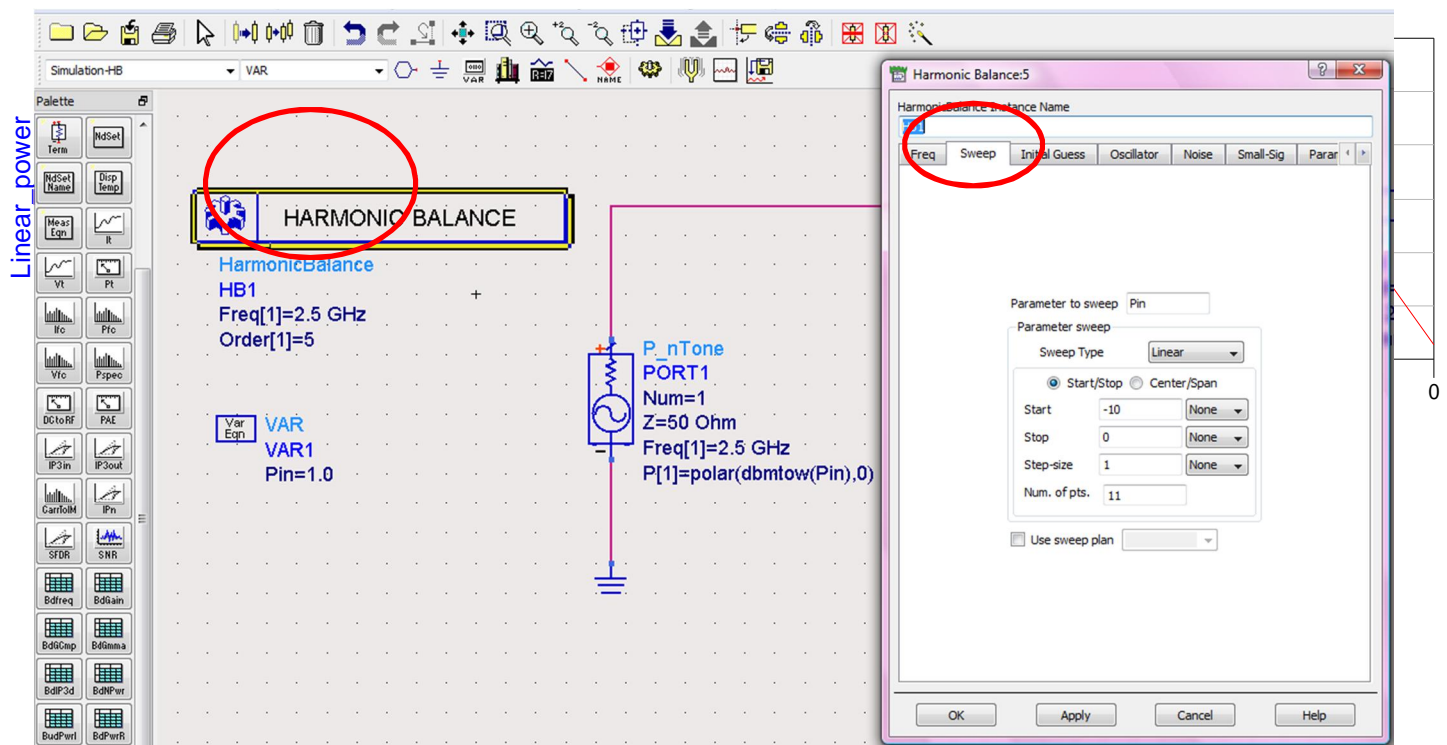
To visualize the result in a good way some equations need to be created.

For calculating the power in watt, Pout = 0.5*real(Vout*conj(I_out.i))

For calculating the power in dBm, Pout_dBm = wtodbm(Pout)

For calculating the Gain, Gain = Pout_dBm[1] – Pin

For calculating the linear power, Linear power = Gain[1]+Pin

Amplifier construction

When the performance and layout of the final circuit has been verified, contact the course assistant. Circuit boards are made for all groups at the same time through Prinel Ltd (www.prinel.fi). The circuit layout has to be submitted to the PCB manufacturer in GERBER format. Instructions of how to export such a file are provided later.

When the PCB is ready, the manufacturing of the circuit is done in the RAD-department's workshop (3rd floor), where a desk is reserved for this purpose. The assistant will provide the transistor, the DC-block capacitors, the necessary biasing components, the metal base, SMA connectors and nylon screws. First drill 3-mm holes for the nylon screws in the PCB at the right places. Solder DC-block capacitors and other components in place *before* inserting the transistor. Determine the order of the transistor pins. Solder first a small amount of soldering tin onto the transistor patches. Then press the transistor onto the circuit board and solder all the pins. Note that static electricity can easily destroy the transistor. Therefore, it should be noted that:

• Some clothes collect more static electricity than others, a woolen sweater is dangerous!
• Before handling the transistor remove static charges from yourself (use anti-static wrist-band).
• When soldering the transistor ground all parts thoroughly.

For the measurements, attach the circuit board with nylon screws to the metal base. Good electrical contact between the PCB ground plane and the metal block is important, especially near the RF connectors (hence the location of the nylon screws). First solder the bases of the SMA connectors to the PCB ground, and their center conductors to the 50-Ω microstrip lines. Do not use too much solder. Now, the amplifier is ready to be measured.

Measurements

Each group reserves a time slot for the measurements. Reservation lists will be available on the RAD announcement board on the 3rd floor. The available equipment is listed below in List 1 (equipment can still change, will be announced separately). Before the measurements, each group thoroughly plans their measurements (how measurements are performed, what parameters are measured, the measurement connections and biasing arrangements) and briefly presents them to the instructor, who then gives the "go ahead" for the measurements. If the group is not sufficiently well prepared for the measurements, the instructor may, if necessary, send a group away to reconsider the arrangements for measurements.

Four measurements are carried out for the amplifier (see course material). The measurements are as follows:

1. S parameter measurements.
2. 1-dB compression point. Measurement is made at one frequency.
3. Third-order intermodulation and intercept point.
4. Amplifier noise factor and temperature. Measurements are performed with the Y-factor method with help of a noise diode, across a suitable frequency band.

Guidelines for the measurements:

• Measurements must follow general precautionary measures such as for using power supplies.
• All connections must be checked before the power / voltage is supplied.
• The maximum power rating for equipment and components must not be exceeded.
• The expensive measuring devices must be treated carefully.
• The RF connectors must be handled correctly (do not break the center pin by turning the whole connector, only turn the sleeve). Tighten/open SMA connectors with a special torque wrench!
• RF cables must not be bent too much. A sharp bending can ruin the cable.
• Avoid mechanical stress on the connectors by properly supporting components etc.
• At the end of the measurements, all measuring devices are switched off, the connections are dismantled and components are returned to their storing places.

List 1

• Vector Network Analyser
• Signal Generators:
  - HP8340B f = 10 MHz ... 26.5 GHz, P = -100 ... 12 dBm
  - HP8350 & 83540 A f = 2 GHz ... 8.4 GHz, P = -2 ... + 22 dBm
• Power meter, HP437 B & HP8487 A, P = 1 μ W ... 100 mW, f = 50 MHz ... 50 GHz
• Fixed attenuators, for example, Narda, 1 ... 10 dB, SMA connector
• Spectrum Analyzer, HP8596 E, 9 kHz ... 12.8 GHz
• Power Divider, HP116673, f = 0 GHz ... 26.5 GHz
• Amplifier Chain, "AMP", f = 10 ... 100 MHz, G = 26 dB, F = 1.7 dB, U = 10 V
• Amplifier, Avantek AWT-6033, f = 2 ... 6 GHz, U = 12 V
• Double-balanced mixer, Mini-Circuits ZEM-4300, f_{RF} & f_{LO} = 0.3 ... 4.3 GHz, f_{IF}= 0 ... 1 GHz, L = 6.5 ... 11 dB, P_{LO} = +9 dBm, P_{RFmax} = +1 dBm
• Low Pass Filter, f_{3dB} = 30 or 40 MHz
• Noise diode ENR = 13.2 dB
• Voltage sources, 0 - 30 V
• Multimeter, transitions, cables

Measurement file handling

The measurement results with the VNA can be saved and imported to ADS. (Bring a floppy disk to store the results.) Hereby, the measured and simulated results are later easily comparable on the same coordinate system, convenient for the final report.

In ADS open your amplifier (*Amplifierlab*) and open the results window. Select

<div align="center">

`Tools-Data File Tool…`

</div>

Choose as file type `Touchstone` or `Citifile` depending on the measurement file format. Locate the file and save it in the desired `Dataset`. Now the measurement results are available in the desired output file.

Alternatively, you can also copy the simulation results to your own computer and compare the results using for example Matlab.

Final report

When the amplifier has been built and tested, the final report is compiled. The final report includes the following:
• a clear description of the goals of the work (preliminary exercise results)
• a clear description of the ADS simulations, optimization, and final results;
• the final layout of the amplifier as well as other information necessary for the structure (biasing, groundings, DC blocks, etc.);
• measured amplifier characteristics and comparison to simulated and initial calculated values;
• a description of measurement methods.

In the final report for the measurement of at least the following is required:

• necessary calculations and explanations, etc.

• figures and tables presented objectively, coordinate systems with clear scales

• error estimations (Note that all the nominal component values can not be directly regarded as an absolute truth, especially if they can significantly affect the measurement result)

• comparisons of calculated and measured values, conclusions of causes and consequences, etc.

• you can compare your results with the measurement results of other groups. Though all groups got different specifications, you can discuss how much the computer-aided optimization helped?

• Suggestions, comments are appreciated, as well as everything else that fits well in such documentation.

Design flow summary:

1. Manual solutions
2. Find Γ_S and Γ_L in ADS (.s2p)
3. Matching Networks in ADS (.s2p)
4. Optimization in ADS (.s2p)
6. Insert nonlinear transistor model (.zap)
7. Determine bias point and Optimize with model (.zap)
8. Change to real components (.zap)
9. Layout
10. Manufacturing
11. Measurements