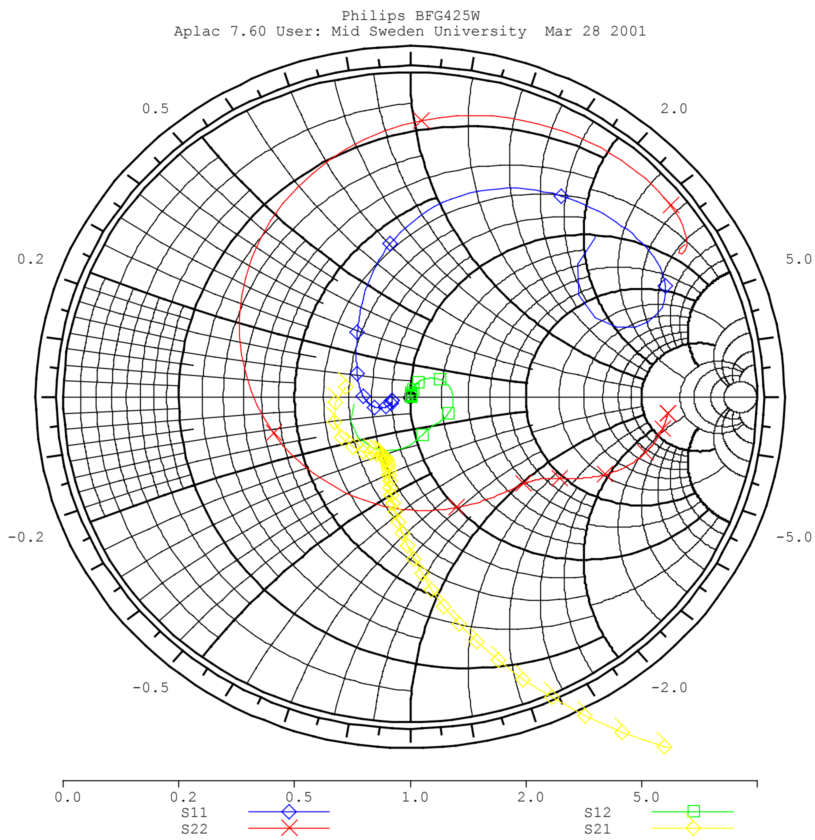
Laboratory exercise
High-Frequency Electronics
Spring 2002



Kent Bertilsson 2002-04-09

General instructions

The laboratory course in GHz electronics consists of three laboratory exercises.

- Introduction to scattering parameters
- Matching Networks
- Amplifier design

All of these are performed using the APLAC 7.6 simulation tool. The manual for APLAC can be borrowed during the course or bought for 300 sek.

The manual can also be reached online with the command

/>aplachelp

To setup the APLAC environment in the Unix system add the following line to your .cshrc file

'source /home/kenber/.aplacrc'

The software also runs under Windows and is installed in the electronic laboratory rooms L-213 and 214. This might be useful for copying pictures from the simulator into documents and also running simulations.

Valuable links to the APLAC simulator is

'www.aplac.hut.fi/aplac/'
'www.aplac.com'

Documentation

A final report covering the three laboratory exercises should be handed in at the end of the course. In the report all steps should be well documented and easy to follow. In the report all of the following thing are required.

- All calculations that have been performed.
- All input files to the Aplac simulator.
- Results obtained from all the simulations.
- Both the input files and the results should be well documented.
- Some of the exercises are given in several steps before a final design is reached. In these cases every change that has been made from the previous design must be documented.

1 Introduction to Scattering Parameters

This laboratory exercise is an introduction to the scattering parameters, which are widely used in microwave electronics. After the laboratory you should be able to calculate the scattering parameters for simple circuits and be familiar to the scattering representation in both bode-diagram and smith charts.

To reduce the time spent in the laboratory environment the calculations of the scattering parameters (exercise 1.1.1, 1.2.1 and 1.3.1) are preferably performed prior the laboratory occasion.

1.1 Shunt element

The first exercise is to investigate the scattering parameters for a shunt element. The S parameters should both be calculated theoretically and simulated and equal results should be obtained.

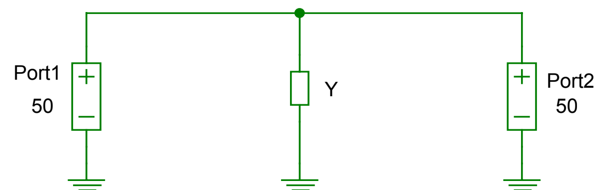


Fig. 1.1. Schematic figure of the shunt element, *Y*.

1.1.1 Calculate the S-parameters

Assume arbitrary admittance *Y* and calculate S_{11}, S_{12}, S_{21}, S_{22}.

- S_{11} is the reflection at the input port with terminated load $S_{11} = \Gamma_1\big|_{\Gamma_L=0}$.
- $\Gamma_1 = \dfrac{Y_0 - Y_1'}{Y_0 + Y_1'}$ where Y_1' is admittance of the shunt element *Y* in parallel with the transmission line Y_0.
- S_{21} is the transmission from port 1 to port 2 with terminated load, $S_{21} = \dfrac{V_2^-}{V_1^+}\bigg|_{V_2^+=0}$
- Use the fact that the input and output voltage must be equal since it a pure shunt element, $V_1^+ + V_1^- = V_2^+ + V_2^-$
- Use the symmetry to calculate other Scattering parameters.
- Calculate the scattering parameters up to 10 GHz using matlab assuming the admittance to be a resistance R = 100 Ω in parallel with an inductance L = 1 nH and a capacitance C = 1 nF.

1.1.2 Simulate the S-parameters

Create an Aplac simulation file according to the Fig. 1.2 and the list below.

- Draw the schematics
  - Place components by right click and select the appropriate device.
  - Change device properties by double-clicking on the device.
  - Devices can be rotated by ctrl-R.
  - Draw connections by pressing ctrl-W.

- Click “Show object list box” in the “Presentation menu” and place the box.
- Double click on the box and then the line containing “Isweep”
- Change the object type from “Isweep” to “Sweep”
- Fill in the Sweep properties according to the Fig. 1.2.
    - Line 1: Automatically placed
    - Line 2: Graph title
    - Line 3: Sweep the frequency linearly in 201 simulation points between 0 and 10 GHz.
    - Line 4: Create on Smith chart
    - Line 5: Normal diagram. Y axis is between –20 and 0 dB
    - Line 6-8: Plot S_{21} and S_{11} in the smith chart
    - Line 9-11: Plot S_{21} and S_{11} in the other diagram

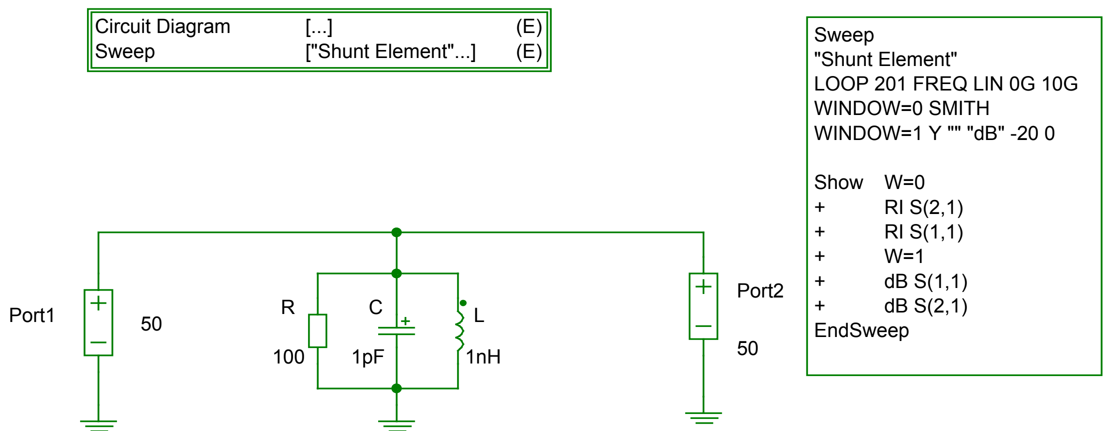


Fig. 1.2. Aplac file for the shunt element simulation.

- Simulate the Schematic by pressing ctrl+S.
- What is the value of S_{21} and S_{11} at 5 GHz.
- What is shown in the bode plot?
- What is shown in the Smith chart?
    - Observe the value in the window tile at a specific point both as you drag the mouse over it and as you trace a curve, using “probe” under the “option” menu.
    - Under matched conditions S_{11} is small (below –30 dB) corresponding to negligible reflections at the input. In the smith chart S_{11} is in the center of the smith chart (1+j0) under matched conditions.
        - What is actually shown in the Smith chart?
        - How can this value be extracted easily from the chart?
        - How can S_{11} be extracted from the Smith chart?
- Remove the resistance (Disable it or set R to 100 kΩ) and perform another simulation
    - What affect have the resistance in the circuit?

1.1.3 Verify the results

Verify both the calculations and measurements by printing the graphs of the scattering parameters achieved both from calculations and simulations. Present both the graphs in equal size with the same axis

In the report all of the following must be included

- All calculations of the Scattering parameters.
- Matlab code to generate figures of the S-parameter.
- Matlab figures
- Printout of the Aplac simulation file
- Aplac simulation results.
- Conclusions

1.2 Series element

The scattering parameters for a series element should be calculated and simulated.

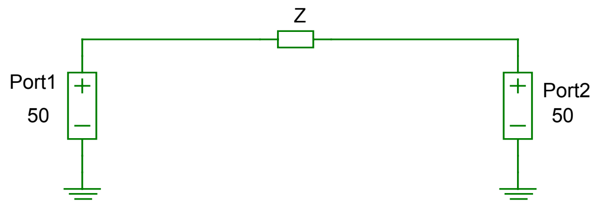


Fig. 1.3. Schematic figure of the shunt element, *Z*.

1.2.1 Calculate the S-parameters

Assume arbitrary admittance Z and calculate $S_{11}, S_{12}, S_{21}, S_{22}$.

- $\Gamma_1 = \frac{Y_0 - Y_1'}{Y_0 + Y_1'} = \frac{Z_1' - Z_0}{Z_1' + Z_0}$ where Z_1' is the input impedance ($Z+Z_0$)
- Use the relation between V_1 and V_2, which can be derived from elementary circuit theory.
- Calculate the scattering parameters up to 10 GHz using matlab assuming the impedance to be a resistance R = 20 Ω in series with an inductance L = 1 nH and a capacitance C = 1 nF.

1.2.2 Simulate the S-parameters

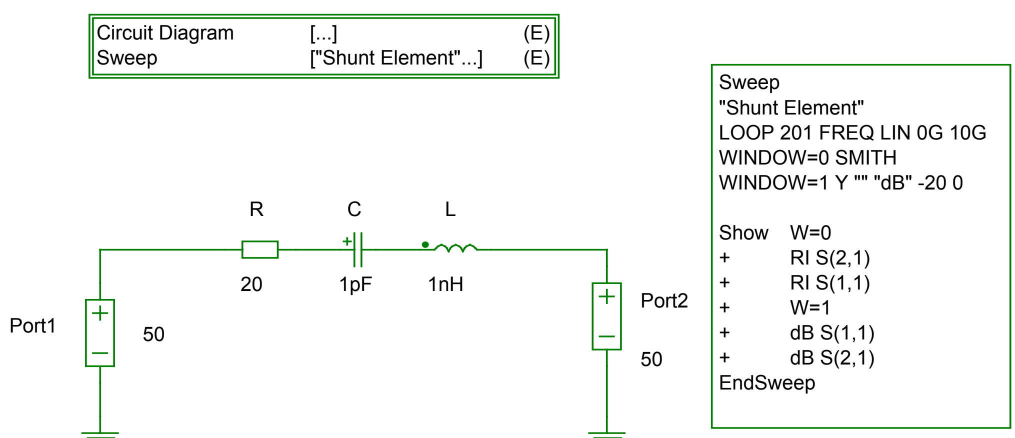


Fig. 1.4. Aplac file for the series element simulation.

- Simulate both with and without the resistance (R = 0 Ω) and investigate the influence of the resistance on the scattering parameters.

1.2.3 Verify the results

Verify both the calculations and measurements by printing the graphs of the scattering parameters achieved both from calculations and simulations. Present both the graphs in equal size with the same axis

All the tasks as in the previous exercise must be included in the report.

1.3 Series-Shunt element Combination

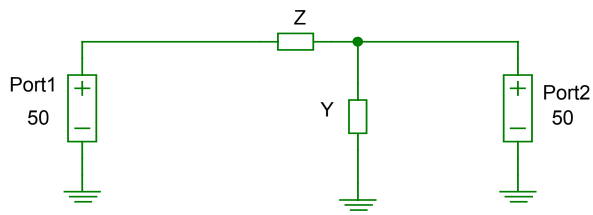


Fig. 1.5. Schematic figure of the series-shunt elements

1.3.1 Calculate the S-parameters

- Calculate the scattering parameters up to 10 GHz using matlab assuming the series impedance to be a resistance R = 20 Ω in series with an inductance L = 1 nH and a capacitance C = 1 nF. The shunt element is a resistance R = 150 Ω in parallel with an inductance L = 1 nH and a capacitance C = 1 nF.

1.3.2 Simulate the S-parameters

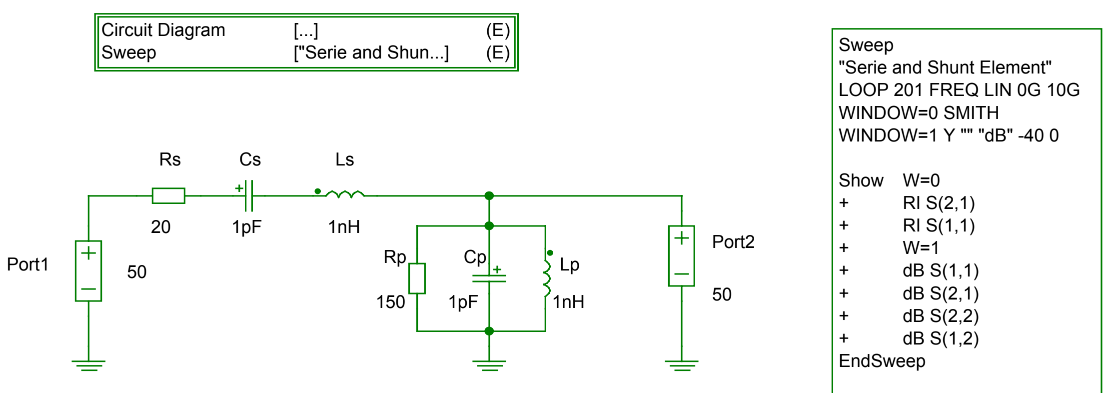


Fig. 1.6. Aplac file for the series-shunt element simulation

- Simulate the structure and study all of the scattering parameters.
- What is the main difference between this circuit and previous ones?
- How is this shown in the smith diagram?

1.3.3 Verify the results

Verify both the calculations and measurements by printing the graphs of the scattering parameters achieved both from calculations and simulations. Present both the graphs in equal size with the same axis

All the tasks as in the previous exercise must be included in the report.

2 Matching networks

This laboratory exercise aim to give an understanding how different matching networks can be realized for microwave purposes. Matching networks will be designed both with discrete (lumped) elements as well as using different wave-guides to establish matched conditions.

It should also be explained how impedance transformers can be realized for microwave applications, transforming the impedance of a resistive load.

2.1 Discrete element matching network

2.1.1 Matching of inductive load.

A load impedance Z_L (20 Ω in series with 0.32 nH) should be matched to a 50 Ω system at 5 GHz.

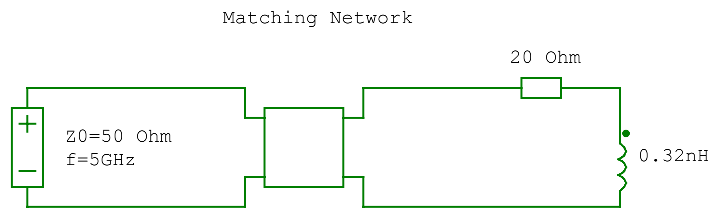


Fig. 2.1. Discrete component matching network for inductive load.

- Which types of L-C topologies can be used for the matching circuit?
- Why cannot other be used?
- Use discrete inductances and capacitances to realize a matching network for 5 GHz.
- Simulate the circuit to verify the solution.
- Realize the matching network with the other possible topology.
- Simulate this circuit also.
- Is there any difference in the performance?

2.1.2 Matching of capacitive load.

A load impedance Z_L (40 Ω in series with 1.59 pF) should be matched to a 50 Ω system at 2 GHz.

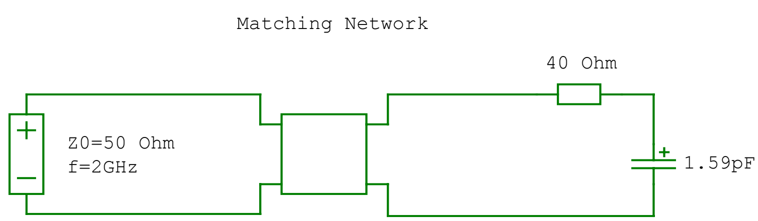


Fig. 2.2. Discrete component matching network for capacitive load.

- Which types of L-C topologies can be used for the matching circuit?
- Why cannot other be used?
- Use discrete inductances and capacitances to realize a matching network for 2 GHz.

- Simulate the circuit to verify the solution.
- Realize the matching network with the other possible topology.
- Simulate this circuit also.
- Is there any difference in the performance?

2.2 Stub matching network

2.2.1 Stub matching of capacitive load

The load Z_L (R = 90 Ω parallel with C = 4pF) should be matched to a Z_0 = 50 Ω impedance system using single stubs.

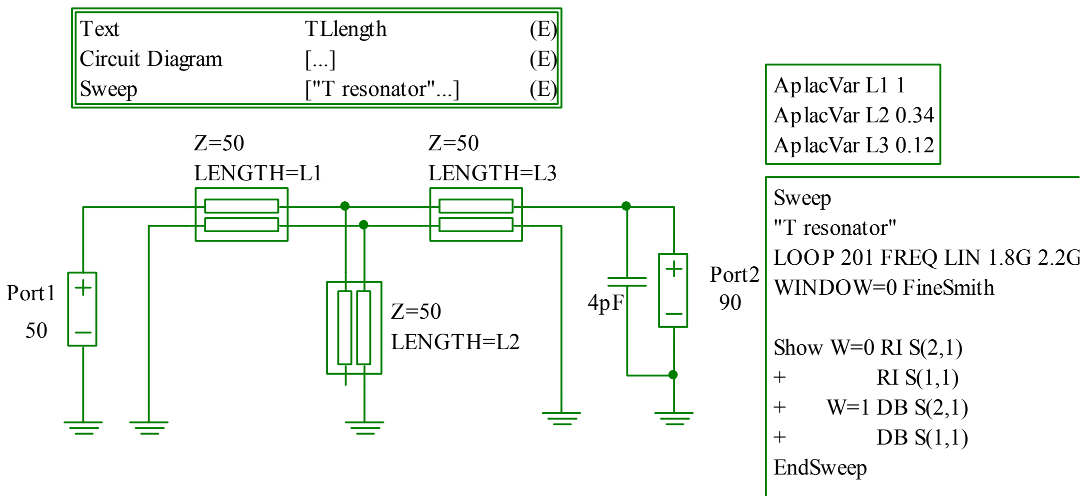


Fig. 2.3. Stub matching using transmission lines.

- Design an input stub match for the load at the frequency f = 2 GHz.
- Both an open and a shorten stub should be designed using smith diagrams
- Calculate the length l_1, l_2 and l_3 to achieve a matched.
    - The smith diagrams should be commented and so that they are easy to follow and included in the report
- Simulate both the designs in Aplac to verify the solutions.
- Assume that a shorted stub must be replace with an open one? How is this easiest accomplished? Verify this with an Aplac simulation.

2.2.2 Stub matching using microstrip technology

The same load as in the previous exercise should be matched on a circuit board using microstrip technology with Z_0 = 50 Ω. A 'Duroid' substrate with a copper conductor is to be used. The substrate parameters are to be found in Fig. 2.4..

- Design the microstrips fulfilling the matching requirements for both an open and a shortened stub. The impedance of the microstrips shold be Z_0 = 50 Ω.
    - Use the Aplac command 'Mlin_u' to calculate the width of the microstrips.
    - Use the Aplac command 'Mlin_epse' to calculate the effective permittivity (ε_r) of the microstrip.
    - The length of the microstrips is the calculated from the previous exercises taking ε_r into consideration.
- Simulate the performance of the two matching networks.
- Introduce a tee junction in the simulation in order to simulate a realistic solution. The stub matching degrades by introducing the tee junction.
    - Restore an optimized matching network by adjust the length or widths of the microstrips.
    - Use the built in optimization routines in Aplac or make a qualified guess how the dimensions should be altered.

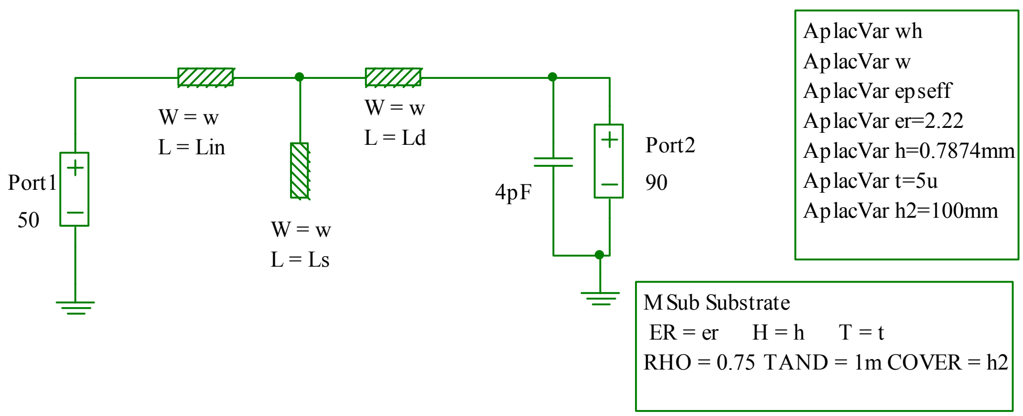


Fig. 2.4. Stub matching using micro strip technology.

2.3 Impedance matching

A resistive load R = 30 Ω should be matched to a 50 Ω system at f = 2 GHz using impedance transformers.

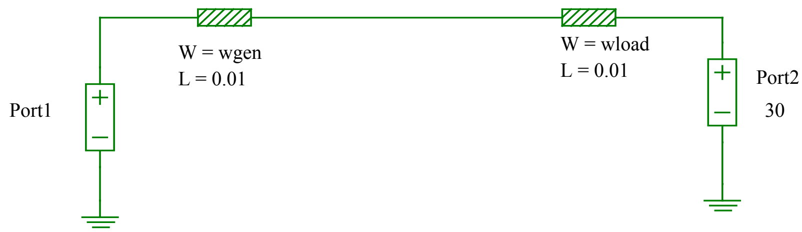


Fig. 2.5. 30 Ω load connected to a 50 Ω system.

- The same substrate as in the previous exercise should be used.
- Simulate the load connected directly to the generator without any matching network

2.3.1 Quarter wave impedance matching network

Design an impedance transformer using a quarter wavelength transformer.

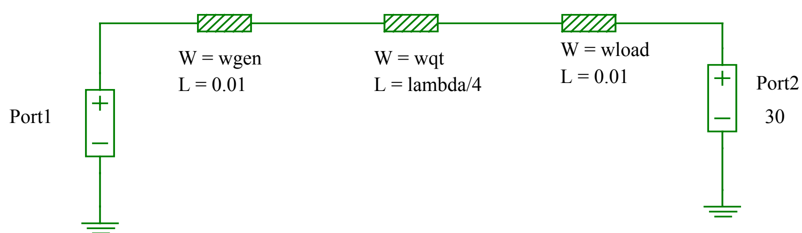


Fig. 2.6. Impedance transformation using a quarter wavelength transformer.

- Design a quarter wave transformer to transform 30 Ω to 50 Ω at 2 GHz.
- Simulate the performance of the design

2.3.2 Taper structure impedance matching

Use a taper structure to realize the impedance transformation.

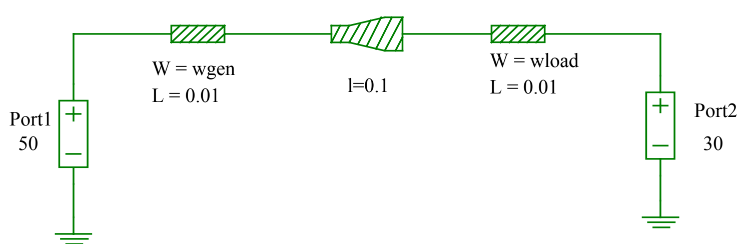


Fig. 2.7. Impedance transformation using a taper structure.

- Design a taper structure that transforms 30 Ω to 50 Ω at 2 GHz.
- Simulate the performance of the design
- What is the advantage with the taper structure over the quarter wavelength transformer?

3 Amplifier design

In this laboratory exercise you should design and characterize a microwave amplifier for the 1.8 GHz band. In the exercise you will need to match the bias the transistor correctly and match both the input and output simultaneously. In the exercise you will also be familiar with some different gain expression which are commonly used for microwave amplifiers.

3.1 Narrow-band design

A narrow-band bipolar amplifier for 1.8 GHz is to be designed. The input and output reflection should be less than -10 dB and the gain as large as possible. The device that should be used is Philips 25 GHz NPN-transistor BFG425W and the transistor model is implemented as a sub-circuit, which can be found in:

www.ite.mh.se/~kenber/labbar/hf2_vt02/BFG425W

3.1.1 Preliminary Design

- Design the schematic for the amplifier
  - To invoke the transistor choose the insert menu and select submodel.
- Design the biasing network for the amplifier using a standard common emitter layout.
  - Chose an operating point using the data sheet for the transistor.
  - Radio Frequency Coils (RFC) up to 100 nH and coupling capacitors up to 1 nF is allowed to be used in the biasing network.

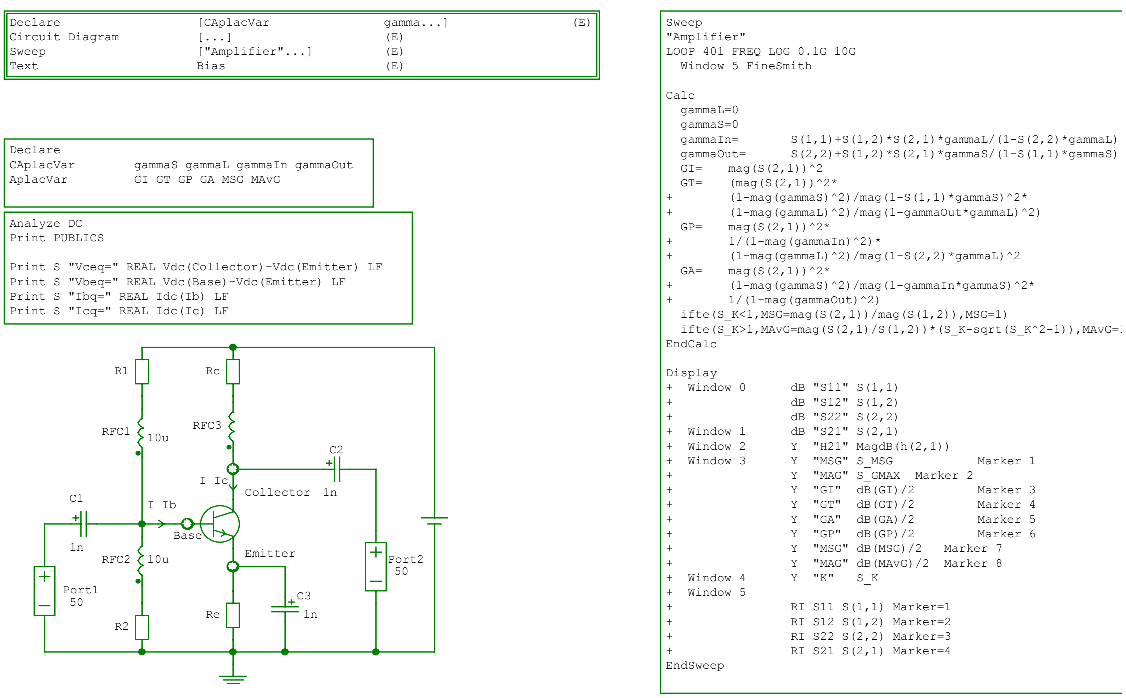


Fig. 3.1. Aplac file for extraction of scattering parameters and different gain expressions.

- Simulate the transistor and extract the scattering parameters between 0.1 and 10 GHz under the biasing conditions that has been calculated.
- Calculate the gain definitions listed below for the amplifier between 0.1 and 10 GHz.
  - Some important parameters and relations necessary for the gain calculations
    - Input and output reflection, Γ_{IN} and Γ_{OUT}.

$$\Gamma_{IN} = S_{11} + \frac{S_{12}S_{21}\Gamma_L}{1 - S_{22}\Gamma_L}$$

$$\Gamma_{OUT} = S_{22} + \frac{S_{12}S_{21}\Gamma_S}{1 - S_{11}\Gamma_L}$$

    - Source and Load reflections, Γ_S and Γ_L.
    - Stability factor, k. The stability factor is calculated in Aplac in the variable *S_K*.

$$k = \frac{1 + |S_{11}S_{22} + S_{12}S_{21}|^2 - |S_{11}|^2 - |S_{22}|^2}{2|S_{12}S_{21}|}$$

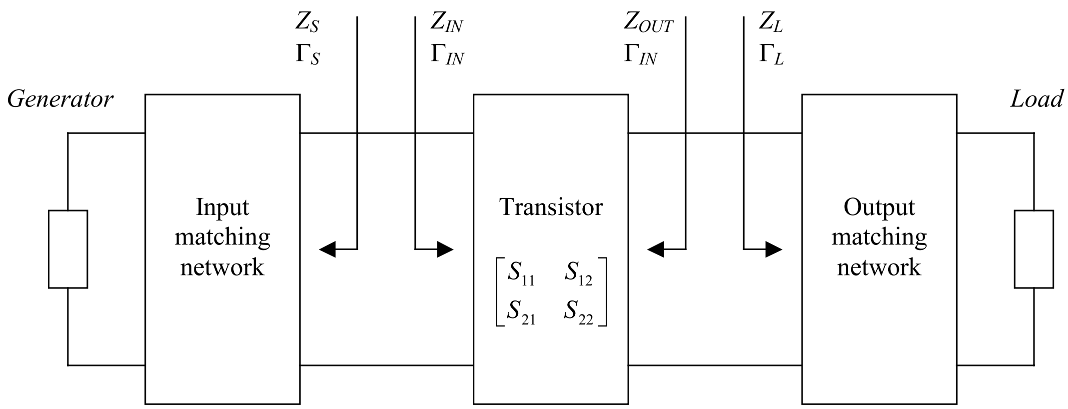


Fig. 3.2. Schematic representation of a microwave amplifier.

- Gain definitions
  - (Operating) Power Gain
    - The power gain, G_P, is simply the relation between the power delivered to the load and power feeded into the network.

$$G_P = \frac{P_L}{P_{IN}} = |S_{21}|^2 \frac{1 - |\Gamma_L|^2}{|1 - S_{22}\Gamma_L|^2 \left(1 - |\Gamma_{IN}|^2\right)}$$

  - Insertion Gain
    - The insertion gain G_{IN} is the comparison of the power delivered into a given load either connected directly to a given generator or via the amplifier, inserted between this generator and the given load.

$$G_{IN} = \frac{P_L}{P_L|_{\substack{Without\\Amplifier}}} = |S_{21}|^2$$

- Transducer Gain
  - The transducer gain G_T is a standard expression where the power delivered to a load is compared to the power that could be delivered directly by the generator under matched conditions.

$$G_T = \frac{P_L}{P_{AVG}} = |S_{21}|^2 \frac{\left(1-|\Gamma_S|^2\right)\left(1-|\Gamma_L|^2\right)}{|1-S_{11}\Gamma_S|^2 |1-\Gamma_{OUT}\Gamma_L|^2}$$

  - Assuming that there are no reflections in the generator or in the load ($\Gamma_S = \Gamma_L = 0$) the transducer gain (G_T) will be the same as the insertion gain (G_I).
- Available Gain
  - The available gain G_A is the transducer gain in the case where the output of the amplifier is matched

$$G_A = \frac{P_{AVL}}{P_{AVG}} = |S_{21}|^2 \frac{1-|\Gamma_S|^2}{|1-S_{11}\Gamma_S|^2 \left(1-|\Gamma_{OUT}|^2\right)}$$

- Maximum Available Gain
  - The maximum available gain *MAG* is the available gain in the case where the input is matched as well as the output.

$$MAG = \frac{P_{AVLopt}}{P_{AVG}} = \frac{S_{21}}{S_{12}}\left(k-\sqrt{k^2-1}\right) \quad \text{when k > 1}$$

- Maximum Stable Gain
  - The maximum stable gain *MSG* is the

$$MSG = \frac{P_{AVLopt}}{P_{AVG}} = \frac{S_{21}}{S_{12}} \quad \text{when k < 1}$$

  - *MSG* and *MAG* can be extracted directly form Aplac using the *S_MSG* command.

$$S\_MSG = \begin{cases} MSG & \text{when } k \le 1 \\ MAG & \text{when } k > 1 \end{cases}$$

3.1.2 Transistor Matching

Design a matching network for both the input and output using transmission lines using simultaneous conjugate match method.

- The simultaneous conjugate match method is described in chapter 3.6 in “Microwave Transistor Amplifier design”
- Calculate the required source and load reflections
- Use the smith-chart to calculate a stub-matching network that gives the desired reflections.
- Implement the solution in Aplac
  - Simulate the design and verify that the matching conditions are fulfilled.
  - Is the power gain fulfilling the expectations previously calculated?

3.1.3 Biasing network

- Improve the biasing network using the stub-endings as entry-points for biasing purposes.
    - It might be required to modify the amplifier design slightly in order to alter the biasing.
- Simulate the design and verify that the performance is not destroyed.

3.1.4 Microstrip implementation

- Convert the transmission lines to microstrips (Mlin)
    - Simulate the design and verify that the performance is maintained.
    -
- Add T-junctions (Mtee) between the microstrips
    - Restore an matched amplifier by optimizing design parameters

3.1.5 Stability

- Plot the stability parameters u (S_u), k (S_K) and Δ (S_D) over a wide range of frequencies (DC-100 GHz)
    - Determine frequencies that are not unconditionally stable.
- Plot stability circles for both the in- and output for the frequencies where the amplifier is potentially unstable.
    - S_CL, S_RL – Output stability center and radius.
        - If $|S_{11}| < 1$ the center of the smith chart represents a stable region.
        - If $|S_{11}| >1$ the center of the smith chart represents an unstable region.
    - S_CS, S_RS – Input stability center and radius.
        - If $|S_{22}| < 1$ the center of the smith chart represents a stable region.
        - If $|S_{22}| >1$ the center of the smith chart represents an unstable region.
- Calculate the constant available gain and constant power gain for 20 dB gain.
    - S_CA(gain), S_RA(gain) – Constant available gain circle.
    - S_CP(gain), S_RP(gain) – Constant power circle.

3.2 Wide-band Amplifier

In a narrow band amplifier the input and output reflection is low only in a very narrow interval. To improve the bandwidth a balanced amplifier should be designed using the amplifier constructed above and Lange couplers:

- Increase the bandwidth of the amplifier in the range from 1.8 to 2 GHz using optimization.
    - It might be necessary to adjust the impedance of the microstrips as well as the length in order to fulfill the specifications.
    - Don't care about the input and output reflection in the optimization.
- Create a sub-model of the improved amplifier that may be used in the balanced amplifier.
    - Replace "Circuit Diagram" with "DefModel <Name>"
    - Remove simulation information from the file.
    - Remove the ports from the schematic.
    - Place an input and output at appropriate places.
    - When saving the diagram a dialog box appears. Select all three boxes and press "save".

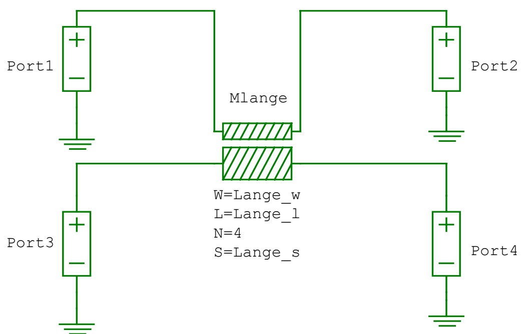
- Optimize a Lange coupler with maximal flat coupling characteristics between 1.8 and 2 GHz.
  - The coupling from port 1 to 2 (S_{21}) and port 1 to 4 (S_{41}) should both be close to –3 dB.
  - The Isolation between port 1 and 3 (S_{31}) should be below –30 dB.

```
Text                        Variabels                 (E)
Circuit Diagram             [...]                     (E)
Text                        Optimization              (E)
```

```
Aplacvar Lange_w 314u OPT MIN 0.1m  MAX 1m
AplacVar Lange_l 33.9m  OPT MIN 1m    MAX 100m
AplacVar Lange_s 11.6u  OPT MIN 0.05m MAX 1m

MSUB Substrate
+ER=2.22 H=0.7874m T=5u
+RHO=0.75 tand=1m cover=100m

Read OpenFile "OptVarLange.txt" OPT_VAR
```




```
OptimMethod Gradient

Sweep  "Lange Coupler"
  +LOOP 201 FREQ LOG 1.8G 2G
  +WINDOW=0 SMITH
  +WINDOW=1 Y "" "dB" -40 0 Grid

  Show W=0 RI S(2,1)
  +        RI S(1,1)
  +    W=1 Y MagdB(S(1,1)) Marker=1 Width=2 color=black
  +    W=1 Y MagdB(S(2,1)) Marker=2 Width=2 color=black
  +    W=1 Y MagdB(S(3,1)) Marker=3 Width=2 color=black
  +    W=1 Y MagdB(S(4,1)) Marker=4 Width=2 color=black

  Goal MagDB(S(2,1)) GT -3.3
  Goal MagDB(S(4,1)) GT -3.3
  Goal MagDB(S(1,1)) LT -35
  Goal MagDB(S(3,1)) LT -35
EndSweep

Print OpenFile "OptVarLange.txt" OPT_VAR

Print Optimize
```

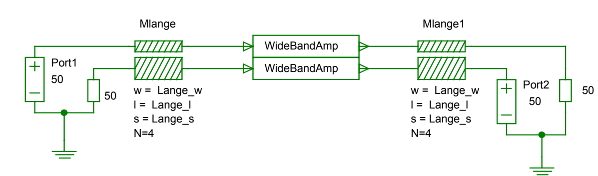
Fig. 3.3. Lange Coupler

- Design the balanced amplifier and simulate the design to verify the performance.
- What are the advantages with this configuration?
- Calculate theoretically both S_{21} and S_{11} for the balanced amplifier
  - Assuming ideal couplers with S_{11}=S_{31}=0, S_{21}=$\sqrt{2}\angle 30°$, S_{41}=$\sqrt{2}\angle -60°$
  - The coupler is symmetric S_{11}=S_{22}=.., S_{21}=S_{12}=S_{34}=S_{43}, S_{14}=S_{41}=S_{23}=S_{32},…
  - Assume two identical amplifiers with constant S_{21} and non-zero S_{11}.

```
Text                Substrate               (E)
Circuit Diagram     [...]                   (E)
Text                PrintBias               (E)
Sweep               ["Optimized Ampl...]    (E)
Sweep               ["Stability Swee...]    (E)
```

```
Sweep
"Optimized Amplifier"
LOOP 501 FREQ LIN 1.8G 2.2G
  Window 2 FineSmith
```

```
Sweep
"Stability Sweep"
LOOP 25 FREQ LOG 1k 2G
Window 0 FineSmith
Window 1 FineSmith
```




```
Declare AplacVar
+ er h t h2 w wh epseff lambda dS k MAvG
+ Lange_w Lange_l Lange_s

Read  OpenFile "OptVarLange.txt" AplacVars

Calc
 er=2.22
 h=0.7874mm
 t=5u
 h2=100mm
EndCalc
MSub Substrate
+ ER = er     H = h     T = t
+ RHO = 0.75 TAND = 1m COVER = h2
Calc
  wh=Mlin_u(50,er)
  w=wh*h
  epseff=Mlin_epse(wh,t/h,h2/h,er)
  lambda=3e8/sqrt(epseff)/10e9
EndCalc
Print S "w=" REAL w LF
```

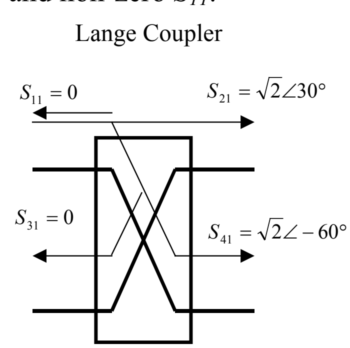


Fig. 3.4. Balance Amplifier