# Using ADS to simulate Noise Figure

ADS can be used to design low noise amplifiers much in the same way you have already used it for MAG or MSG designs. Noise circles and available gain circles are the tools that give the most guidance on design tradeoffs. Refer to Chap. 4 and Appendices K and L of Gonzalez for the theory behind these analyses. The ADS files shown below are available from the course web page as a .zap file.

Here are 3 cases that you might encounter with device models when analyzing a low noise amplifier.

1. The ADS large signal transistor model is used to represent the device. This is the ideal case, but unfortunately, the large signal models sometimes do not produce accurate S parameters. If you use this model, you should check the simulated S parameters with the manufacturer's data sheet to verify that it provides reasonably accurate results. It should be useful for DC simulations however.

2. The ADS S-parameter transistor model is used to represent the device. This is the most accurate case. Of course, no DC simulations will be possible with this model, but it will represent the S parameters and noise parameters accurately. Be sure to select the model that represents the actual bias condition to be used in your analysis.

3. No S parameter model is available for the device in the ADS library. In this case, check the manufacturer's web site and download an S2P file for the device. Place an S2P block from the Data Items menu in your schematic and identify the file name. ADS will look for the file in the ADS project's data directory. See the appendix at the end of this tutorial for more information on S2P files. In this case, you may have to enter the noise parameters as well using equations on the data display panel.

Case 1. Using ADS large signal model library.

First you must bias the transistor. Let's say VCE = 5V and IC = 5 mA. This bias condition might be selected from the plot of noise figure vs. bias condition provided by the manufacturer.

NE85635
NOISE FIGURE
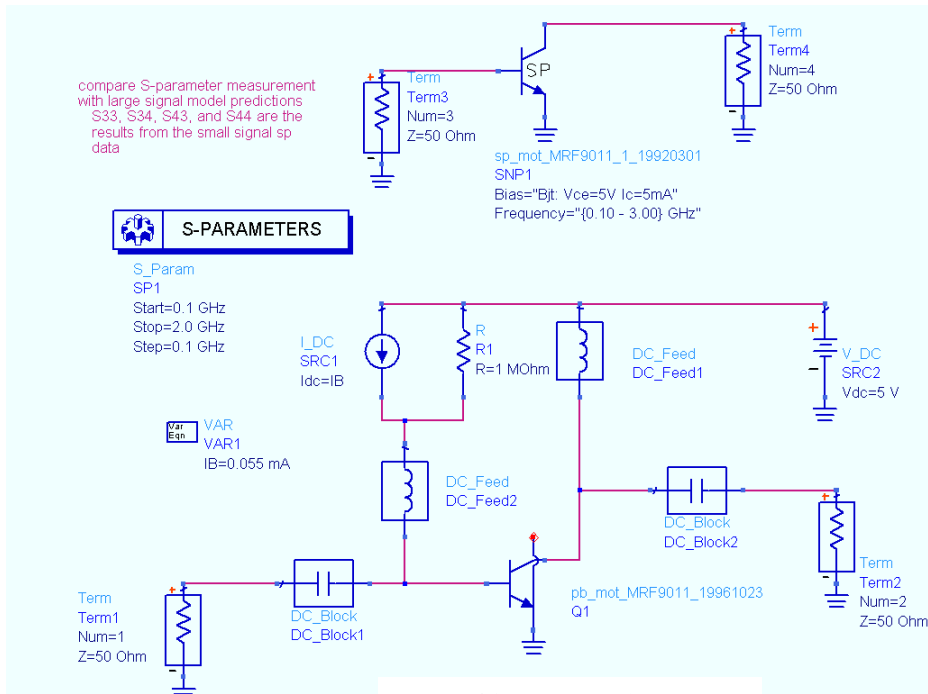## vs. COLLECTOR CURRENT
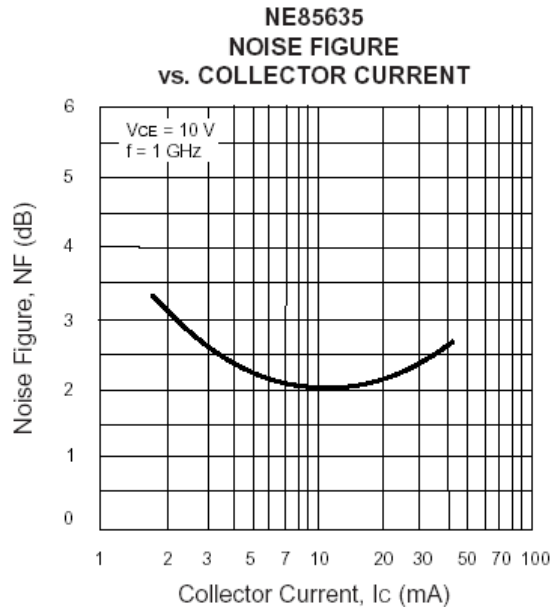
Set up a biasing circuit such as the one below. Select a large signal device model from the Analog/RF – RF Transistor/Packaged BJT library. Then perform a DC simulation. To see the results of the DC simulation, you go to the Simulate Menu > Annotate DC solution. Sometimes it is helpful to move component text aside so that the annotation is easier to read. Use F5 to move text.
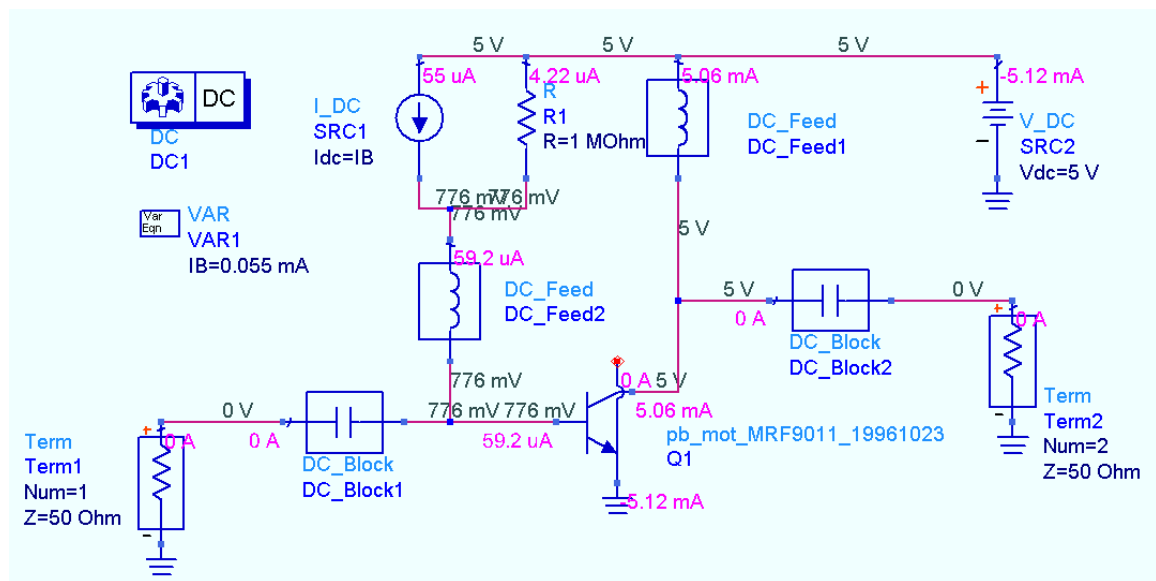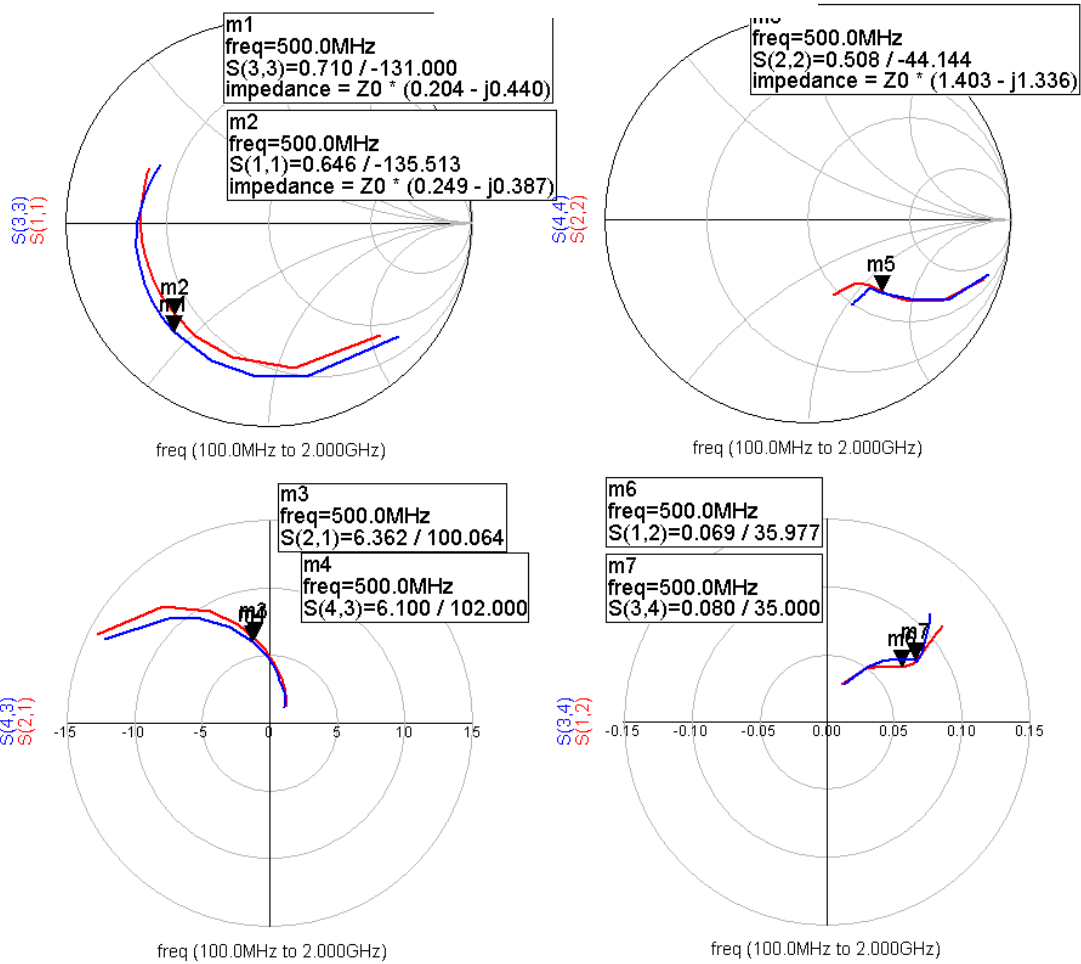

This is the result. We have $V_{CE} = 5V$ and $I_C = 5.06$ mA. (MRF901_DC schematic file)

Next, compare the S parameters from the large signal model with those from the small signal measured S parameter model. This model can be found in the Analog/RF S Parameter component library. Simulate over the specified frequency range.

compare S-parameter measurement
with large signal model predictions
S33, S34, S43, and S44 are the
results from the small signal sp
data

Term
Term3
Num=3
Z=50 Ohm

SP

Term
Term4
Num=4
Z=50 Ohm

sp_mot_MRF9011_1_19920301
SNP1
Bias="Bjt: Vce=5V Ic=5mA"
Frequency="{0.10 - 3.00} GHz"

S-PARAMETERS

S_Param
SP1
Start=0.1 GHz
Stop=2.0 GHz
Step=0.1 GHz

I_DC
SRC1
Idc=IB

R
R1
R=1 MOhm

DC_Feed
DC_Feed1

V_DC
SRC2
Vdc=5 V

VAR
VAR1
IB=0.055 mA

DC_Feed
DC_Feed2

DC_Block
DC_Block2

Term
Term1
Num=1
Z=50 Ohm

DC_Block
DC_Block1

pb_mot_MRF9011_19961023
Q1

Term
Term2
Num=2
Z=50 Ohm

(MRF901_sparamtest)

m1
freq=500.0MHz
S(3,3)=0.710 / -131.000
impedance = Z0 * (0.204 - j0.440)

m2
freq=500.0MHz
S(1,1)=0.646 / -135.513
impedance = Z0 * (0.249 - j0.387)

m5
freq=500.0MHz
S(2,2)=0.508 / -44.144
impedance = Z0 * (1.403 - j1.336)

S(3,3)
S(1,1)

m2
m1

S(4,4)
S(2,2)

m5

freq (100.0MHz to 2.000GHz)

freq (100.0MHz to 2.000GHz)

m3
freq=500.0MHz
S(2,1)=6.362 / 100.064

m4
freq=500.0MHz
S(4,3)=6.100 / 102.000

m6
freq=500.0MHz
S(1,2)=0.069 / 35.977

m7
freq=500.0MHz
S(3,4)=0.080 / 35.000

S(4,3)
S(2,1)

m3
m4

-15   -10   -5   0   5   10   15

S(3,4)
S(1,2)

m6
m7

-0.15   -0.10   -0.05   0.00   0.05   0.10   0.15

freq (100.0MHz to 2.000GHz)

freq (100.0MHz to 2.000GHz)

From this simulation, we can see that the agreement at our design frequency of 500 MHz is fairly good. S12 is off by about 20%, which will have an effect on the predictions of MSG, but the other parameters fit pretty well at this frequency. Let's assume this is sufficient. Now we must test for stability.

Load plane stability circle

Source plane stability circle
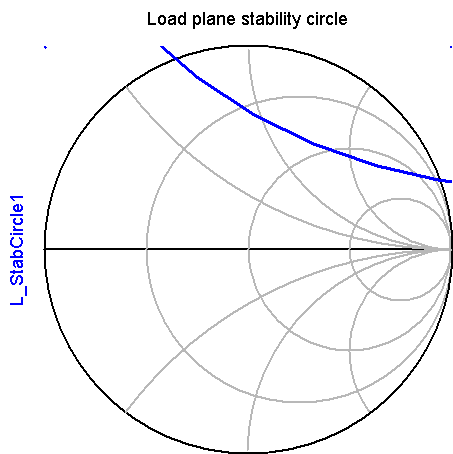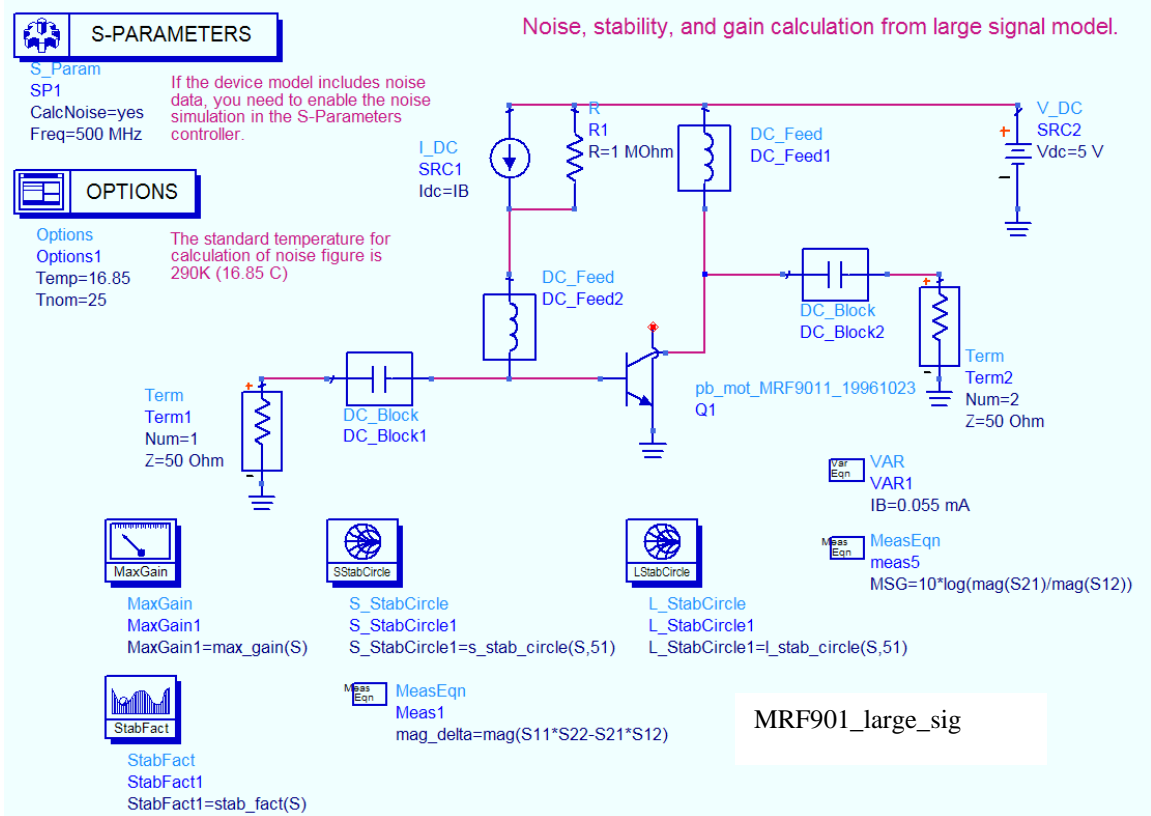
freq	mag_delta	StabFact1
500.0MHz	0.299	0.485
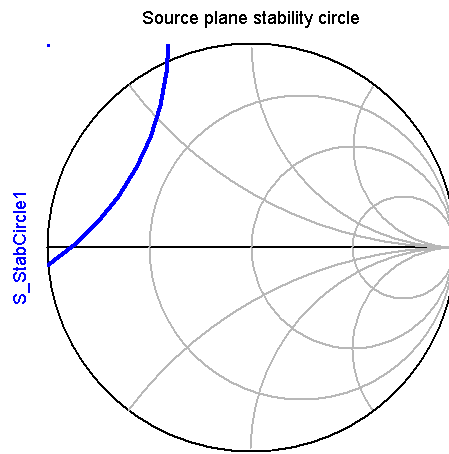
freq	MSG	MaxGain1
500.0MHz	19.769	19.769

(Display file: stability.dds)

We can see that the device is potentially unstable, but if we are careful and can give up some gain, we may be able to find a stable solution and still retain low noise figure. This may give us lower noise figure than if we first make the device unconditionally stable with resistive loading. To evaluate this possibility, we need to look at the available gain and noise figure circles. It is more convenient to calculate the gain and noise circles (Available gain for noise calculations) on the display panel rather than the schematic panel so you can change the noise figure without having to resimulate the circuit.

ADS can be used to calculate the noise parameters from the large signal transistor models. You must first enable the S Parameter controller icon for noise simulation. You will generally want to calculate noise parameters at one frequency, so use the Single Point Sweep option on the frequency panel. Also choose Options and set the temperature to 16.85 degrees (standard temperature for noise is 290K).

Scattering-Parameter Simulation:1

S_Param Instance Name

SP1

| Frequency | Parameters | Noise | Display |

☑ Calculate noise

OPTIONS

Options
Options1
Temp=16.85
(Standard temperature for noise analysis)
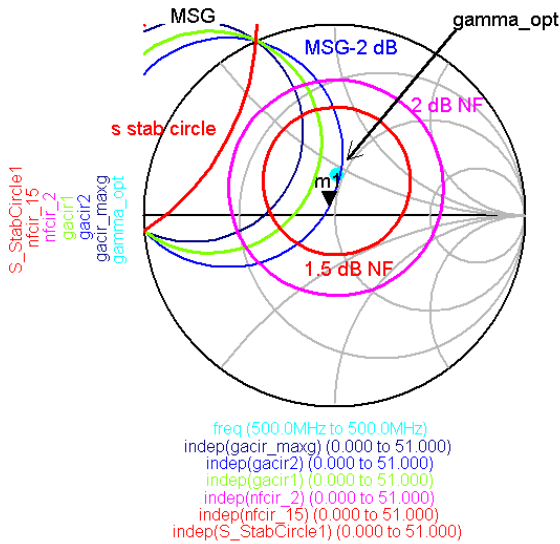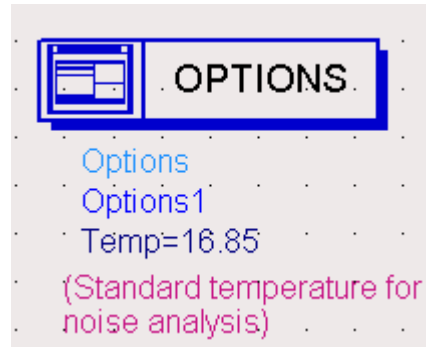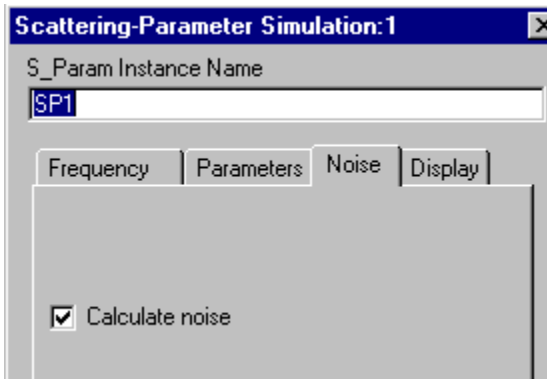
```
m1
indep(m1)=47
gacir2=0.052 / 122.559
freq=500.0000MHz
impedance = Z0 * (0.942 + j0.083)
```

freq	NFmin	Rn	Sopt
500.0MHz	1.167	7.560	0.213 / 86.426

freq	MaxGain1	MSG
500.0MHz	19.769	19.769

calculate available gain circles relative to MSG or MAG:

Eqn gacir_maxg=ga_circle(S,MaxGain1,51)

Eqn gacir1=ga_circle(S,MaxGain1-1,51)

Eqn gacir2=ga_circle(S,MaxGain1-2,51)

Eqn gacir3=ga_circle(S,MaxGain1-3,51)

Eqn gamma_opt=Sopt

calculate noise circles at 1.5 and 2.0 dB

Eqn nfcir_15=ns_circle(1.5,NFmin,gamma_opt,Rn/50,51)

Eqn nfcir_2=ns_circle(2,NFmin,gamma_opt,Rn/50,51)

MSG
MSG-2 dB
gamma_opt
2 dB NF
s stab circle
m1
1.5 dB NF

S_StabCircle1
nfcir_15
nfcir_2
gacir1
gacir2
gacir_maxg
gamma_opt

freq (500.0MHz to 500.0MHz)
indep(gacir_maxg) (0.000 to 51.000)
indep(gacir2) (0.000 to 51.000)
indep(gacir1) (0.000 to 51.000)
indep(nfcir_2) (0.000 to 51.000)
indep(nfcir_15) (0.000 to 51.000)
indep(S_StabCircle1) (0.000 to 51.000)

Eqn Zopt=50*(1+gamma_opt)/(1-gamma_opt)

| freq	Zopt
500.0MHz	46.855 + j20.854

(Display file: circles.dds   Use page menu to select gain and noise circles page)

The syntax used in the available gain and noise circle equations will be discussed in Section 2. However, you can see in this example, that the MSG is high (19.8 dB), and if we give up 2 dB in gain, a perfect noise source match can be achieved. The noise parameters for the transistor under the given bias condition are shown in the table. These were calculated from the large signal model.

Now, you could go back and vary the bias to determine whether you could improve on gain or noise, or you could complete the design by determining matching networks, biasing and wideband stability. Completing the design follows the same procedure described below in Section 2.
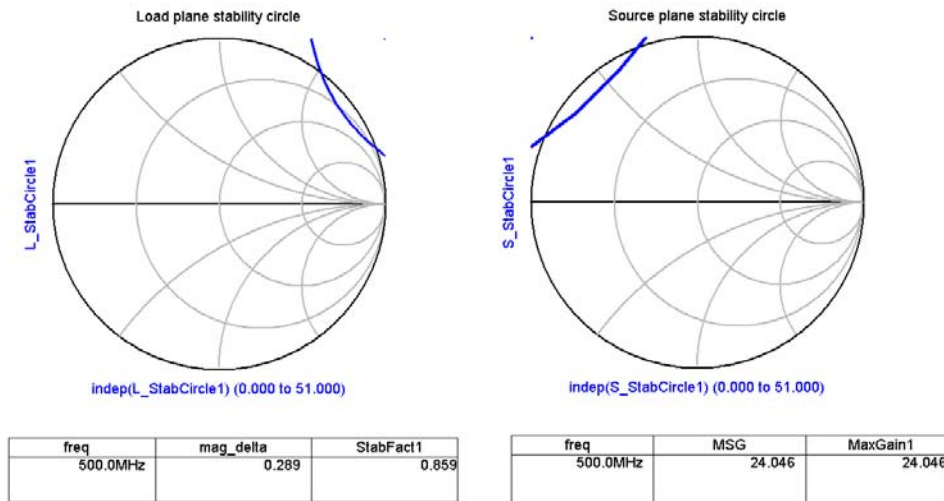
2. Simulation from S Parameter Model Library

The next example illustrates how the noise figure simulations can be carried out using the S Parameter Model Library in ADS. Choosing a different device, we start again by evaluating stability at the design frequency of 500 MHz.
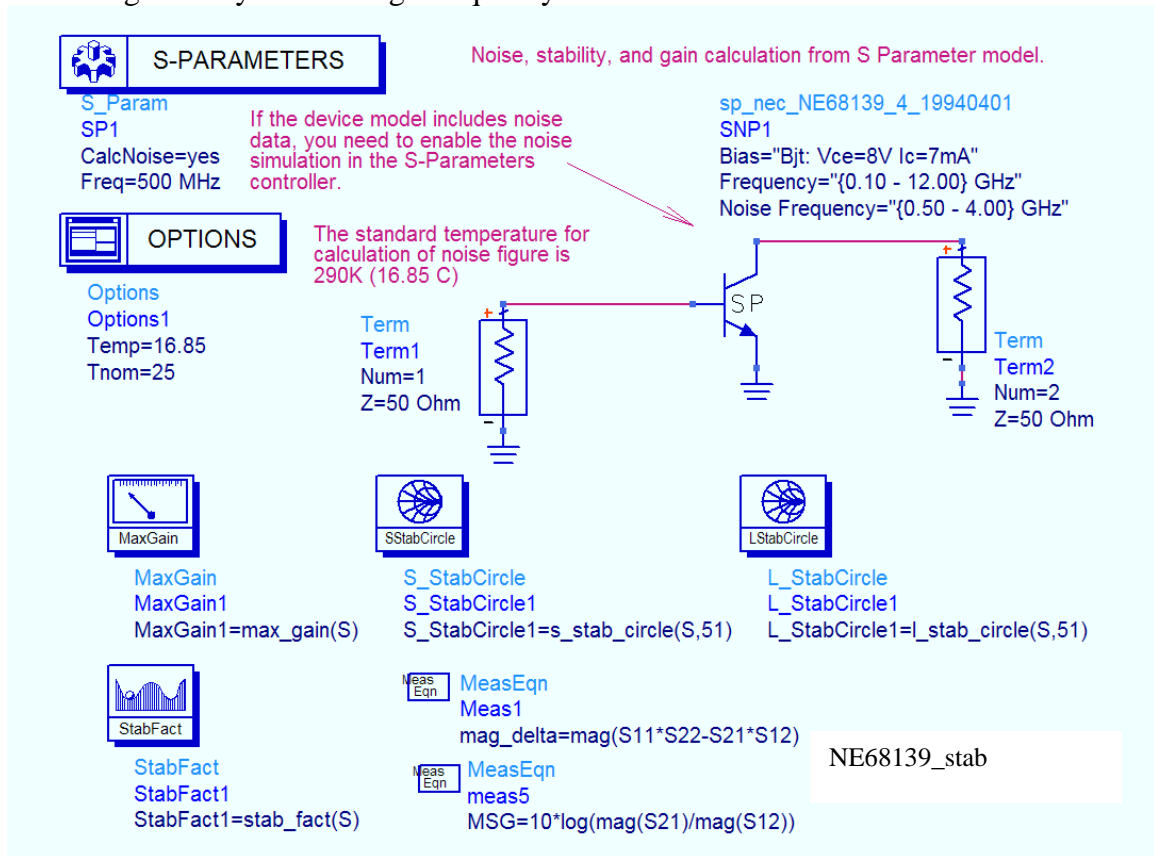
If the device model includes noise data as this one does in the example above, enable the noise simulation in the S Parameter controller. This will then calculate NFmin, Γopt (called Sopt), and Rn (un-normalized). Also include the maximum gain (MaxGain1) equation (calculates either Maximum Available Gain MAG or Maximum Stable Gain, MSG) and stability circle equations on the schematic panel. You must first determine stability before proceeding with the noise analysis. Since this is a low noise design, we

will try first for a solution that does not require resistive stabilization which would degrade the noise figure.

Load plane stability circle

Source plane stability circle

indep(L_StabCircle1) (0.000 to 51.000)

indep(S_StabCircle1) (0.000 to 51.000)

freq	mag_delta	StabFact1
500.0MHz	0.289	0.859

freq	MSG	MaxGain1
500.0MHz	24.046	24.046

(Display file: stability.dds)

We can see that the device is potentially unstable, but if we are careful and can give up some gain, we may be able to find a stable solution and still retain low noise figure. To evaluate this, we need to look at the available gain and noise figure circles. The noise parameters needed for noise figure calculation are not included in the S Parameter model library, so they must be added to the display panel. It is more convenient to calculate the gain and noise circles (Available gain for noise calculations) on the display panel rather than the schematic panel so you can change the noise figure without having to resimulate the circuit.

The syntax for calculating available gain circles (ga_circle equation) is the same as that for power gain circles. It is usually convenient to plot the gains relative to the MaxGain1 value. The Gav circles must be plotted on the source plane. They assume that the load is conjugately matched to the output for all Γ_S values.

The noise circle parameters (for the ns_circle equation) are defined as:

      ns_circle(nf, NFmin, gamma_opt, rn, 51)

nf = noise figure of the device represented by Γ_S values that fall on the circle.

NFmin = minimum possible noise figure of device at this bias and frequency

Gamma_opt = Γopt = the optimum input match for best noise figure

rn = the noise resistance parameter (normalized to 50 ohms)

51 = number of points plotted on the circle

If the transistor model does not include noise data (not all of them do), you must enter the NFmin, gamma_opt, and rn from the transistor data sheet manually using equations (Eqn) in order to calculate noise circles.

m1
indep(m1)=30
gacir3=0.849 / 171.825
freq=500.0000MHz
impedance = Z0 * (0.082 + j0.071)

MSG
MSG-2 dB
2 dB NF
1.5 dB NF
gamma_opt
m1
MSG-3

gacir3
S_StabCircle1
nfcir_15
nfcir_2
gacir1
gacir2
gacir_maxg
gamma_opt

freq (500.0MHz to 500.0MHz)
indep(gacir_maxg) (0.000 to 51.000)
indep(gacir2) (0.000 to 51.000)
indep(gacir1) (0.000 to 51.000)
indep(nfcir_2) (0.000 to 51.000)
indep(nfcir_15) (0.000 to 51.000)
indep(S_StabCircle1) (0.000 to 51.000)
indep(gacir3) (0.000 to 51.000)

Eqn Zopt=50*(1+gamma_opt)/(1-gamma_opt)

freq	Zopt
500.0MHz	68.442 + j25.541

Noise parameters calculated from device model

freq	NFmin	Rn	Sopt
500.0MHz	1.150	8.500	0.260 / 42.000

if noise parameters are all equal to 0, then the transistor model does not include noise data. You must then use equations (Eqn) to manually enter NFmin, rn, and Sopt
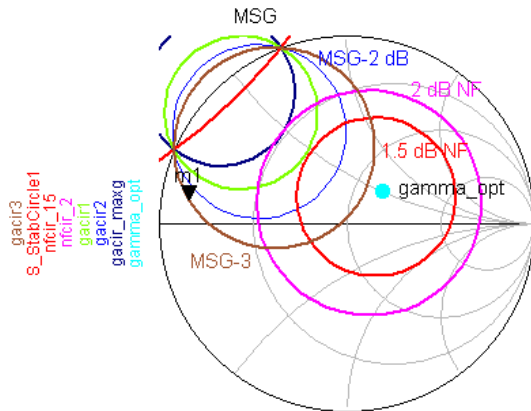
freq	MaxGain1	MSG
500.0MHz	24.046	24.046

calculate available gain circles relative to MSG or MAG:

Eqn gacir_maxg=ga_circle(S,MaxGain1,51)

Eqn gacir1=ga_circle(S,MaxGain1-1,51)

Eqn gacir2=ga_circle(S,MaxGain1-2,51)

Eqn gacir3=ga_circle(S,MaxGain1-3,51)

Eqn gamma_opt=Sopt   **Eqn** rn=Rn/50

calculate noise circles at 1.5 and 2.0 dB

Eqn nfcir_15=ns_circle(1.5,NFmin,gamma_opt,rn,51)
Eqn nfcir_2=ns_circle(2,NFmin,gamma_opt,rn,51)

(Display file: circles.dds   Use page menu to select gain and noise circles page)

In this example, you can see that MSG is quite high: 24 dB at 500 MHz. If we mismatch the input such that we give up 3 dB of available gain (brown gain circle), we can come quite close to the minimum noise figure of the device (1.15 dB). In fact, a $\Gamma_S = 0$ (very simple match!) could give a NF somewhere between 1.15 and 1.5 dB.
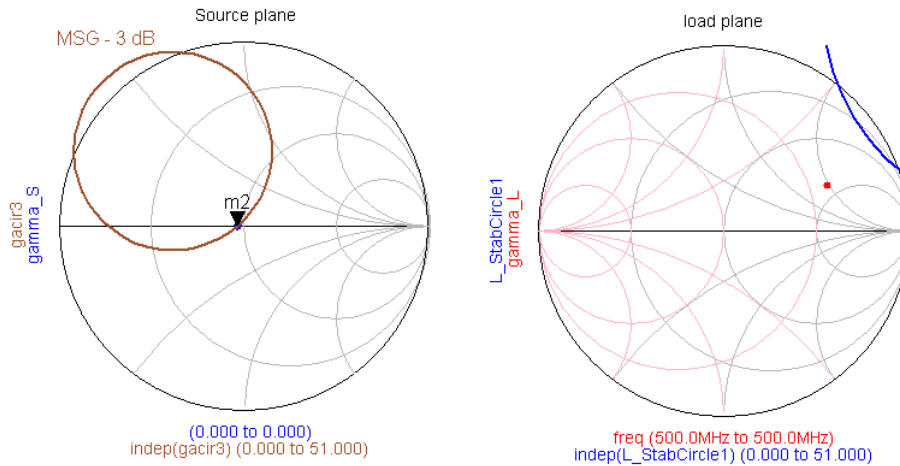
Once you have decided on the Γ_S value, place the marker there on the display. You need to design matching networks for the source and load. Calculate the corresponding Γ_L with the usual equation as shown below. Using the circles data display file, use the page menu to select the Gamma L page. Check the load stability circle to make sure Γ_L does not fall in an unstable region, and design the output matching network to provide Γ_L.

Eqn gamma_S=m2[0]

Eqn gamma_L=conj(S22+(S12*S21*gamma_S/(1-S11*gamma_S)))

```
m2
indep(m2)=44
gacir3=0.037 / -173.696
freq=500.0000MHz
impedance = Z0 * (0.929 - j0.008)
```

freq	gamma_L	gamma_S
500.0MHz	0.613 / 23.683	0.037 / -173.696

Source plane

MSG - 3 dB

gacir3
gamma_S

m2

(0.000 to 0.000)
indep(gacir3) (0.000 to 51.000)

load plane

L_StabCircle1
gamma_L

freq (500.0MHz to 500.0MHz)
indep(L_StabCircle1) (0.000 to 51.000)

(Display file: circles.dds   Use page menu to select gamma L page)

Then, design the matching networks, provide for bias insertion, and simulate the amplifier over a range of frequencies to verify stability out of band just as you did in the Stability and Gain tutorial.

Appendix 1: Representing devices as an S2P file

In the case of the NE34018, the ADS component library does not contain a set of S parameters for the device. Instead, we can use an S2P file provided by NEC. The file for our device was measured at the VDS = 2V; ID = 5 mA bias point. An S2P file is simply a tab delimited ASCII text file containing S parameters measured on a 2 port device. The format is:

```
Frequency     S11              S21              S12              S22
In GHz    mag    ang     mag    ang     mag    ang     mag    ang
```
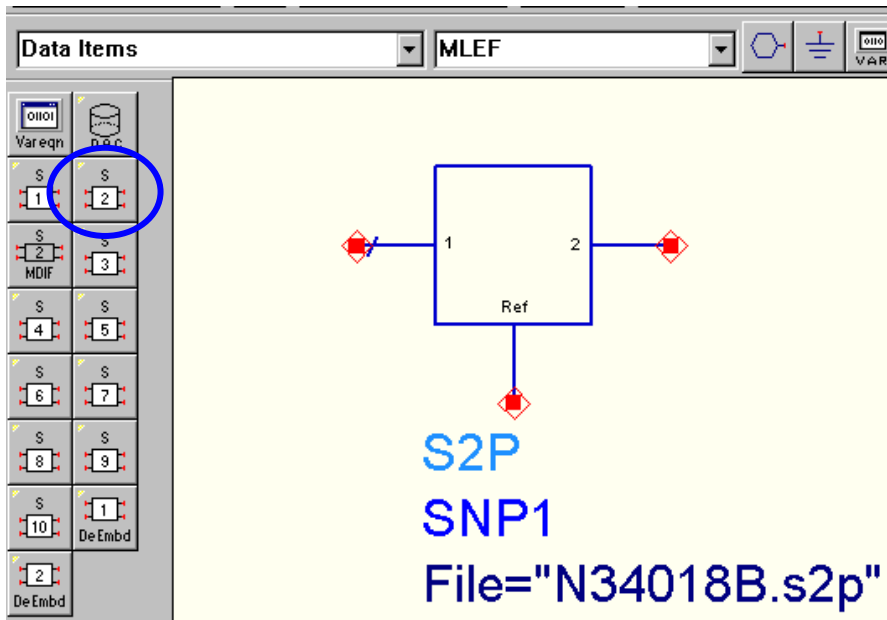
Place an S2P block from the Data Items menu and identify the file name. ADS will look for the file in the ADS project's data directory.