Chapter 10: Microwave Amplifier Design

Article originally published in Microwave Product Digest, January 2006 issue with title Amplifier Design Made Easy. Reproduced with permission

ADS Licenses Used:

- **Linear Simulation**

Chapter 10: Microwave Amplifier Design

Abstract

The purpose of this paper is to help designers understand a simple procedure to design and develop Amplifiers. There are many text books available on the Amplifier theory and design but they always leave a gap between theory and practical considerations that should be understood by a designer to produce a good amplifier circuit that compares well with the simulated data so that minimal or almost no post production tuning is required for the Amplifier. This paper tries to collect basic theory of Amplifier design as well as the practical procedure that needs to be adopted for making design first time success so that designers could save their time and efforts. This paper focuses more on CAD aided design procedure to design amplifiers because CAD software has become a necessity for a design house to design accurately and shorten time to market. The design process utilized in this paper makes use of Agilent Advanced Design System (ADS) software.

1. Introduction:

Amplifier is integral part of any communication system. The purpose of having an amplifier in a system is to boost the signal to the desired level. It also helps in keeping the signal well above noise so that it could be analyzed easily and accurately. Choice of amplifier topology is dependent on the individual system requirements and they could be designed for Low Frequency applications, medium to high frequency applications, mm-wave applications etc. and depending upon the system in which they are used, amplifiers can adopt many design topologies and could be used at different stages of system and accordingly they are classified as Low Noise Amplifiers, Medium Power Amplifiers, and Power Amplifiers etc. The most common structure that still finds application in many systems tends to be a Hybrid MIC amplifier. The main design concepts for amplifiers regardless of frequency and system remain the same and they need to be understood very clearly by a designer. Specific frequency ranges pose their own unique design challenges and needed to taken care by designers appropriately. This paper focuses on design of a small signal C-band Hybrid MIC amplifier. This procedure is equally valid for other amplifiers operating in other frequency ranges with minor changes in the design procedure.

2. Amplifier Theory:

There are few things that need to be understood by designer before he can start designing amplifiers like stability and matching conditions. These are discussed in the section below, there are many references available on amplifier basic concept and design procedure, the formula presented in this paper are taken from one of them [1]:

2.1 Stability Condition:

Stability analysis is the first step in any amplifier design. The stability of an amplifier or its resistance to oscillate is an important consideration in a design and can be determined from S-parameters, the matching networks, and the terminations. In a two-port network, oscillations are possible when either the input or output port presents a negative resistance [1]. This occurs when $|\Gamma_{IN}| > 1$ or $|\Gamma_{OUT}| > 1$, which for a unilateral device occurs when $|S_{11}| > 1$ or $|S_{22}| > 1$.

Unconditional stability of the circuit is the goal of the amplifier designer. Unconditional stability means that with any passive load presented the input or output of the device, the circuit should not become unstable i.e. it will not oscillate. In general, for a linear 2-port device characterized by S-parameters, the two necessary and sufficient conditions to guarantee unconditional stability are a) K >1 and b) $|\Delta| < 1$, where

$$\Delta = S_{11}S_{22} - S_{21}S_{12}$$

$$\mathbf{K = (1 + |\Delta|^2 - |S_{11}|^2 - |S_{22}|^2) / (2\ |S_{21} S_{12}|)}$$

2.2 Matching Conditions:

Amplifier could be matched for variety of conditions like Low Noise applications, unilateral case and bilateral case. The formulae for each condition are given below for designer's knowledge [1].

Optimum Noise Match:

The matching for lowest possible noise figure over a band of frequencies require that particular source impedance be presented to the input of the transistor. The noise optimizing source impedance is called as $\Gamma_{opt,}$ and is obtained from the manufacturers data sheet. The corresponding load impedance is obtained from the cascade load impedance formula.

$$\Gamma_L = \left(\frac{S_{22} - \Gamma_{opt}\Delta}{1 - \Gamma_{opt} S_{11}} \right)^*$$

Unilateral Case:

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

$$\Gamma_{OUT} = S_{22} + \left(\frac{S_{12} S_{21} \Gamma_S}{1 - S_{11} \Gamma_S} \right)$$

Bilateral Case (when $S_{12} \neq 0$):

$$\Gamma_{Ms} = \frac{B_1 \pm \sqrt{B_1^2 - 4|C_1|^2}}{2C_1}$$

$$\Gamma_{ML} = \frac{B_2 \pm \sqrt{B_2^2 - 4|C_2|^2}}{2C_2}$$

$$B_1 = 1 + |S_{11}|^2 - |S_{22}|^2 - |\Delta|^2$$

$$B_2 = 1 + |S_{22}|^2 - |S_{11}|^2 - |\Delta|^2$$

$$C_1 = S_{11} - \Delta S_{22}^*$$

$$C_2 = S_{22} - \Delta S_{11}^*$$

The common source configuration is normally chosen for the highest gain per stage. If the stability factor K>1, the network gives MAG. If K<1, the network could cause oscillations i.e. Gmax is infinite and given as

$$G_{max} = \frac{S_{21}}{S_{12}} \left(k - \sqrt{(k^2 - 1)} \right)$$

This should be avoided by locating the region of instability in Γ_S and Γ_L planes.

3. CAD oriented design procedure:

The CAD oriented design procedure consists of following steps, which will be described one by one for reference and understanding of the designers.

3a). DC Analysis

3b). Bias circuit design

3c). Stability analysis

3d). Input and Output matching network design

3e). Overall Amplifier performance optimization

Amplifier Specifications:

Frequency Band: 5.3 GHz – 5.5 GHz

Gain: 12 dB or more

Gain Flatness: +/- 0.25 dB (max.)

Input/Output Return Loss: < -15 dB

DC Power Consumption: 50 mW (max.)

3a). DC Analysis:

Based on the frequency range and the gain requirement CFY67-08 HEMT device was selected for the present amplifier design. The first analysis that needs to be performed is the DC analysis to find out the right bias points for the amplifier. Fig. 5.1 shows the DC analysis setup and Fig. 5.2 shows the DC analysis results for the same. Based on the DC Power consumption and Gm requirement, bias points are selected as Vgs=-0.1V and Vds=3V. ***To get DC IV sweep setup shown below, please click on Insert->Template->FET Curve Tracer, insert the active device (FET) and connect the proper wires and modify the Sweep parameters as per the device selected.***

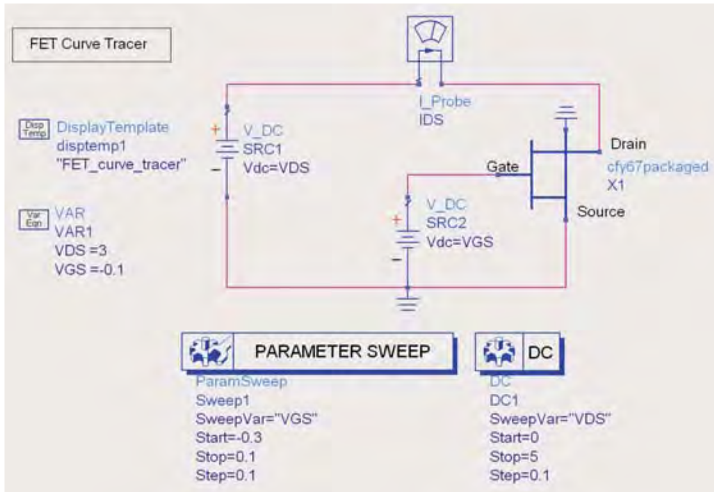

Fig. 5.1 DC Analysis setup for CFY67-08

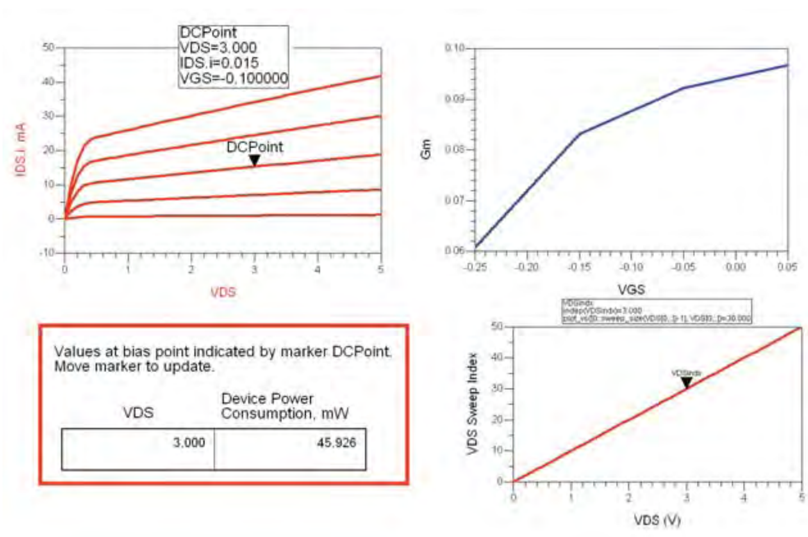

Fig. 5.2 DC Analysis Results

3b). Bias network design:

Bias network design for amplifier is dependent on the frequency range in which amplifier needs to be designed, that is to say if amplifier needs to be design for low frequency application then a choke (inductor) is used but getting discrete inductors at microwave frequencies is difficult so high-impedance quarter wavelength line (λ/4) at centre frequency is the best possible choice which designers can choose to design bias network. ***The thing that needs to be noted in bias circuit design is that more often than not this λ/4 is followed by a resistor or a bypass capacitor and these components adds extra length to the λ/4 line which designers sometime neglect and this could cause some desired RF frequency power to be dissipated in this branch which affects the gain and frequency response of the amplifier, so this***

calculated λ/4 line needs to adjusted by taking proper care of these extra elements and their footprints. One probable and commonly used method is to use Radial stub immediately after λ/4 high impedance bias line which will help to achieve proper isolation at desired RF frequency, no matter what component is added after λ/4 long bias line.

Fig. 5.30 shows the circuit design for bias circuit where it could be seen that high impedance λ/4 bias line is immediately followed by a Radial stub and then by a resistor and capacitor to ground. For more illustration layout of the bias network is shown in Fig. 5.31.

Fig. 5.4 shows the results for bias circuit and it can be seen that this design is acting as a near perfect bias network between 5.3 GHz – 5.5 GHz.

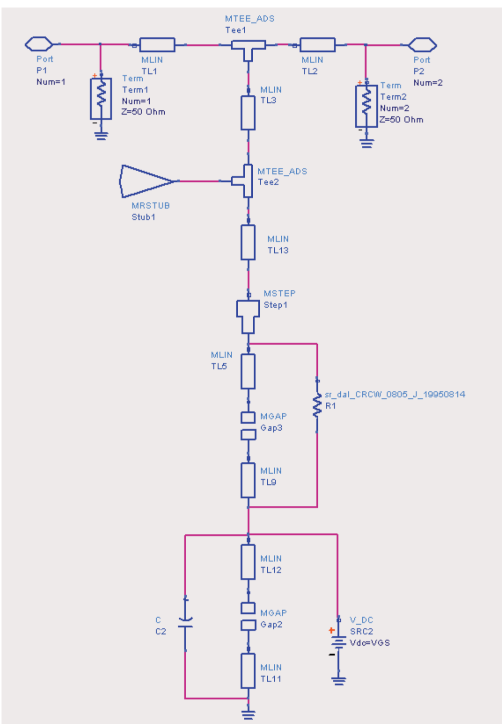

Fig. 5.30 Distributed Bias Network

Fig. 5.31 Layout of Bias Network

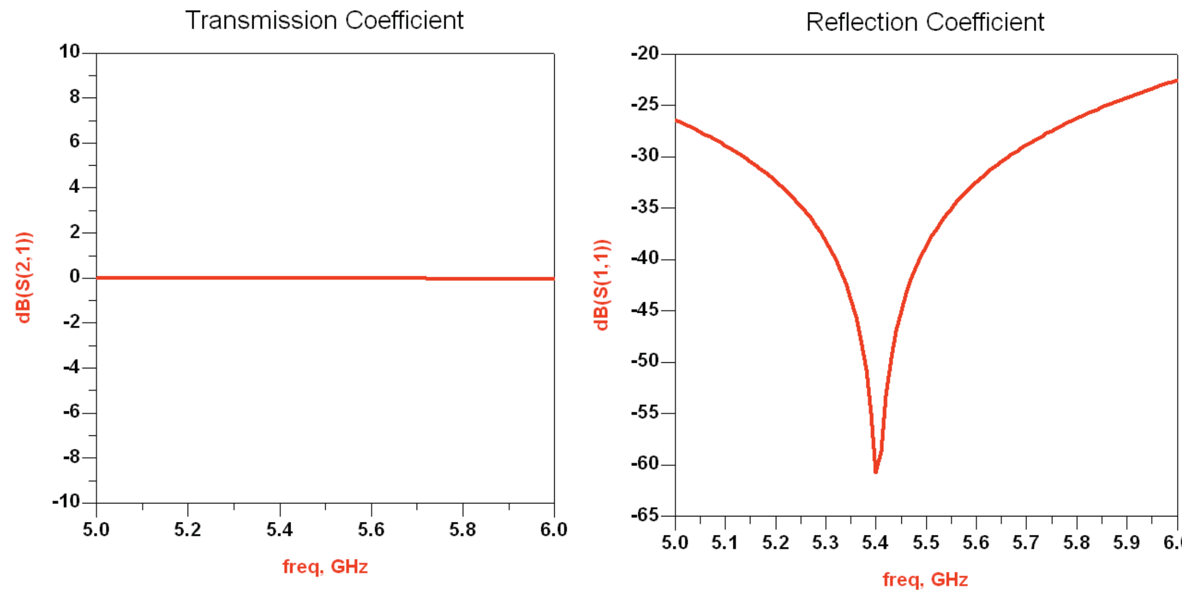

Fig. 5.4 Bias network response

3c). Stability Analysis:

Stability analysis is a very important aspect of any active circuit design and it is equally important in Amplifier design too. Fig 5.50 below shows the circuit that was obtained after adding input and output bias networks. ***Insert StabFact component from the Palette Simulation – S_Parameter to calculate the Stability Factor for the amplifier circuit as shown below.***

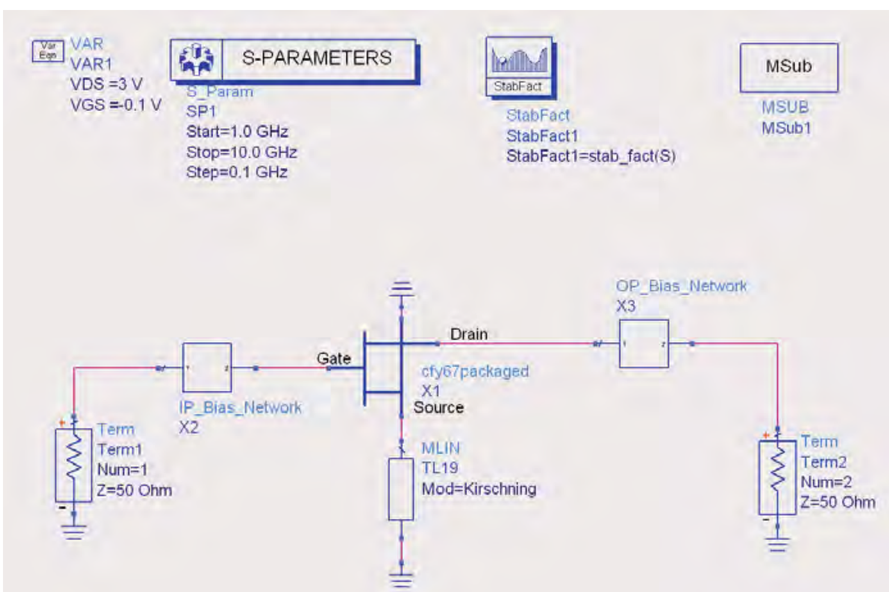

Fig. 5.50 Circuit with input and output bias networks added (shown as sub-circuits)

The results obtained from the circuit above is shown below in the Fig 5.51, which shows that circuit is unstable from ~ 2.1 GHz to 7 GHz and it needs to be stabilized before we can match input and output impedances.

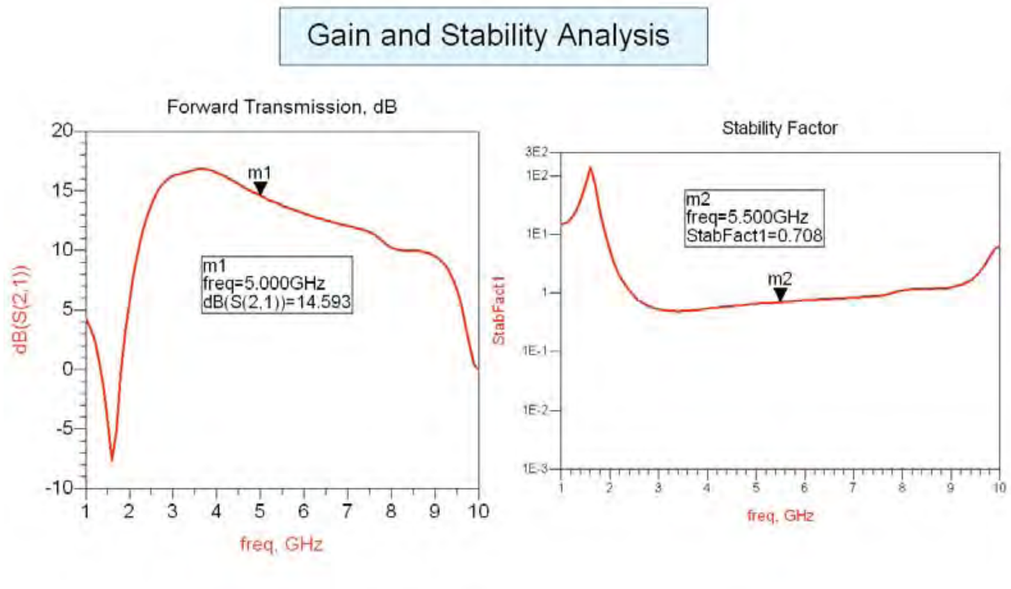

Fig. 5.51 Stability Analysis Results (showing circuit is unstable as K<1)

There are various stability configurations which could be used to stabilize the circuit, the most popular being using resistive loading of the circuit and choice is made depending upon the region of stability and type of amplifier being designed. Fig 5.52 shows one of the techniques to stabilize the circuit.

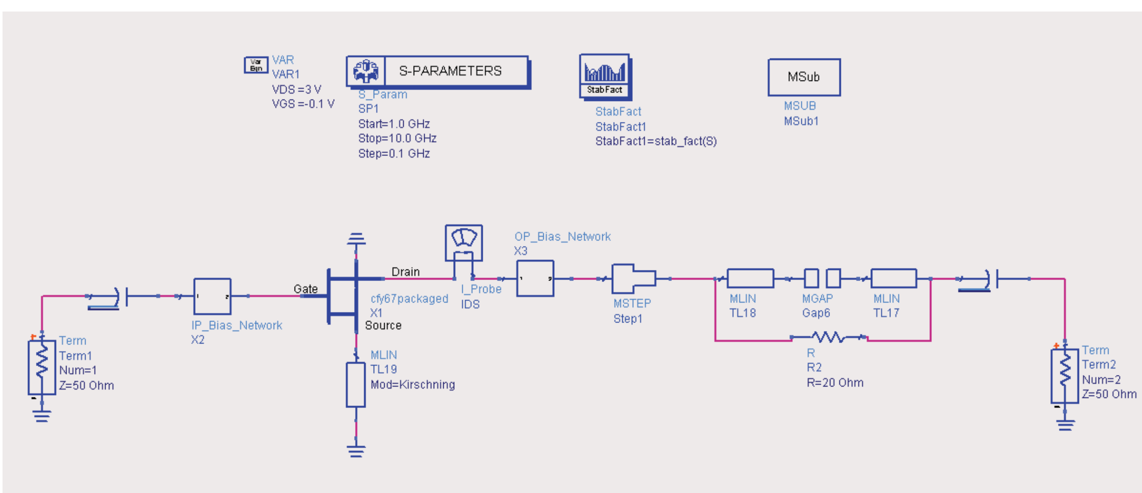

Fig. 5.52 Stabilized Circuit with Resistive loading at the output side (Please note the modeling of Resistor layout footprint which is connected in parallel to resistor, this will allow us to take care of mismatch or distortion introduced because of discrete component's footprint)

One output resistor was used at the output side of the amplifier and then the value of resistor was tuned to achieve the proper stability. Fig 5.53 below shows the results after stabilization.

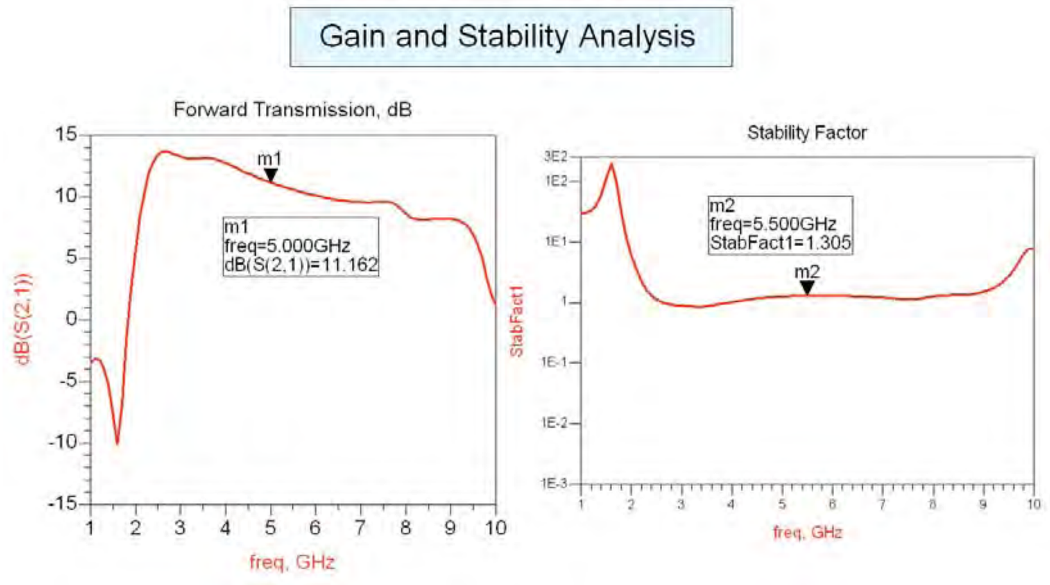

Fig. 5.53 Stabilized Circuit response

3d). Input and Output Matching Network Design:

After the circuit is stabilized in the broadband range, now we can start the design of the input and output matching networks so that we could achieve the desired specification of the amplifier. Fig 5.60 below shows the amplifier after adding bias networks, the stability components and the input and output coupling capacitors. Designers must note the proper layout footprint modeling for lumped components in schematic as shown in Fig. 5.60 below so as to take care of the discontinuities which signal will undergo in practical circuit and this should accompany each lumped components. This is quite important while designing amplifiers in the microwave range.

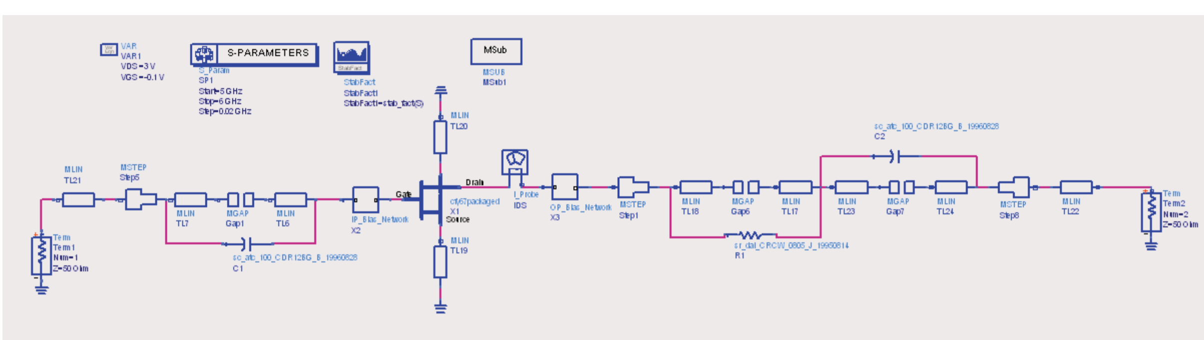

Fig. 5.60 Circuit used for designing input and output matching networks

The choice of the matching networks topology mainly depends on the bandwidth of the amplifier so that designer can choose between Single stub and double stub matching networks. Fig 5.61 below shows input and output impedances on the smith chart which needs to be matched with the 50-ohm impedance.

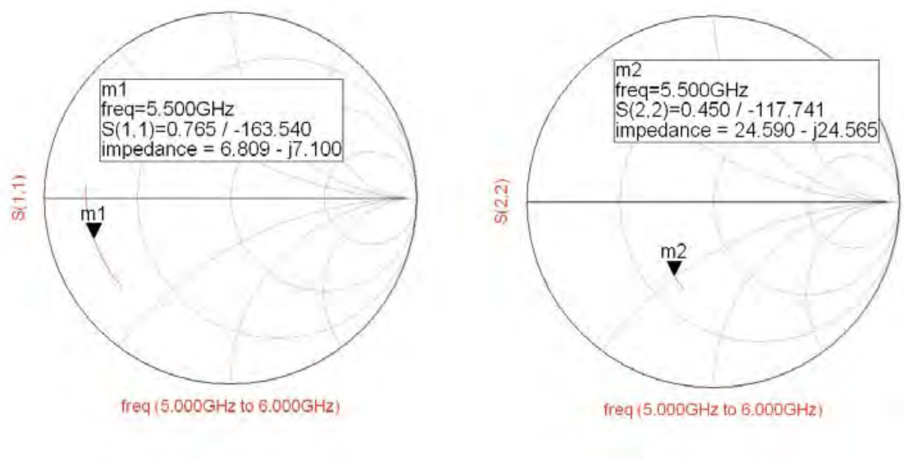

Fig. 5.61 Input and Output impedances on Smith Chart

For the present amplifier design a double stub approach was used to design the input and output matching networks to achieve the best possible input and output return losses. Fig 5.62 and Fig 5.64 below shows the input and output matching networks that were designed using Matching networks synthesis utility available in ADS software. Fig 5.63 and Fig 5.65 shows the results after connecting input and output matching networks to the amplifier circuit.

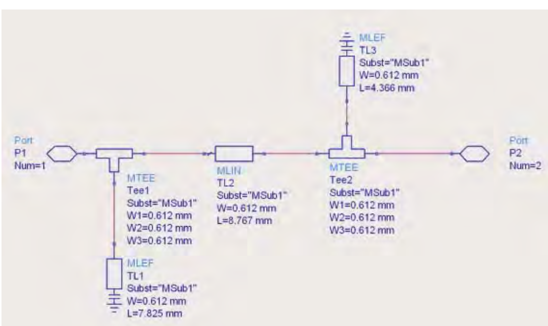

Fig. 5.62 Input Matching network

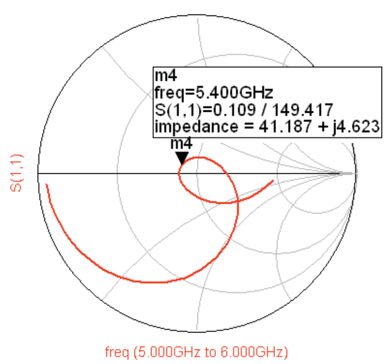

Fig. 5.63 Input Return Loss

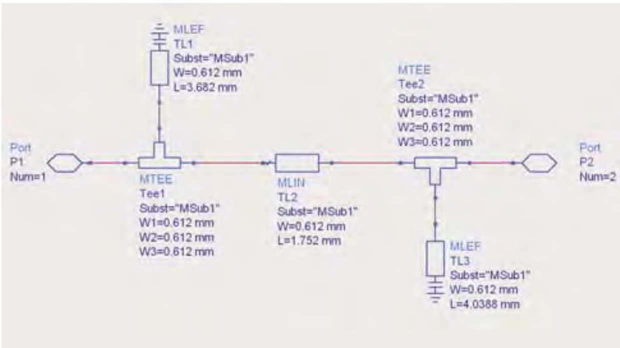

Fig. 5.64 Output Matching network

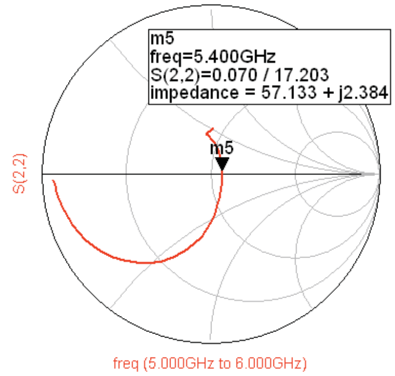

Fig. 5.65 Output Return Loss

3e). Overall Amplifier Performance Optimization:

The only thing remaining now in amplifier design is to connect all the sub-networks together and see the overall amplifier performance and to optimize the overall circuit if needed. Fig 5.70 shows the complete designed amplifier, Fig 5.73 shows the complete layout of the designed amplifier and Fig 5.74 shows the amplifier results and these were obtained after minimal manual tuning of the matching stub lengths to achieve the desired results after each of the blocks together. For clarity the input and output sections of the amplifier are shown in Fig 5.71 and Fig 5.72 respectively.

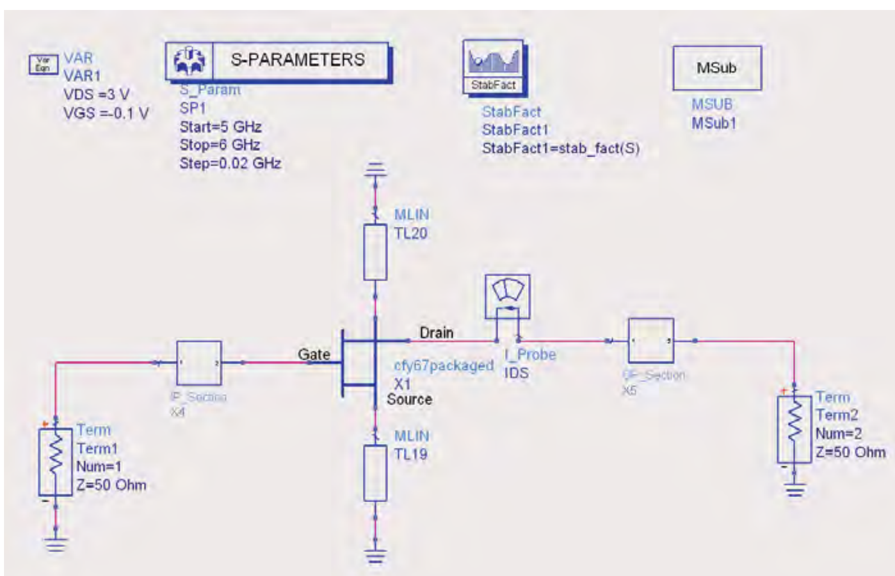

Fig.5.70 Complete Amplifier Schematic (Sub-circuits represents the Input and Output sections)

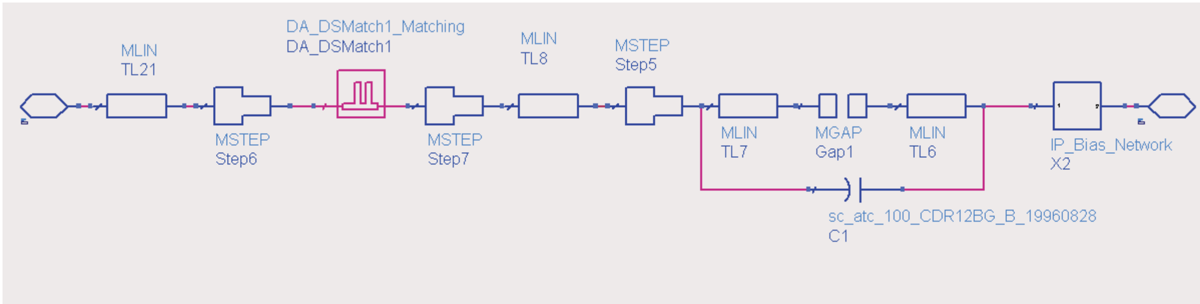

Fig.5.71 Input section of Amplifier

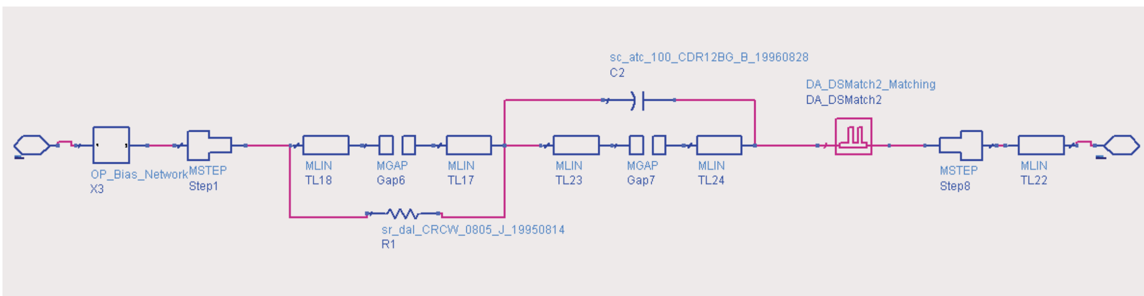

Fig. 5.72 Output Section of Amplifier

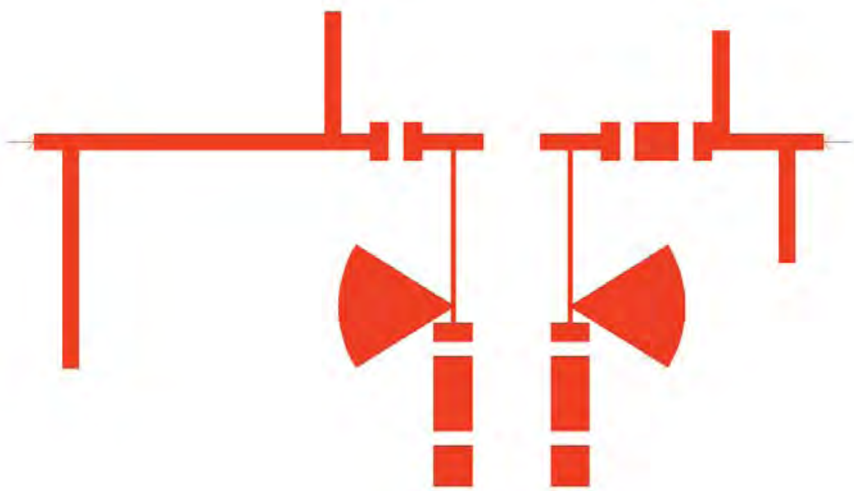

Fig. 5.73 Complete Amplifier Layout (Circuit size can be further reduced by folding the input match line)

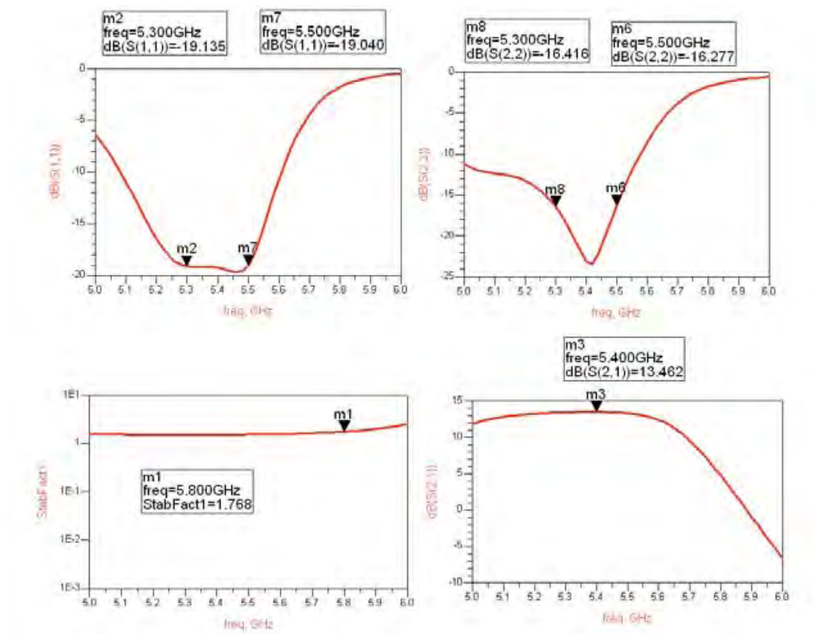

Fig. 5.74 Amplifier Results

4. Conclusion:

It could be seen from above that amplifier designs could be easily done if a well defined procedure is followed and designer could save lot of his time which is spent in fine tuning the amplifier performance optimization.