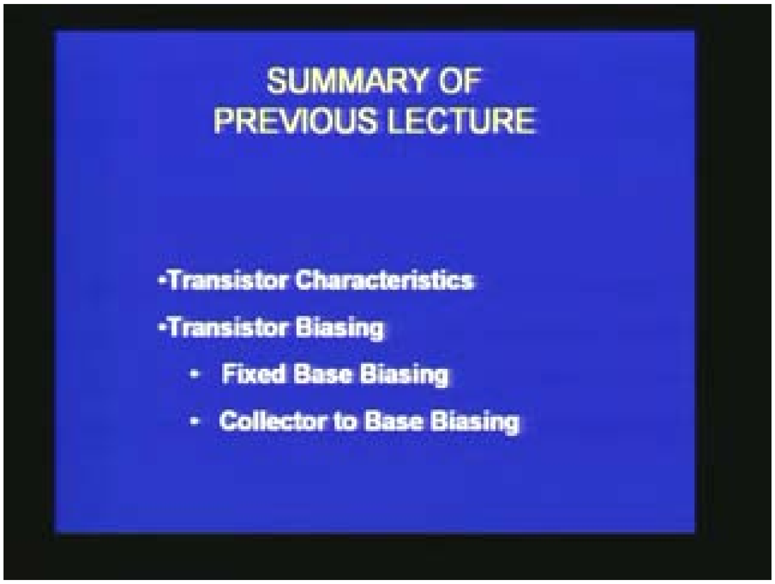
BASIC ELECTRONICS
PROF. T.S. NATARAJAN
DEPT OF PHYSICS
IIT MADRAS

LECTURE-12
TRANSISTOR BIASING
Emitter Current Bias Thermal Stability
(RC Coupled Amplifier)

Hello everybody! In our series of lectures on basic electronics learning by doing we will move on to the next lecture. Before we do that let us quickly recapitulate what we learnt in the previous lecture. In the previous lecture as you can see on the screen we discussed the basic characteristics of transistor especially the output characteristics of a common emitter configuration. We also saw about the basic principles of transistor biasing after looking at the concept of a load line for a given load in a common emitter amplifier.

[Refer Slide Time: 2:06]

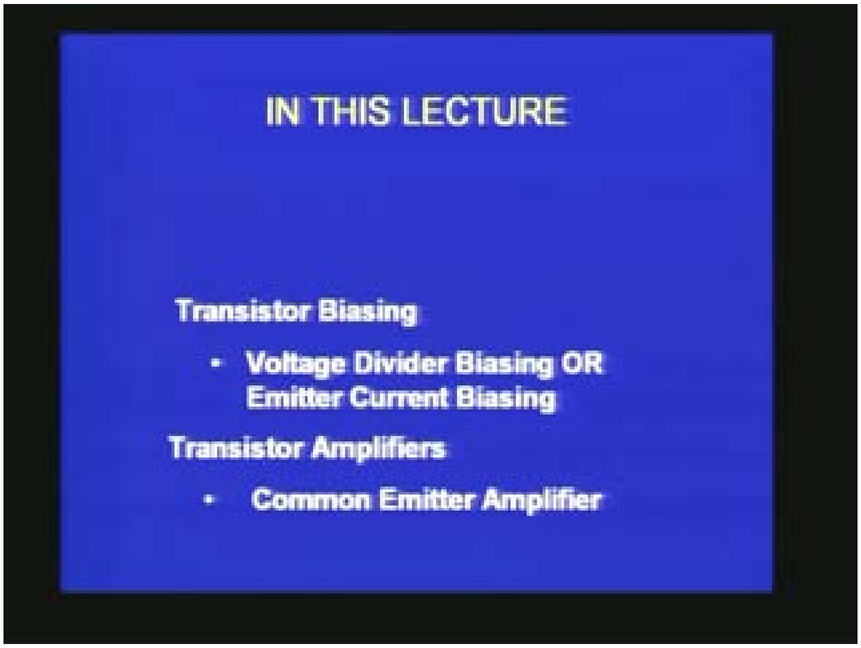


SUMMARY OF
PREVIOUS LECTURE

- Transistor Characteristics
- Transistor Biasing
  - Fixed Base Biasing
  - Collector to Base Biasing

Then we saw with some examples two types of biasing schemes one is called a fixed base biasing the other is the collector to base bias. In this lecture we will try to look at one other very important biasing scheme which is called the voltage divider biasing or emitter current biasing.

[Refer Slide Time: 2:37]



IN THIS LECTURE

Transistor Biasing

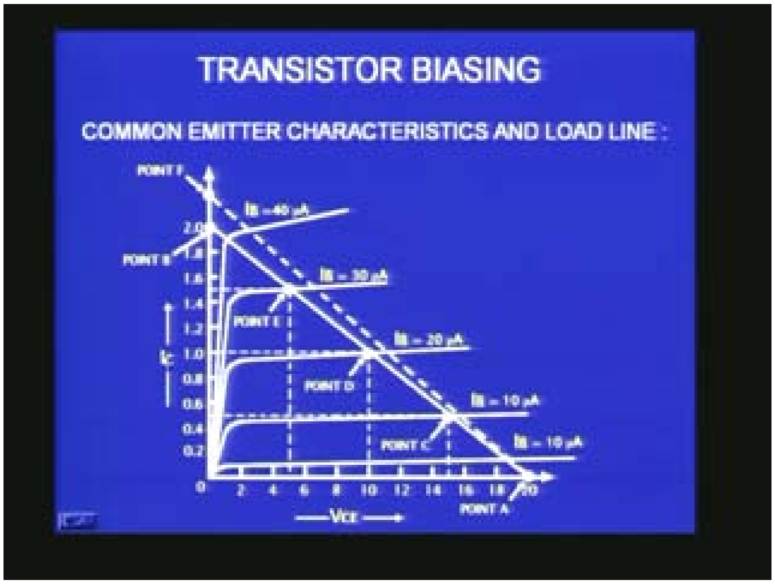
- Voltage Divider Biasing OR Emitter Current Biasing

Transistor Amplifiers

- Common Emitter Amplifier

We also perhaps will be able to see the basic principle of a simple RC coupled amplifier which makes use of common emitter configuration of a transistor. This is what we planned during this lecture and when I say biasing we mean that we want to put the transistor in action at a particular point which is called as the operating point or the q point on the output characteristics of a transistor especially in the common emitter mode. You can see on a graph a load line drawn on the output characteristics which is a characteristics between V_{CE} which is the voltage across the collector emitter junction we have collector emitter terminals and I_C is the collector current.

[Refer Slide Time: 3:38]



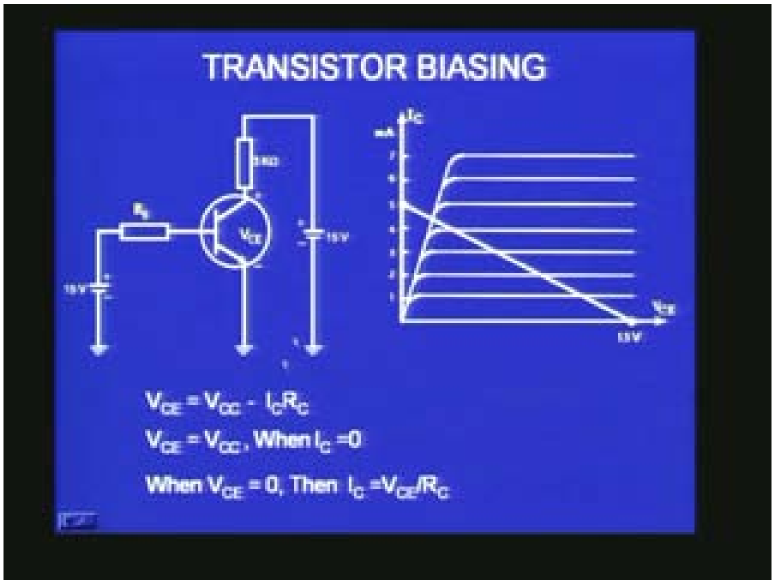
TRANSISTOR BIASING

COMMON EMITTER CHARACTERISTICS AND LOAD LINE :

This graph V_{CE} versus I_C is called the output characteristics and this is one of the load line that we discussed in the previous lecture. You can choose different points like point A or point C, D, E or B as your operating point and we said this region which is close to the Y-axis is called a saturation region and this region which is very close to the X-axis is called the cutoff region and if you want to use transistor as an amplifier then you must confine yourself to the active region which is the region which is generally in between these two. We can see that only the point C or D or E can be used or something in between that can be used as our operating point or q point.

If I say one point on let us say the point D then you see the moment I operate the transistor at the q point corresponding to point D on the graph then my collector current should be 1 milliampere as you see on the X-axis and I must have the voltage between the collector emitter to be around 10V with a power supply which will be operating at 20V. The V_{CC} supply is 20V and the 10V will be the voltage across the collector and the emitter and 1 milliampere current will be the current through the collector. How do we arrange so that my transistor is at this point is what we call as the biasing. We discussed two different types of biasing. We will move on to the next scheme of biasing. This is again one more scheme to recapitulate what we mean by the load line.

[Refer Slide Time: 5:41]



TRANSISTOR BIASING

$V_{CE} = V_{CC} - I_C R_C$

$V_{CE} = V_{CC}$, When $I_C = 0$

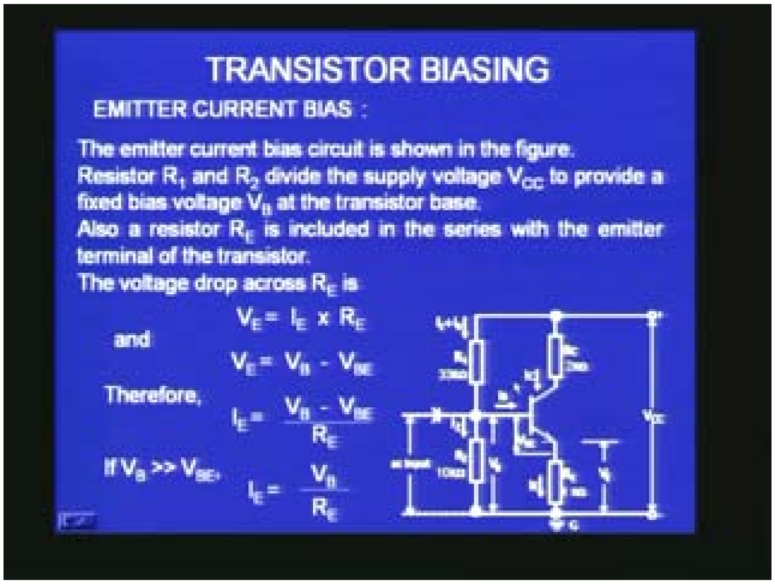
When $V_{CE} = 0$, Then $I_C = V_{CE}/R_C$

The V_{CE}, the voltage between the collector and the emitter is nothing but the total voltage or the V_{CC} minus the voltage drop across the R_C. V_{CE} is nothing but V_{CC}- I_C R_C. By Ohms law current into resistance gives me the voltage. I_C into R_C gives me the voltage drop due to the current I_C across the load resistor here R_C. V_{CE} is equal to V_{CC}-$I_C R_C$ is the equation to a straight line and if I draw a graph between V_C and I_C this will be a straight line with the intercept at V_{CC} and the maximum current I_C being V_{CE}/R_C. That is what is shown. V_{CE} is equal to V_{CC} when there is no collector current; I_C is equal to zero and when V_{CE} is equal to zero and I_C is the maximum value V_{CE}/R_C. This is V_{CE}/R_C in the graph and this is V_{CC}. If I know two points I can easily draw a line and that line becomes my DC load line.

We already saw this in the previous lecture. It is more to recapitulate and by biasing I have to put my transistor to operate somewhere along this line preferably at the middle of the line so that it may be able to move on either side equally well when I apply an external signal.

Let us move on to the actual emitter current bias. This is one of the very important biasing schemes and here what we do is in the circuit that I have shown there are two resistors R_1 and R_2 forming a potential divider dividing the voltage V_{CC} so that at the middle point of R_1 and R_2 I will have a definite voltage which I can use as the voltage to bias my transistor.

[Refer Slide Time: 7:40]



TRANSISTOR BIASING

EMITTER CURRENT BIAS :

The emitter current bias circuit is shown in the figure.
Resistor R_1 and R_2 divide the supply voltage V_{CC} to provide a fixed bias voltage V_B at the transistor base.
Also a resistor R_E is included in the series with the emitter terminal of the transistor.
The voltage drop across R_E is

$$V_E = I_E \times R_E$$

and

$$V_E = V_B - V_{BE}$$

Therefore,

$$I_E = \frac{V_B - V_{BE}}{R_E}$$

If $V_B >> V_{BE}$,

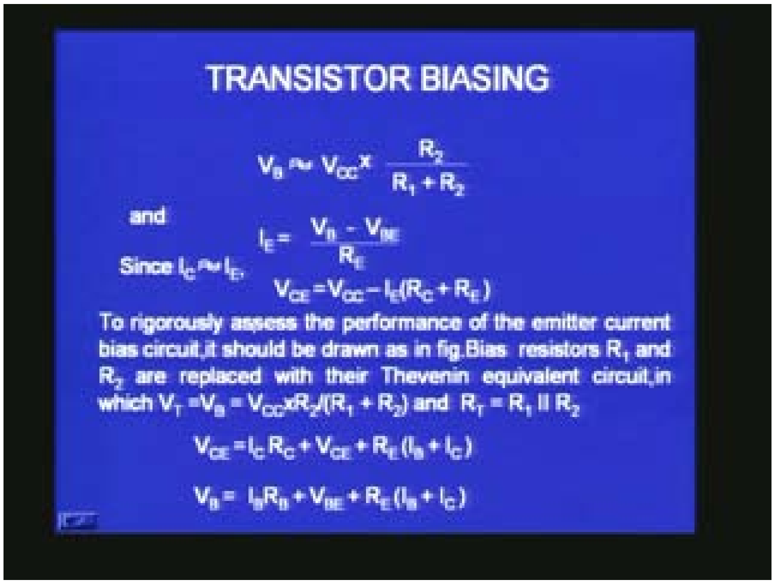
$$I_E = \frac{V_B}{R_E}$$

If you want to bias the transistor you should provide a specific voltage at the base with a specific base current and you should provide a specific voltage between the collector and the emitter and you should also make sure that there is a specific current passing through the collector. That is what we mean and instead of using a separate battery we are now trying to generate the voltage required to bias the base by using a simple scheme of potential divider which we have already seen under ohms law and the resistances R_1, R_2 makes a potential divider arrangement and the midpoint of R_1 and R_2 have some definite proportional voltage corresponding to V_{CC} and that is being applied as the battery for the base. The base current will be provided by this voltage and there will be a base current and this corresponds to a corresponding collector current and an emitter current.

Now what is going to happen? Let us find out what is the voltage V_E at the emitter junction and what is the voltage at the collector point and what is the voltage at the V_B? If I know all the V_E, V_B, V_C and I_B, I_C and I_E I have made everything ready for the biasing scheme. We know what is V_E? Most of the application here is the simple concept of ohms law. That is what we are going to apply every where here. What is V_E, the voltage at the emitter? This is the emitter and what is the voltage at this point? The voltage between the ground and this point is nothing but the I_E. If I know emitter current I_E and if I know the resistance value R_E if I multiply these two i.e., voltage current into resistance is voltage and that voltage should be equal to V_E. That is what is written in this equation here. V_E is equal to I_E multiplied by R_E. That we all know from ohms law. What is V_E? You can also look at it in terms of the other voltages. I know the voltages at the base as V_B and the voltage between the base and the emitter as V_{BE}. We all know for the silicon transistor V_{BE} is about 0.7V. So what is V_E? V_E is also the voltage at the base minus a small drop across the base emitter junction which is called V_{BE} and that is what V_E is. So V_E is V_B minus V_{BE} and now I can calculate I_E. I_E is nothing but V_E/R_E and instead of V_E I can write V_B minus V_{BE}/R_E. Usually V_{BE} is very small compared to V_B and we can ignore V_{BE} in some cases. That is what I have written here. If V_B is very, very large compared to V_{BE} for example for germanium transistor V_{BE} is about 0.2 or 0.3V. That is very, very small compared to V_B which can be 10V or 5V. In that sense it can be simplified as I_E is equal to V_B/R_E by ignoring the V_{BE}. But this is a more rigorous equation which we can use to calculate I_E.

If you know I_E it becomes very easy for you to know I_C because you know I_E is almost equal to I_C.

[Refer Slide Time: 11:22]



TRANSISTOR BIASING

$$V_B \approx V_{CC} \times \frac{R_2}{R_1 + R_2}$$

and

$$I_E = \frac{V_B - V_{BE}}{R_E}$$

Since $I_C \approx I_E$,

$$V_{CE} = V_{CC} - I_E(R_C + R_E)$$

To rigorously assess the performance of the emitter current bias circuit,it should be drawn as in fig.Bias resistors R_1 and R_2 are replaced with their Thevenin equivalent circuit,in which $V_T = V_B = V_{CC} x R_2/(R_1 + R_2)$ and $R_T = R_1 \parallel R_2$

$$V_{CE} = I_C R_C + V_{CE} + R_E(I_B + I_C)$$

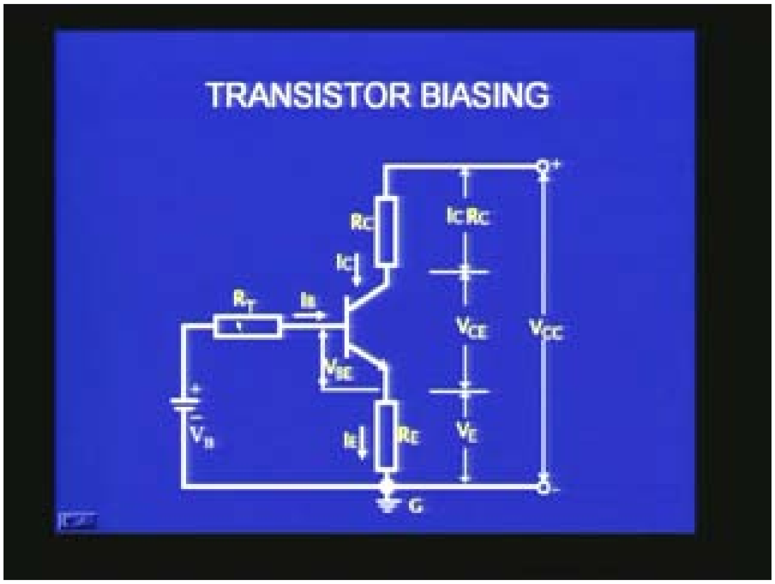
$$V_B = I_B R_B + V_{BE} + R_E(I_B + I_C)$$

The base current in a common emitter configuration is very, very small component. I_E will almost be equal to I_B. It is true for all transistors. What about V_B? If you look at the picture I said already R_1 and R_2 forms the potential divider and that only provides the V_B, voltage at the base. That is what is done here. V_B is approximately or nearly equal to V_{CC} multiplied by the potentiometer R_2 divided by R_1+R_2. This we have seen earlier in our discussion. V_{CC} by R_1+R_2 gives the current through the two resistors multiplied by the R_2 gives the voltage across the R_2 by simple ohms law. V_B is nearly equal to V_{CC} into R_2 divided by R_1+R_2 and this is the value of the V_B. Now what happens to V_E? V_E is V_B minus V_{BE} by R_E and we know V_B from the V_{CC} and the resistor combination that we have used for R_1 and R_2. So we can get I_E and since I_C is almost equal to I_E we also know I_C and what is the voltage across the collector emitter V_{CE}? That is nothing but V_{CC} minus I_E or I_C multiplied by R_C+R_E. I hope you see the point

Let me quickly go back to the picture. You can see in previous biasing scheme we did not have any R_E emitter resistance. But now we have another resistance in the emitter which is R_E. If I want the voltage across the collector and the emitter points of the transistor of the total V_{CC} some will be dropped across R_C, some will be between the collector and the emitter and the other will be across the resistor R_E. If I want this voltage V_{CE} then I should subtract from V_{CC} the voltage drop across R_C and the voltage drop across R_E. The voltage drop across R_C is I_CR_C. The voltage drop across R_E is I_ER_E. V_{CC} minus I_CR_C minus I_ER_E will give me the V_{CE} by Kirchoff's voltage law. But because I_C is almost equal to I_E we are not distinguishing between I_E and V_C. That is why here we have written V_{CE} is nothing but V_{CC} minus I_C or I_E into R_C+R_E. Because I assume the same current is almost going across both R_C and R_E I will find the total resistance R_C+R_E multiplied by I_E that is the total voltage drop across the two resistors. If I subtract it from V_{CC} what is left will be the voltage between the collector and the emitter. We have already said this is nearly equal because we have to make sure certain conditions are certified to make this exactly true.

You rigorously assess the performance of the emitter current bias circuit. It should be drawn as in the other next figure. I have replaced the R_1 R_2 the voltage divider bias into a equivalent Thevenin's power supply and Thevenin's resistor.

[Refer Slide Time: 15:01]

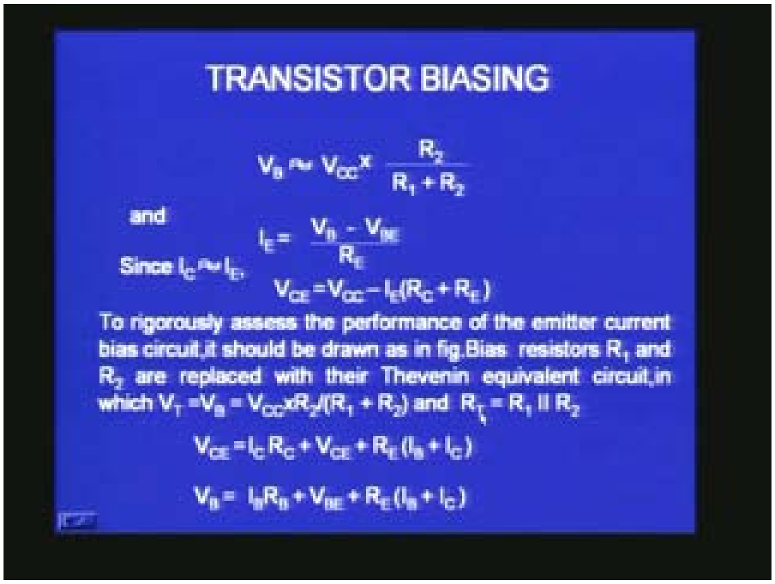


TRANSISTOR BIASING

We have seen also the Thevenin's theorem. We did some problems and then assignment with that and we have to look at what is V_B and what is V_{RT}?

What is V_T? V_T is nothing but the voltage that is V_B at the base and it is also equal to V_B and that is nothing but what we have already written V_{CC} into R_2 divided by R_1+R_2. So this is the Thevenin's equivalent resistance of the simple R_1 R_2 network and that here is the voltage at the base and V_B is V_{CC} into R_2 by R_1+R_2 two which we have already seen.

[Refer Slide Time: 15:41]



TRANSISTOR BIASING

$$V_B \approx V_{CC} \times \frac{R_2}{R_1 + R_2}$$

and

$$I_E = \frac{V_B - V_{BE}}{R_E}$$

Since $I_C \approx I_E$,

$$V_{CE} = V_{CC} - I_E(R_C + R_E)$$

To rigorously assess the performance of the emitter current bias circuit,it should be drawn as in fig.Bias resistors R_1 and R_2 are replaced with their Thevenin equivalent circuit,in which $V_T = V_B = V_{CC} x R_2/(R_1 + R_2)$ and $R_T = R_1 \parallel R_2$

$$V_{CE} = I_C R_C + V_{CE} + R_E(I_B + I_C)$$

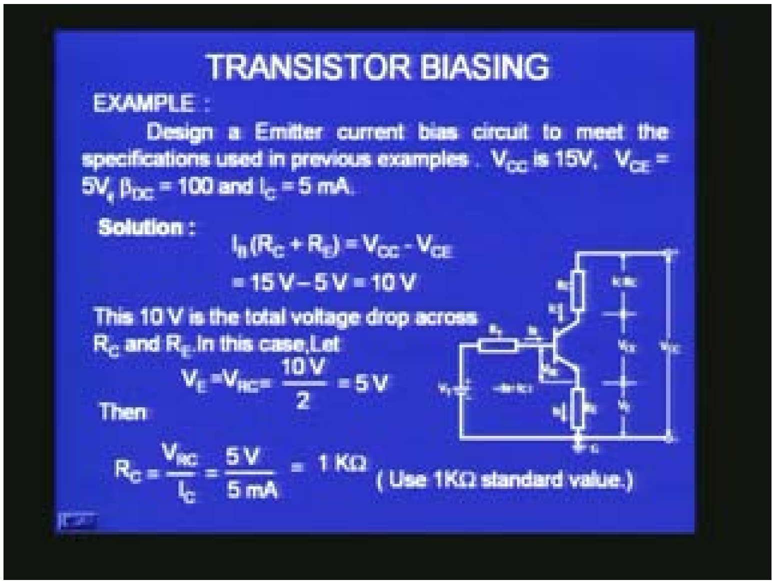
$$V_B = I_B R_B + V_{BE} + R_E(I_B + I_C)$$

What is the Thevenin's equivalent resistance? The Thevenin's equivalent resistance is R_1 parallel R_2. You should short all the power supply and if you short the power supply R_2 the other end of R_2 will also come to the ground. R_1 and R_2 will both be parallel and the effective resistance or the parallel resistance of the R_1 and R_2 will be the equivalent Thevenin's resistance. So R Thevenin's is R_1 parallel R_2. That is what is shown here. This is the R Thevenin which is nothing but R_1 parallel R_2 and this is nothing but V_{CC} multiplied by R_2 divided by R_1+R_2.

The two equations, basic equations of the base side and the emitter side can be written as V_{CE} is equal to I_CR_C plus V_{CE} plus R_E into I_B+I_C. We are ignoring the contribution from the I_B. This is a more rigorous equation and that is why we wrote a simpler equation here for the V_{CE}. Then what about V_B? Voltage at the base is nothing but I_B into R_B where R_B is actually the Thevenin's resistance R_T plus the V_{BE}. This is the application of the Kirchoff's law, V_{BE} which is the voltage between the base and the emitter and the voltage across the R_E which is actually the combination of I_B+I_C or this is equal to I_E. These things we have already seen.

It is in principle possible that we can look at any divider bias emitter current bias and then analyze the circuit completely because we know what all the different voltages and different currents how we can get. We must try to look at an example. That is what we do in the next screen. You want for example to design an emitter current bias circuit. To meet the specification what I have already mentioned V_{CC} for example is 15V, V_{CE} the voltage between the collector emitter is 5V.

[Refer Slide Time: 17:45]



TRANSISTOR BIASING

EXAMPLE :

Design a Emitter current bias circuit to meet the specifications used in previous examples . V_{CC} is 15V, V_{CE} = 5V, β_{DC} = 100 and I_C = 5 mA.

Solution :

$$I_B(R_C + R_E) = V_{CC} - V_{CE}$$
$$= 15\,V - 5\,V = 10\,V$$

This 10 V is the total voltage drop across R_C and R_E. In this case,Let

$$V_E = V_{RC} = \frac{10\,V}{2} = 5\,V$$

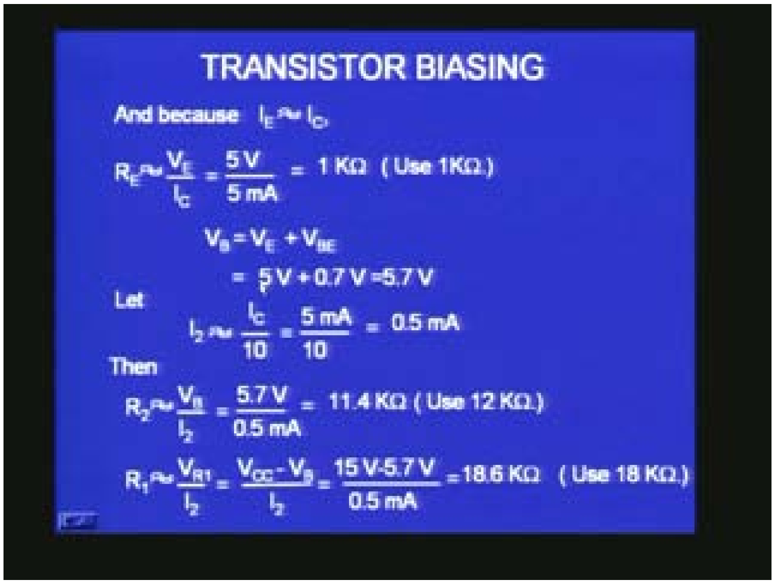
Then

$$R_C = \frac{V_{RC}}{I_C} = \frac{5\,V}{5\,mA} = 1\,K\Omega$$

(Use 1KΩ standard value.)

We want the voltage here to be 5V. The total voltage applied here is 15V and the beta dc the current gain of the common emitter amplifier is 100 and the collector current will be 5 milliampere. How do we get that? The V_{CE} should be 5V. 15-5V is 10V and that 10V is the voltage that will be dropped across these two resistors R_C and R_E. This 10V is the total voltage drop across the R_C and R_E. In this case we just take the R_C. The voltage across R_C should be half of the total voltage. That means 10/2=5V. We require 5V to be dropped here; five volts to be dropped here and five volts to be dropped here. That is what it appears. What is the R_C value then I should choose? R_C will be the total voltage across the R_C divided by the collector current. That is 5V by the 5 milliampere. It is 1K ohm. So we can use 1K ohm for R_C and then because I_E is almost equal to I_C, R_E is V_E by I_C and that is equal to 5V/5mA. That is also 1K ohm. In this bias circuit we will use 1K ohm for R_C, 1K ohm for I_E and what is V_B? V_B is V_E+V_{BE}. We wrote previously VE is VB-VBE. Now we are writing the other way. V_B is equal to V_E+V_{BE}. V_E is 5V; we know already, we have just calculated or assume. V_{BE} is 0.7V assuming that to be a silicon transistor and the total voltage V_{BE} should be 5.7. We want this to be 5.7 and that means I must make sure that the R_1 and R_2 values are chosen such that the value at the base will be 5.7V. Then we have designed the biasing circuit according to our initial starting specification.

[Refer Slide Time: 19:46]



TRANSISTOR BIASING

And because $I_E \sim I_C$,

$$R_E \sim \frac{V_E}{I_C} = \frac{5\text{ V}}{5\text{ mA}} = 1\text{ K}\Omega \quad (\text{Use } 1\text{K}\Omega)$$

$$V_B = V_E + V_{BE} = 5\text{ V} + 0.7\text{ V} = 5.7\text{ V}$$

Let

$$I_2 \sim \frac{I_C}{10} = \frac{5\text{ mA}}{10} = 0.5\text{ mA}$$

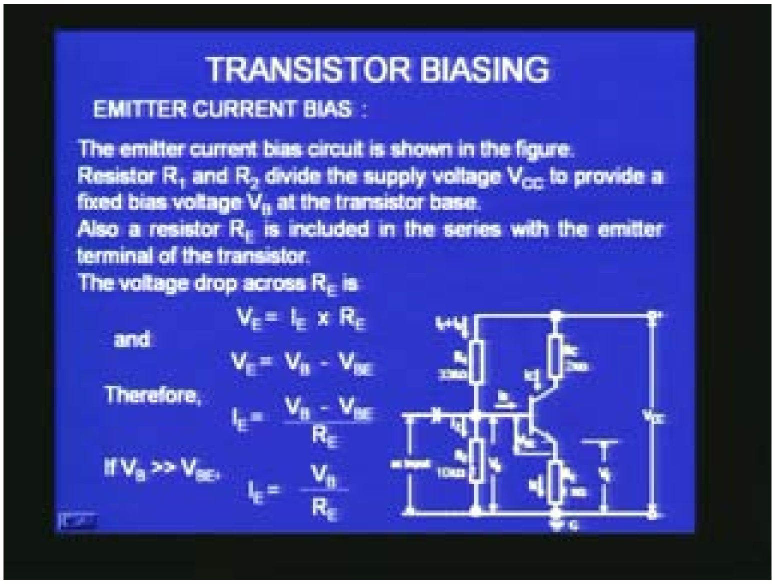
Then

$$R_2 \sim \frac{V_B}{I_2} = \frac{5.7\text{ V}}{0.5\text{ mA}} = 11.4\text{ K}\Omega \quad (\text{Use } 12\text{ K}\Omega)$$

$$R_1 \sim \frac{V_{R1}}{I_2} = \frac{V_{CC} - V_B}{I_2} = \frac{15\text{ V} - 5.7\text{ V}}{0.5\text{ mA}} = 18.6\text{ K}\Omega \quad (\text{Use } 18\text{ K}\Omega)$$

The current flowing through the resistor R_2 here is called I_2. This current should be some what about ten times less than the current I_C or I_E. Then only you make sure that I_B is very small. Only when I_B is very small I will be able to make use of the R_1 and R_2 itself for finding the potential divider.

[Refer Slide Time: 20:19]



TRANSISTOR BIASING

EMITTER CURRENT BIAS :

The emitter current bias circuit is shown in the figure.
Resistor R_1 and R_2 divide the supply voltage V_{CC} to provide a fixed bias voltage V_B at the transistor base.
Also a resistor R_E is included in the series with the emitter terminal of the transistor.
The voltage drop across R_E is

$$V_E = I_E \times R_E$$

and

$$V_E = V_B - V_{BE}$$

Therefore,

$$I_E = \frac{V_B - V_{BE}}{R_E}$$

If $V_B >> V_{BE}$,

$$I_E = \frac{V_B}{R_E}$$

Otherwise what will happen is this entire combination of the transistor base emitter junction and R_E will come in parallel to R_2 and if I connect resistors in parallel the effective resistance will be smaller than the smallest in the combination. We should make sure that this combination does not come into the significant contribution at this stage. For that we should ensure that the I_2 current that is flowing will have to be very, very large compared to the I_B. That means what? The R_1 and the R_2 resistances should be very,

very small in value. But you have another problem. The problem is any amplifier, this become later on as an amplifier, if it becomes an amplifier the input impedance or the input resistance that is the resistance as I look from the input side into the amplifier should be very large. Why so will be discussed at a later stage.

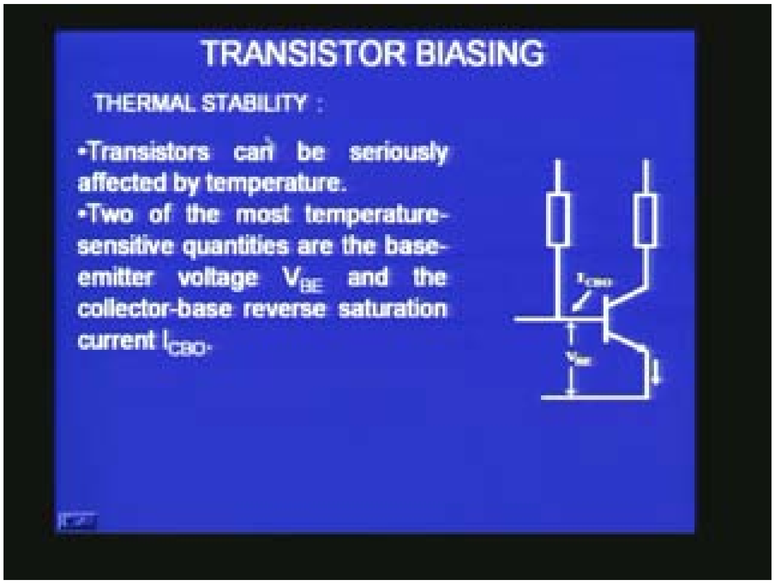
In principle now let us assume the input resistance in general for a good voltage amplifier should be very high and that will be ensured only if I choose R_1 R_2 values very large. Now here I have a problem. What is that? If I want R_1 R_2 to decide the base voltage without bringing in the reflected voltage coming from the R_E on the transistor emitter side then I must make R_1 and R_2 very small so that the current I_2 is very large compared to the base current. But if I want an amplifier with very high input impedance then R_1 and R_2 should be chosen very large. So R_1 R_2 for these two conditions will have to be both small as well as large which is not possible. We should try to get a compromise value. The compromise value in general is assumed corresponding to an I_2 value which is about one tenth of the I_C value. This is a thumb rule; something like a simple guide line for the design. I_2 approximately should be I_C divided by ten I_C. 5mA/10 gives me 0.5mA or about 500 microamperes. That should be the value of I_2.

Now I know I_2. I can calculate R_2 because I know V_B. V_B we have already found here is 5V+0.7= 5.7V. V_B is 5.7/0.5mA. That is the value of I_2 that we want. That comes out approximately to be about 11.4K ohms which we can correspond to a 12K ohm preferred resistor value available in the lab. Now we have obtained the value of R_2. R_2 is around 12K ohm. Then the point is how to get R_1. It's very simple because it is again application of ohms law. R_1 is nothing but the voltage across the R_1 divided by I_2. The voltage across R_1 is V_{CC} minus V_B the voltage at the base. This is 15V and V_B is 5.7V divided by I_2 which is again 0.5milliampere. That we have already calculated and if you plug in the values and calculate, it comes approximately to 18.6K ohm and we can use 18K ohm standard value of resistor in the lab. With this you can see that we have decided what will be the values of the various currents I_B, I_C, I_E and we have also found the different voltages, the different resistors R_1, R_2, R_C and R_E. Therefore we have completely designed the voltage divider bias with the emitter current biasing.

I already mentioned to you that I_B should be very, very small compared to I_2. This is a very important condition. Then only the transistor biasing will be stable. The stability of the biasing is a very important consideration. What do we mean by stability? Basically stability is with reference to thermal stability. Let us briefly look at the thermal stability. The transistors are made of semiconductors and semiconductors are very, very susceptible to temperature variations. Even though I make sure that the voltages and the currents are very specifically maintained by using different biasing schemes there is no guarantee that the biasing or the q point will be a constant point on my output characteristics or on my load line. Because if there is a temperature fluctuation then the q point will also start moving about the load line. This will certainly happen. So we must make sure all the factors that will come into the game in bringing about this instability due to temperature.

There are two things which are normally very significant.

[Refer Slide Time: 25:49]



TRANSISTOR BIASING

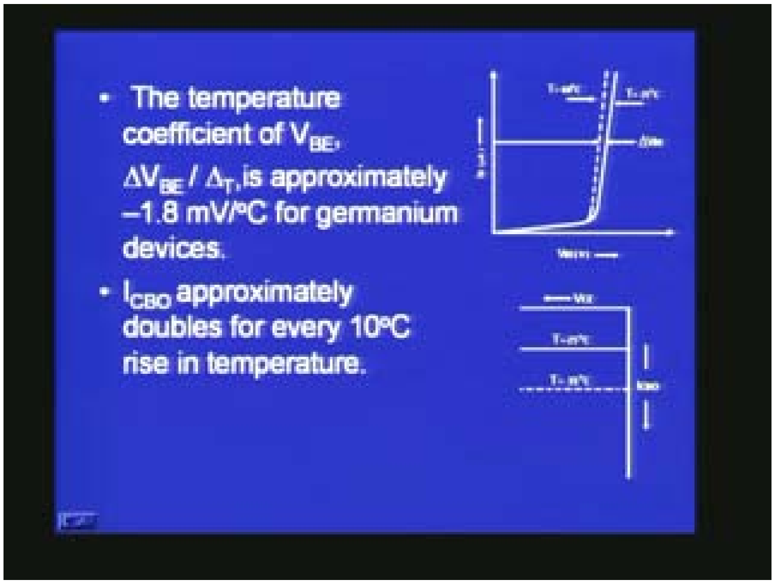
THERMAL STABILITY :

- Transistors can be seriously affected by temperature.
- Two of the most temperature-sensitive quantities are the base-emitter voltage V_{BE} and the collector-base reverse saturation current I_{CBO}.

One is V_{BE}, the voltage between the base emitter junction and the other one is the ICBO about which I have not told you. ICBO is the collector base reverse saturation current. In all transistor action we mentioned that the base emitter junction should be forward biased, the collector base junction should be reverse biased. When I reverse bias the collector base junction there is no forward current going through a reverse biased junction. But there will always be the minority carriers on both sides of the base and the collector which can cross over without much problem and they will contribute to a very small but a finite current which is called ICBO, the collector base reverse saturation. This is saturation current because at a given temperature this will be a constant. If I increase the temperature because of intrinsic conductivity more bonds will be broken in the semi conductor due to which more minority carriers as well as majority carriers will be generated. But the majority carriers are very large in number. The addition due to temperature is very insignificant in the majority carriers. But in the minority carriers this addition due to temperature can be a very significant component and this increase in ICBO can create problem for us with reference to the biasing scheme. Let us see how? I have shown on the picture here the ICBO which is the reverse saturation current flowing through the base and V_{BE} the voltage between the base and the emitter are the two important parameters which can produce problems when we do a biasing. The biasing point will have to be a fixed point with reference to temperature. Then only my amplifier will behave in the same way I want it to.

The temperature coefficient of V_{BE} which is actually represented by the small change in the V_{BE} to the small change in the temperature, delta T, delta V_{BE} by delta T is called the temperature coefficient which is approximately -1.8 millivolts per degree centigrade for a germanium device and is about 2 millivolts for silicon transistors.

[Refer Slide Time: 28:08]



- The temperature coefficient of V_{BE}, $\Delta V_{BE} / \Delta_T$, is approximately –1.8 mV/°C for germanium devices.
- I_{CBO} approximately doubles for every 10°C rise in temperature.

For germanium silicon devices the V_{BE} can increase or decrease by -2 millivolts for every degree centigrade; it can decrease by this amount. It is a very small value for normal considerations but this can become significant when the temperature difference is very large. For example in space flight when the satellite goes into the space the temperature variation can be very, very large there and the biasing will have to be very carefully stabilized otherwise the amplifiers will not perform well.

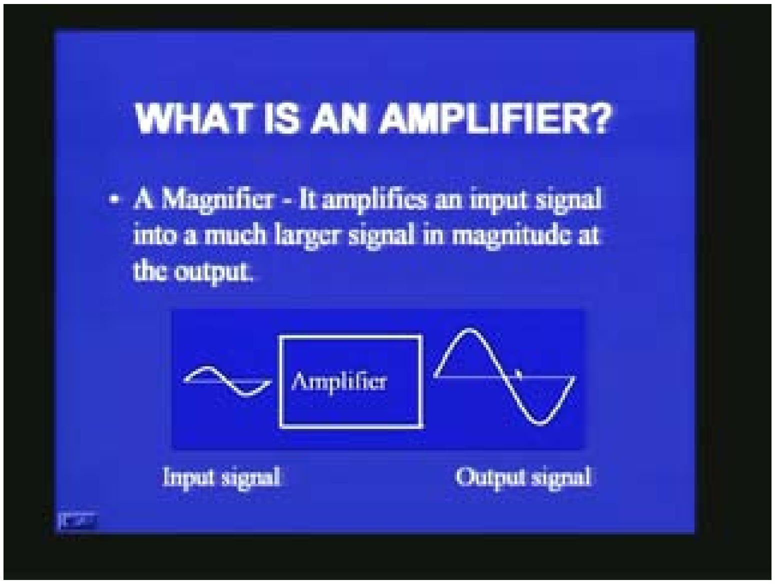
Here I have shown you two forward bias characteristics for two different temperatures; one for 25 degrees and another for 60 degrees. The V_{BE} has decreased here and this will correspond for every degree about 2 millivolts and that is what is shown here. As a mater of fact this variation in V_{BE} with reference to temperature for a constant current can be used as a temperature sensor or a transducer or a sensor for a thermometer. There are germanium and silicon thermometers designed based on this principle that for a given current flowing through the transistor you can have a continuous variation of about two millivolts for every degree and this if I can amplify and then read I will be able to evaluate the temperature. This can be used as a temperature sensor which is also one of the applications that we can think of.

Here we are worried about that variation. We try to make sure that the variation is not spoiling the performance of any amplifier. The ICBO again approximately doubles every 10 degrees. It is this value; after 10 degrees you can see it is almost increasing to the same extent. At 25 degrees it is this much; at 35 degrees after 10 degrees it has doubled. If I go to 45 it will come over here and the ICBO keeps on doubling every 10 degrees. This is much more serious of the two; delta V_{BE} and delta ICBO. The delta ICBO is much more serious. Why is it so? When the delta ICBO increases the I_C component will increase whatever you said. The effect of change in ICBO due to temperature is to increase the collector current or the emitter current. When the collector current or the emitter current increases, it will also increase the temperature. When the current increases the I square R which is the heat dissipation which is generated that will also increase. When the junction temperature increases more ICBO will be generated. More ICBO means larger I_C and larger I_C means larger temperature, each of them helping the other and it will slowly be a cumulative effect. The current of the collector because of the temperature variation will start increasing without any control on its own and after sometime the current can become very large and if you are not able to check this it may ultimately lead to the transistor being spoiled due to the large current flowing through that. If you exceed the power rating capability of the transistor, the transistor may be spoiled; so one has to be very careful. This is what is called thermal run away. The transistor runs away with larger and larger current initiated by the temperature variation and the temperature variation increases I_C. I_C increases the temperature. Each one of them help each other and they both run away and finally we end up with the transistor getting spoiled. We must make sure that the thermal run away condition is not generated in any amplifier. That is one of the very important things. That is why people say ICBO is much more important parameter to worry about than the V_{BE}.

The thermal stability of the circuit is normally assessed by deriving a stability factor, S. I am not going into the details of the stability factor but the stability factor is nothing but delta V_{BE} by delta T or delta ICBO by delta T. One can actually calculate the change or the stability factor for different biasing schemes and we can try to monitor and control that the transistor does not enter into a thermal run away condition.

I am not going into the details of thermal stability as I already mentioned to you. But let me move on with our focussed attention to have an amplifier built with the transistor. After biasing what is that we have to do? We want to build an amplifier using the transistor. Before we build an amplifier it is not out of place to just understand what we mean by an amplifier? What is an amplifier? It is just a magnifier; something which will amplify, magnify an input signal into a much larger signal in magnitude. You know a lens can be used to magnify small objects. In the same way small electrical signals can be magnified using amplifiers at the output. I have shown a block diagram here where the block is actually an amplifier. If I have a small sinusoidal signal at the input, the output becomes a very large signal. I hope you can see on the screen. When I have a small input signal which is in the form of the sinusoidal signal applied to the amplifier at the output you get much larger signal which is an amplified signal. We must also make sure that it is truthful; truthfully reproduced at the output. That means there is no distortion; this wave form does not involve any distortion during the process of amplification.

[Refer Slide Time: 34:46]



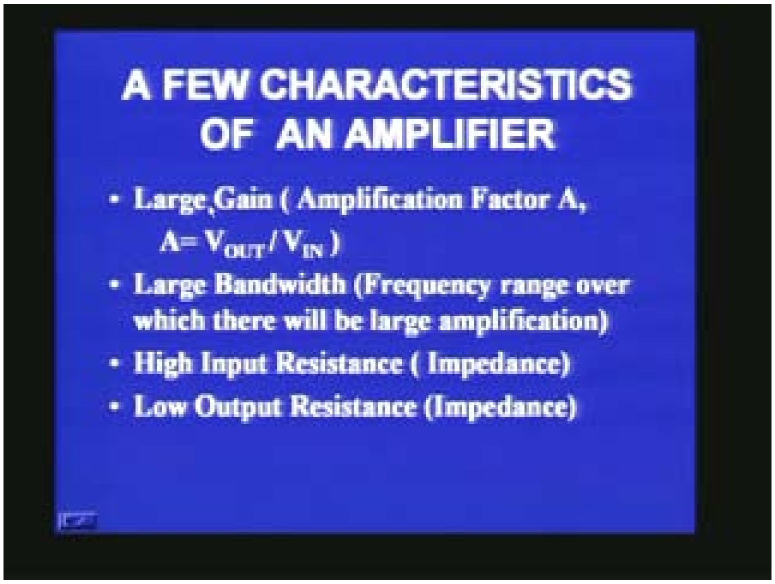
WHAT IS AN AMPLIFIER?

- A Magnifier - It amplifies an input signal into a much larger signal in magnitude at the output.

This is what we mean by an amplifier. High fidelity amplifier means it is having excellent fidelity; highly truthful with reference to the input wave form.

There are certain characteristics of the amplifier which also we should understand. Some of the characteristic for example is gain. An amplifier should have reasonably good gain large gain.

[Refer Slide Time: 35:06]



A FEW CHARACTERISTICS OF AN AMPLIFIER

- Large Gain (Amplification Factor A, A= V_{OUT}/V_{IN})
- Large Bandwidth (Frequency range over which there will be large amplification)
- High Input Resistance (Impedance)
- Low Output Resistance (Impedance)

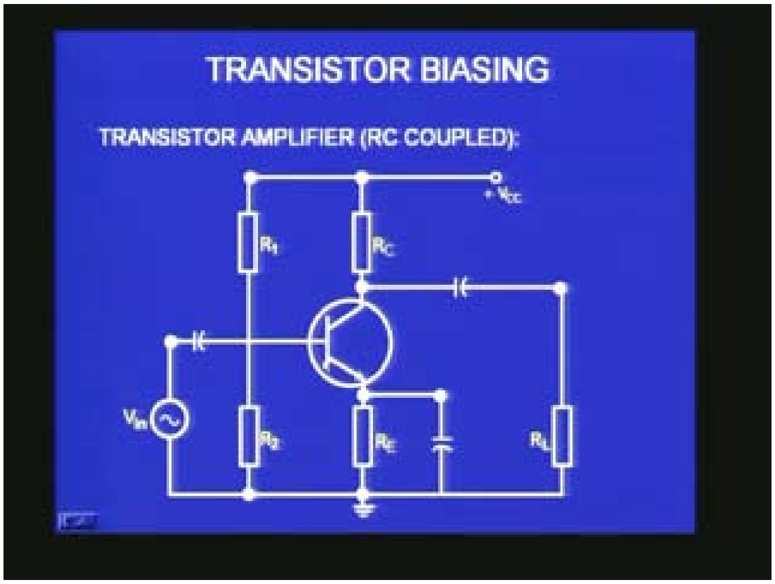
What is gain? Gain is the amplification factor which is normally represented by letter A. A is nothing but if it is a voltage amplifier for example A can be V_{OUT} divided by V_{IN}. What is the voltage output that I will get divided by voltage in. Usually this will be in rms for an amplifier. Only for a voltage amplifier the gain is V_{OUT} by V_{IN} because there are other types of amplifiers for example current amplifier, power amplifier, etc. In the case of a current amplifier what will be the gain? The gain factor will be I output divided by I input. That is the current amplifier or in the case of the power amplifier A will be equal to power output divided by power in. Depending upon the configuration and depending upon the type of transistor or amplifier the amplification factor can be voltage or current or power and it does not have a unit because it is a ratio of either voltages or currents. It's a mere number. Usually it should be very large if you want good amplification.

The other thing that we worry about in an amplifier briefly is large bandwidth. What do you mean by bandwidth? An amplifier will amplify audio signals. For example if it is an audio amplifier that we are all familiar with in a public address system if I speak at the microphone I must get through a loud speaker very loud sound; magnified, amplified signal. But then my speech will not contain one single sine wave. Because it is composed of several sine waves the signal will be a complex signal which is a superposition of several sine waves and my amplifier should be able to faithfully amplify all the different frequencies that are contained in the input signal. For example if it is music or an orchestra you have different types of instruments generating music or signals at different frequencies. All of them should be faithfully magnified and given at the output through the loud speaker. Then only you would get good music coming out of the amplifier. An amplifier cannot in principle amplify all frequencies equally well. There will be few frequencies it will be very well amplifying. There are certain other higher frequencies or the lower frequencies the amplifier will not able to amplify. Bandwidth is one characteristic which helps us to judge how much an amplifier can amplify with reference to the frequency range.

Usually our audio signals for example are 20 kilo hertz and up to 20,000 hertz the amplifier gain should almost remain constant. Then you find the bandwidth is very good for audio amplification. But there are situations where you want to have higher amplification factors or higher frequencies also. There are wide band and several other variations about which we will perhaps discuss little more later on. Apart from these two the amplifier should also have high input resistance. It is more pertinent to call it as impedance because it can be complex it will have both real and imaginary parts and for simplicity I am saying it should have high input resistance and it should also have low output resistance. High input resistance should be there so that it does not load the input and low output resistance should be there so that it can drive any load at the output. About these characteristics we will discuss in detail at a later stage and just I wanted to mention to you that whenever I want to construct an amplifier I should make sure that certain basic characteristics are built properly into the amplifier so that it can be made use of very well.

Let us move on to the transistor biasing circuit which I want to now convert into an amplifier for amplifying the AC signal, alternating signal. The circuit that you see on the screen is one such amplifier which is called RC coupled amplifier, common emitter amplifier.

[Refer Slide Time: 39:46]



TRANSISTOR BIASING

TRANSISTOR AMPLIFIER (RC COUPLED):

You can immediately recognize several of the resistors and the transistors in the configuration because we have just now seen that this is nothing but a voltage divider bias emitter biasing scheme. You have R_1 R_2 here. You have R_C and you have the R_E and you have the transistor about which we just now discussed on biasing scheme. If I want this to become an amplifier I have a signal here, input signal which can be from a microphone or from any standard signal generator. I have to couple it to this amplifier. This is at the q point. It is now positioned at the q point along the load line.

I want to give a signal here and that signal I want to couple to the base. For that I use a capacitor. Why do I use a capacitor? Because I want to only couple the alternating component and by chance if this input signal contains some DC, I do not want the DC to come here. If the DC comes the base current which I maintained constant previously by biasing will get altered. That means my Q point will shift. I do not want that to happen and I use the capacitor here at the input to isolate the DC or the biasing will not be hampered by this capacitor. This is the use. This is called a coupling capacitor because it is coupling the input signal to the base of the transistor and similarly I use another coupling capacitor at the collector for the output which I can couple to any external load R_L that I have connected here. This is the coupling capacitor at the collector terminal and this is the coupling capacitor at the input terminal at the base and these two are the additional things that you see here. These two are meant to couple a signal at the input and couple the output signal to the load.

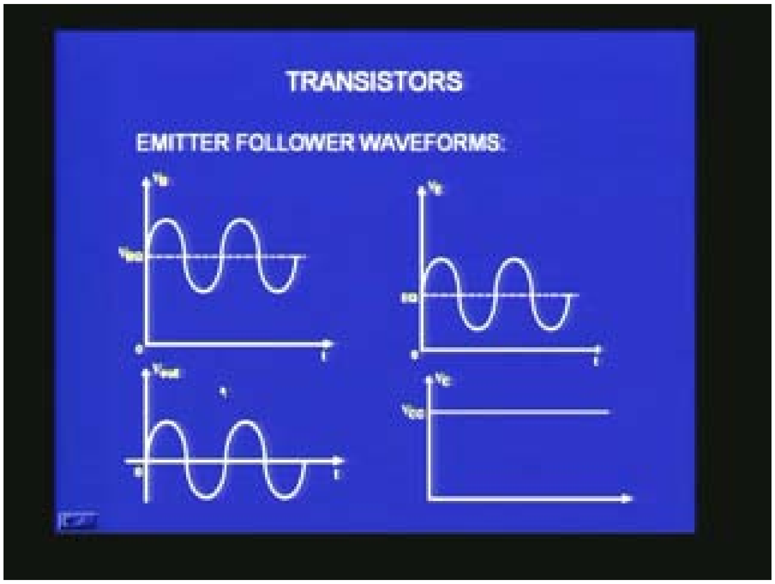
Apart from that you also see there is another capacitor here. This capacitor is called bypass capacitor. This bypass capacitor also is meant to maintain the q point constant because this V_E the voltage at the E, at the emitter point should be constant if I want the biasing to be constant. But when I amplify the signal the collector current or the emitter current can vary and there will be alternating voltage according to the signal here. That means the bias point will also start shifting along the load line. That should not happen. If I use a bypass capacitor all the alternating current component will flow through the lower resistance path provided by the capacitance. Capacitance offers less resistance to alternating current, more resistance to DC. We all know that one by omega C where omega is the 2 pi f, the frequency component and higher the frequency the lower is the resistance offered by the capacitance and all the AC signal will find a easy resistance path compared to R_E and they will go back and that means this voltage V_E will not be modified by the signal that I am amplifying. This is the bypass capacitor. These two are called coupling capacitors and the choice of these capacitors will have to be carefully made.

We must know the lowest frequency that we want to amplify and at that lowest frequency we should offer very low resistance. If you go to higher frequencies automatically the resistance offered by the coupling capacitors will be less because it is one by omega C as I already mentioned to you and the proper choice of capacitors is essential to make the bandwidth very good; the frequency range for which this will amplify very good. With this now you can see if I put a capacitor, apply an input signal I should get an output signal. What is actually going to happen is if I apply a sinusoidal signal at the base that will be coupled here; you can see that repeatedly comes over here and that means the base voltage is going to oscillate according to the signal and because of this the base current is also going to oscillate according to this.

When the base current changes the collector current also will change and because of that when there is larger base current there will be larger collector current. A larger collector current will produce larger voltage drop here and when the voltage here at the base increases the voltage here at the collector decreases because there is an inverse proportion here. This increase in current increases the current and this voltage is a constant voltage minus a voltage across R_C and because this is increasing the voltage here will have to decrease and this voltage at the collector will be inverted or 180 degrees out of phase with reference to the input signal that I give here. This is a very important idea that we should remember. There is a phase inversion. This will be naturally amplified because the variation here is in microamperes, the variation here in milliamperes. That means there is a gain of nearly 100 or so and that will come out here and when it is applied to a large resistor you can get a very large voltage here. That means this acts as a voltage amplifier.

This is a very simple scheme. I have drawn all the graphs here. This is voltage V_B. At the q point you can see it is shifting slowly in a sinusoidal fashion. Similarly the V_{OUT} also is sinusoidal.

[Refer Slide Time: 45:31]



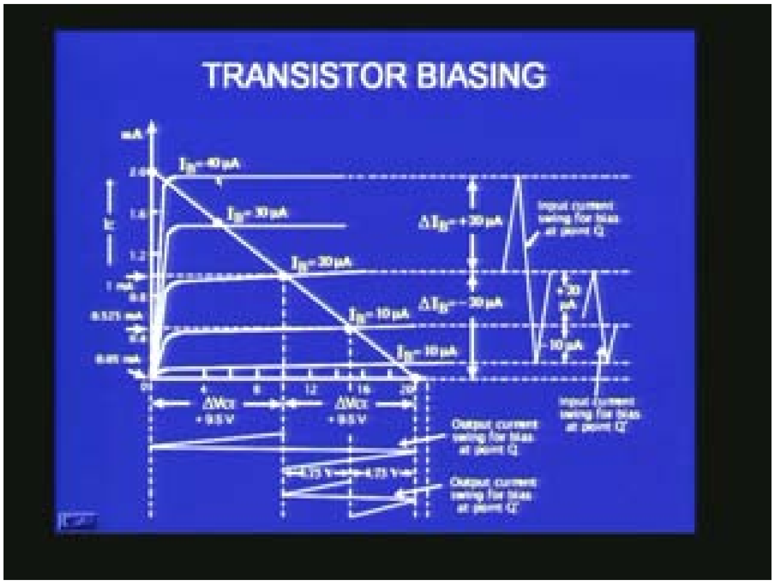
TRANSISTORS

EMITTER FOLLOWER WAVEFORMS:

The difference is even though they look alike this is very small. This should be very large and similarly here the V_E also will show this. But the biasing will not recognize this because it will be bypassed through the capacitor. But the V_{CC} the voltage will be always constant because we are not changing that. That is from a regulated power supply.

I have shown here with the load line. This is the biasing point and for example if I have a signal like this, that means what? The signal is having corresponding to I_B equal to 20 microamperes and it goes up to let us say I_B equal to 40 microamperes.

[Refer Slide Time: 46:13]

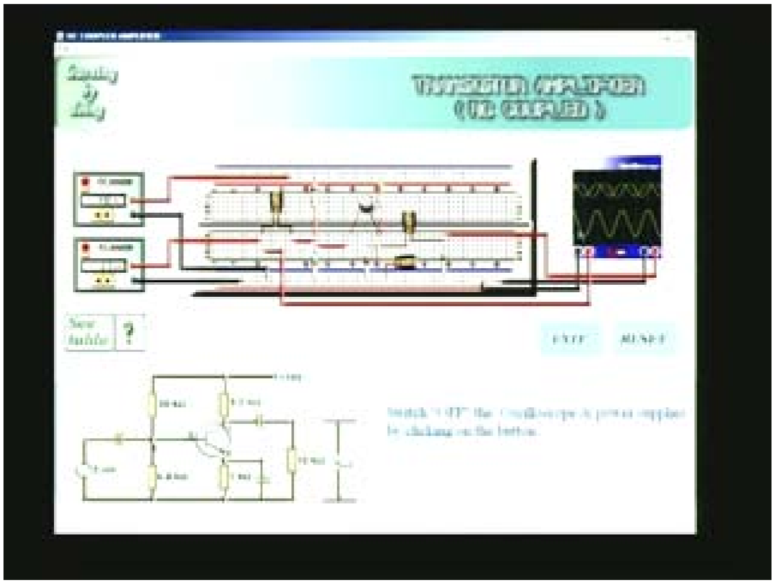


TRANSISTOR BIASING

That means it is changing I_B by nearly 20 microamperes. Similarly on the down side it can go up to this point. That means there is a 20. This input sine wave is producing a 20 microampere plus or minus variation in the base current and that will correspond to a corresponding change in the collector current here. You can see that and that will correspond to a corresponding collector emitter variation here. That is why you get a very large signal here. It is a large signal which is corresponding to the output and if I shift the operating point to some point here I will not be able to have perfect reproduction here in the sine wave. If I give large signal which is varying by 20 microamperes, here only 10 micro amperes is available. Beyond that if I go, there will not be any signal; the signal will get distorted. That is the reason why I say I must always bias on the load line nearly at the middle point mid point of the load line. Then I will have equal excursion possible on both sides of the load line and I will get reasonably good amplifier.

I have taken a simple example where I have put 33K ohms, 6.8K ohms and 3.3K ohm as the load, 1K ohms as the R_E and with this we will just try to see whether there is amplification. We will quickly go through a stimulation experiment. There is a breadboard here and you have the transistor DC 107 and you have R_1 33K and R_2 6.8K and this is the R_C which is 3.3K and you have R_E 1K ohm and you have the coupling capacitor here and you have another coupling capacitor here at the collector and this is the load resistor which is 10K ohm here and this is a bypass capacitor that we see. So it is exactly same as the circuit that is drawn here which is also the circuit which you saw previously. Let me quickly switch on the oscilloscope. I have a 10V V_{CC} and I give about 2 millivolts AC source at about 1KHz frequency and the input signal is very small, the output signal is very large, amplified signal you get.

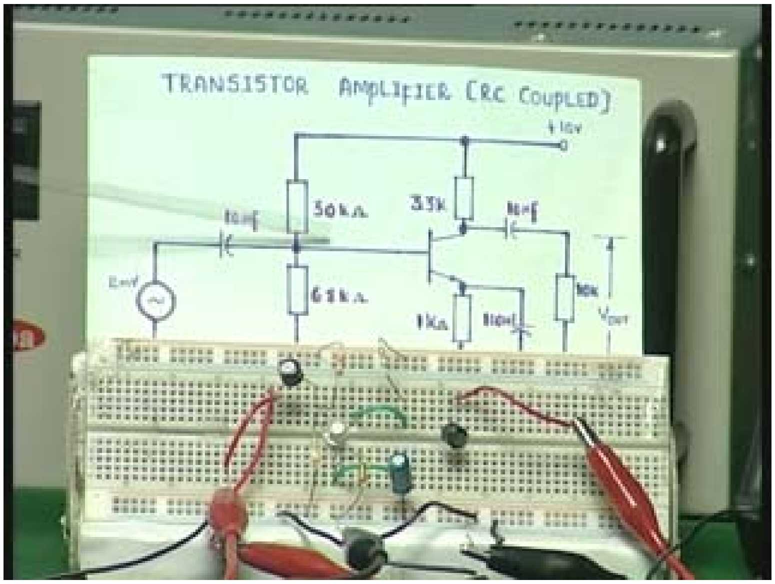
[Refer Slide Time: 48:37]



It is very easy to construct a simple RC coupled amplifier making use of a simple transistor and the voltage divider bias. In this case I have given an example using the voltage divider bias. I will quickly show you an actual demonstration with the bread board; a similar circuit constructed on the bread board and we will try to see whether we are able to see an amplification of the required number.

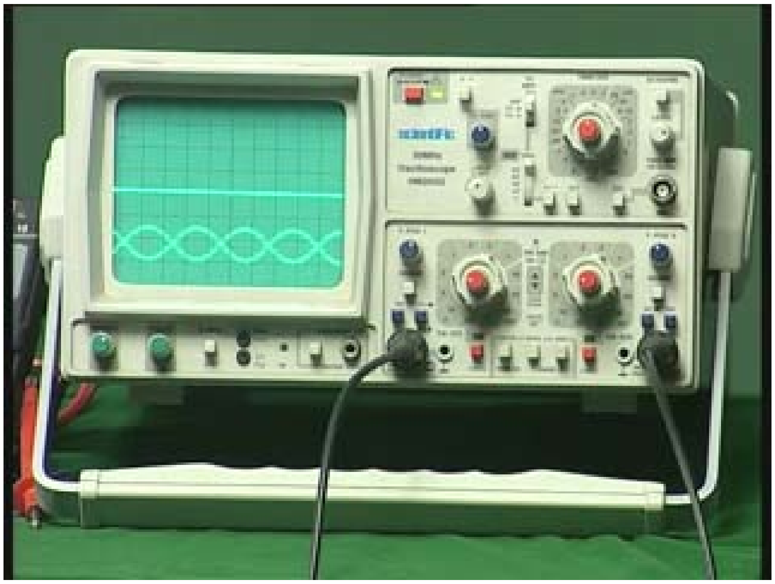
I have on the bread board here the same circuit which I just now showed you. For example we have a 3.3K RC, a transistor which is a DC 107. This is a 1K resistor; this is a 30K ohm resistor, R_1; R_2 is 6.3K. This forms the voltage divider bias that we already discussed. RC, RE transistor; R_1 R_2 this forms the voltage divider bias.

[Refer Slide Time: 49:48]



I have put a coupling capacitor and AC signal source. This is the AC signal source which is a function generator which can generate different types of wave forms; sine, square, triangle. I have chosen the sine wave and these are for selecting the frequency. These are for varying the frequency and the amplitude. I have kept around 1K Hz frequency and I have kept small voltage, few millivolts. This input is given at this point corresponding to this input at the capacitor and I have a power supply V_{CC} which is again about 9.8V that is being applied here and the output I am applying to an oscilloscope. This is the oscilloscope. I have two channels; channel one and channel two. I connect the channel one to the input side and connect the channel two to the output side and in the two channels this will be the input and this will be the output. Right now I have not switched on so you are not able to see the signal.

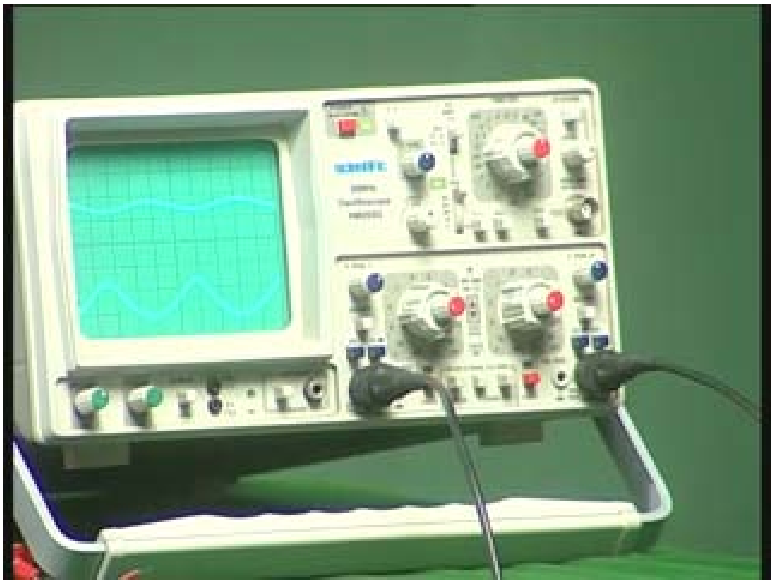
[Refer Slide Time: 50:53]



This oscilloscope which I can use to find the output signal and the input signal simultaneously this is the function generator which I am using as the signal source and applying the signal to the input of the amplifier. This is a power supply which I am using for the V_{CC} supply that I require for my transistor amplifier.

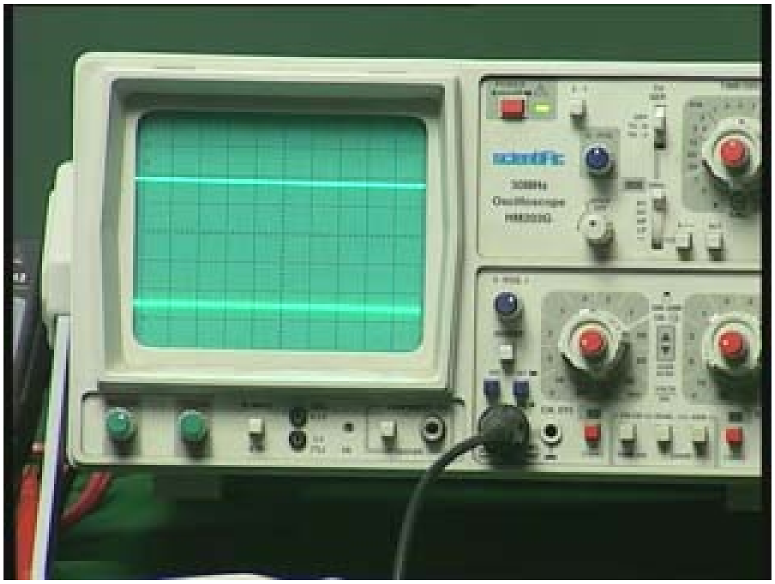
Let me quickly apply the AC volts at the input and both the signals are seen. This small amplitude is the input signal.

[Refer Slide Time: 51:46]



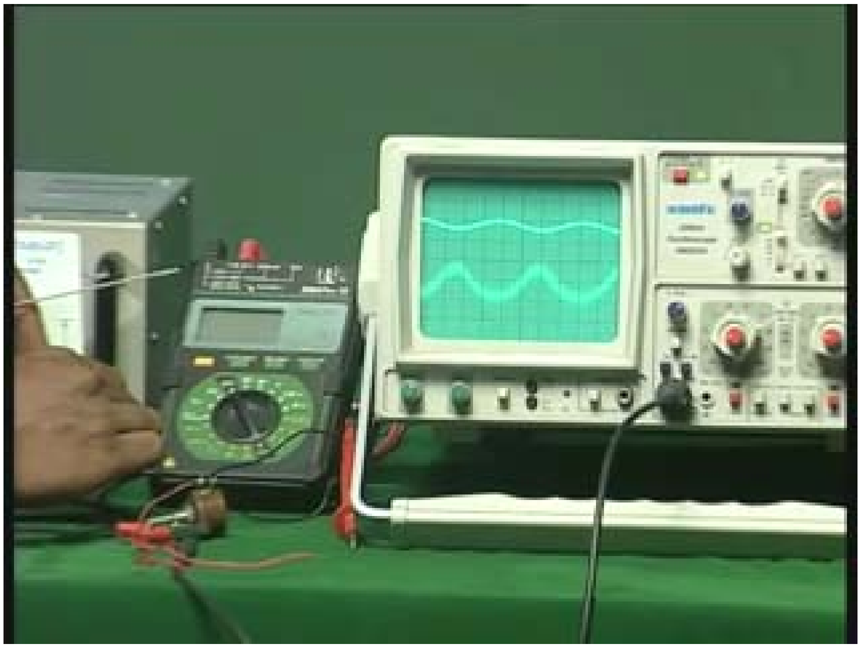
If I remove the signal the output will go blank, straight.

[Refer Slide Time: 51:55]



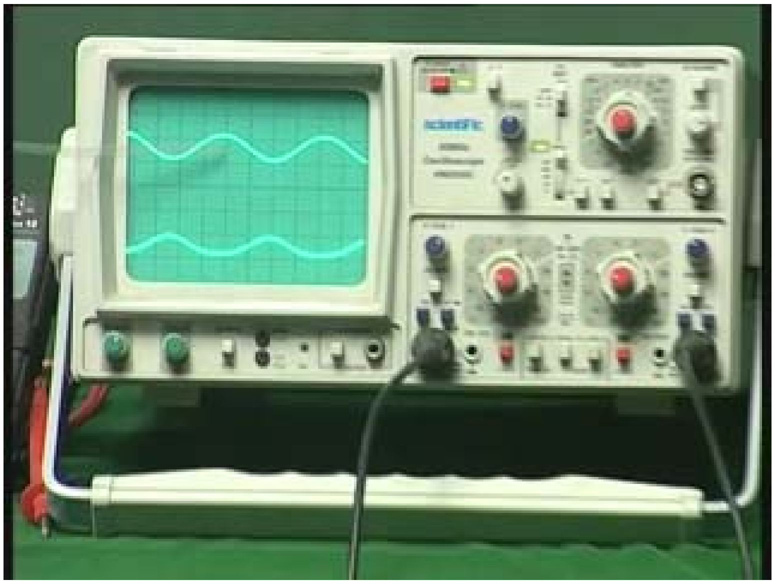
The input also will go. This is the input signal. This is the output signal. If I remove it the output will go flat. There are only noise signals.

[Refer Slide Time: 52:07]



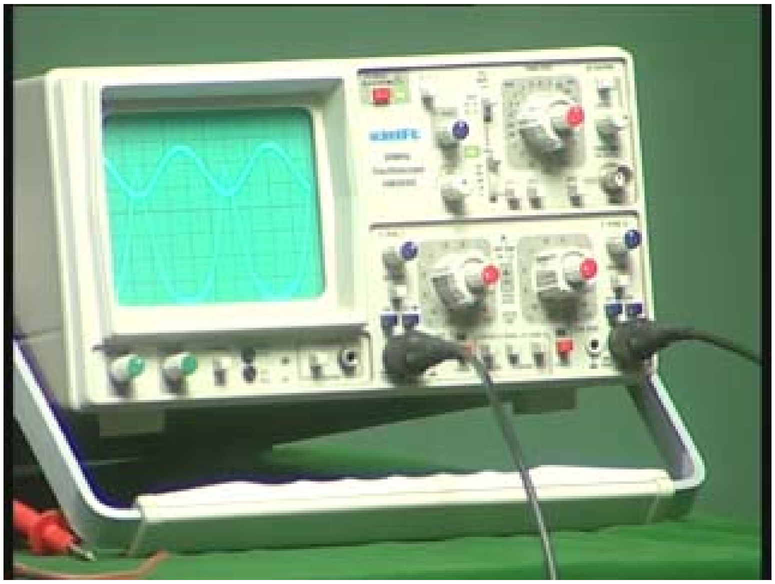
When I connect the output I get an amplified signal. How do I know there is amplification? For that there are amplifiers here in the oscilloscope which is maintained at different points. This is the highest magnification that corresponds to about 5 millivolts. This is around 10 millivolts and this is about 20 millivolts. Input signal is within one division peak to peak.

[Refer Slide Time: 52:54]



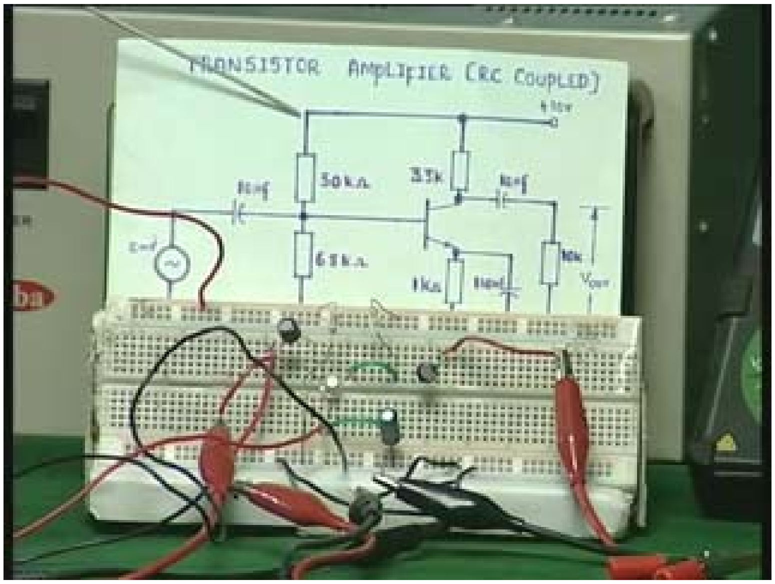
That means it is about 10 millivolts. This is the input signal that I am applying. The output signal is the amplified version. If I increase the input signal the output signal also increases. But there is also a distortion here. This is not an exact sine wave. Now what I will do is I will change one of the resistors with the potentiometer and then try to see what happens? That is I am removing the R_E; I am removing the emitter resistance. I am replacing it with a potentiometer. It is a 10K ohm potentiometer. If I now vary the potentiometer after connecting the power supply you can see highly distorted output. If I now decrease or increase you can see the distortion will increase or decrease. One can choose without distortion by changing this. When I do that I am able to reduce the distortion. If I increase it you will see the distortion increases.

[Refer Slide Time: 55:01]



I want to indicate to you that the choice of the various resistors especially the R_E which is the biasing resistor is very important in choosing the performance of the amplifier. So you do get amplification gain even though we have not actually calculated the gain in this case but we do get enough amplification. The biasing circuit is very important. After that you have to use coupling capacitors and this is the bypass capacitor. Once you do that what you get is a RC coupled amplifier of one single stage.

[Refer Slide Time: 55:45]



If you want larger amplification you can actually go for one more of this and couple this to the base of the next amplifier and still you can have further amplification. One can actually cascade different amplifiers and get very large gains by having multistage amplifiers. We will see the some of the basic theory and the model behind the transistors and then about the construction of multistage amplifiers, etc., in the next lecture. In the next lecture we will try to see how to model a transistor and then how one can design a basic, simple amplifier using that model and obtain the amplification factor and other factors. Thank you!