The McGraw·Hill Companies

and Synthesis of One-port Networks

CHAPTER 18

18.1 HURWITZ POLYNOMIALS

As stated in Chapter 15, the poles of the stable system must lie on the left half of the s-plane. Any network function can be written as the ratio of two polynomials, and is given by

$$Z(s) = \frac{P(s)}{Q(s)}$$

A polynomial must satisfy the following conditions.

(a) $Z(s)$ must be a real function of s

$$Z(s) = \frac{P(s)}{Q(s)} = \frac{a_0 s^n + a_1 s^{n-1} + \cdots + a_{n-1} s + a_n}{b_0 s^m + b_1 s^{m-1} + \cdots + b_{m-1} s + b_m}$$

where all the quotients a_i, b_j are real, and hence $Z(s)$ is real if s is real.

(b) All the roots of $P(s)$ must have zero real parts, or negative real parts.

Hurwitz polynomials have the following properties.

1. All the quotients in the polynomial

   $$P(s) = a_0 s^n + a_1 s^{n-1} + \cdots + a_{n-1} s + a_n$$

   are positive. A polynomial may not have any missing terms between the highest and the lowest order unless all even or all odd terms are missing. For example, the polynomial $P(s) = s^5 + 3s^3 + 5s^2 + 2s + 1$ is not Hurwitz as the term s^4 is missing. At the same time, the polynomial $P(s) = s^3 + 3s$ is Hurwitz because all quotient terms are positive and all even terms are missing.

2. The roots of the odd and even parts of a Hurwitz polynomial $P(s)$ lie on the $j\omega$ axis. Consider the polynomial $P(s)$ having odd and even parts $o(s)$ and $e(s)$, respectively; then

   $$P(s) = o(s) + e(s)$$

   Both have roots on the $j\omega$ axis.

3. If the polynomial $P(s)$ is either even or odd, the roots of $P(s)$ lie on the $j\omega$ axis.

4. All the quotient terms are positive in the continued fraction expansion of the ratio of the odd to even, or even to odd parts of the polynomial $P(s)$. Consider a polynomial

   $$P(s) = s^4 + s^3 + 6s^2 + 3s + 4$$

   The even parts of the polynomial, $e(s) = s^4 + 6s^2 + 4$

   The odd parts of the polynomial $o(s) = s^3 + 3s$

The continued fraction expansion is given by

$$
\begin{array}{l}
s^3+3s)\,s^4+6s^2+4\,(s \\
\qquad\qquad \underline{s^4+3s^2} \\
\qquad\qquad\qquad 3s^2+4)\,s^3+3s\,(\frac{s}{3} \\
\qquad\qquad\qquad\qquad\qquad \underline{s^3+\frac{4s}{3}} \\
\qquad\qquad\qquad\qquad\qquad\quad \frac{5s}{3})\,3s^2+4\,(\frac{9s}{5} \\
\qquad\qquad\qquad\qquad\qquad\qquad\quad \underline{3s^2} \\
\qquad\qquad\qquad\qquad\qquad\qquad\qquad\quad 4)\,\frac{5s}{3}\,(\frac{5s}{12} \\
\qquad\qquad\qquad\qquad\qquad\qquad\qquad\qquad \underline{\frac{5s}{3}} \\
\qquad\qquad\qquad\qquad\qquad\qquad\qquad\qquad\ \ 0
\end{array}
$$

The continued fraction expansion can be written

$$c(s)=\frac{e(s)}{o(s)}=s+\cfrac{1}{\cfrac{s}{3}+\cfrac{1}{\cfrac{9s}{5}+\cfrac{1}{\cfrac{5s}{12}}}}$$

Since all the quotient terms are positive, the polynomial $P(s)$ is Hurwitz.

5. If the polynomial satisfies the condition of Hurwitz, then the polynomial must be Hurwitz to within an even multiplicative factor $\omega(s)$, that is, if

$$P_1(s) = \omega(s)\,P(s), \text{ then } P(s) \text{ is Hurwitz}$$

If $\omega(s)$ is Hurwitz, $P_1(s)$ must be Hurwitz.
Consider the polynomial $P_1(s) = s^3 + 3s^2 + 6s + 18$
The continued fraction expansion is obtained from the division

$$
\begin{array}{l}
3s^2+18\big)\,s^3+6s\big(s/3 \\
\qquad\qquad \underline{s^3+6s} \\
\qquad\qquad\quad 0
\end{array}
$$

The continued fraction expansion has been terminated abruptly. So, the polynomial can be written as

$$P_1(s)=(s^3+6s)\left(1+\frac{3}{s}\right)$$

Here $(1 + 3/s)$ term is Hurwitz. Since the terms $(s^3 + 6s)$ is Hurwitz, then $P_1(s)$ also is Hurwitz.

6. If the ratio of the polynomial $P(s)$ and its derivative $P(s)$ gives a continued fraction expansion with all positive coefficients, then the polynomial $P(s)$ is Hurwitz.
   Consider the polynomial

   $$P(s) = s^4 + 3s^2 + 2$$

   The derivative is $P'(s) = 4s^3 + 6s$
   By taking continued fraction expansion, we get

   $$4s^3 + 6s)\, s^4 + 3s^2 + 2\, (s/4$$
   $$\underline{s^4 + \frac{6}{4}s^2}$$
   $$\frac{3}{2}s^2 + 2\Big)\, 4s^3 + 6s\, \Big(\frac{8}{3}s$$
   $$\underline{4s^3 + \frac{16}{3}s}$$
   $$\frac{2}{3}s\Big)\, \frac{3}{2}s^2 + 2\, \Big(\frac{9}{4}s$$
   $$\underline{\frac{3}{2}s^2}$$
   $$2\Big)\, \frac{2}{3}s\, \Big(\frac{1}{3}s$$
   $$\underline{\frac{2}{3}s}$$
   $$0$$

Since all the quotients in the continued fraction expansion are positive, the polynomial $P(s)$ is Hurwitz.

18.2 POSITIVE REAL FUNCTIONS

As discussed in Chapter 15, the driving point impedance function $Z(s)$ and driving point admittance function $Y(s)$ of a one-port network can be expressed as the ratio of two polynomials,

$$Z(s) = Y(s) = \frac{P(s)}{Q(s)} = \frac{a_0 s^n + a_1 s^{n-1} + \cdots + a_{n-1}s + a_n}{b_0 s^m + b_1 s^{m-1} + \cdots + b_{m-1}s + b_m}$$

Functions possessing the following properties are called positive real functions, and are abbreviated as *prf*.

1. When s is real, $Z(s)$ and $Y(s)$ are real functions because the quotients of the polynomials $P(s)$ and $Q(s)$, that is, a_k and b_k are real. When $Z(s)$ is determined from the impedances of the individual branches, the quotients a_k and b_k are obtained by adding together, multiplying or dividing the branch parameters which are real.

2. The poles are zeros of $Z(s)$ and $Y(s)$ all lie in the left half of the s-plane, or on the imaginary axis of the s-plane. In the later case, the poles and zeros are simple.

   From the above property it should be noted that if the roots of the characteristic equation were lying on the imaginary axis, and the roots $s = \pm j\omega$, were multiples, the solution of the characteristic equation would be of the form

   $$x_t = (c_0 + c_1 t + c_2 t^2 + \cdots + c_{m-1}\, t^{m-1})\sin \omega_1 t$$

   This would cause the transients to build up, which cannot happen in a passive one-port. Under these conditions, all quotients a_n and b_n of the polynomials $P(s)$ and $Q(s)$ must be positive. This can be proved by writing the polynomial $P(s)$ as

   $$P(s) = a_0 s^n + a_1 s^{n-1} + \cdots + a_n$$

   $$= a_0(s - s_1)\,(s - s_2) \cdots (s - s_n)$$

   For each pair of complex and conjugate roots, $s_k = + j\omega_k$ and $s_{k+1} = - j\omega_k$, we have

   $$(s - s_k)(s - s_{k+1}) = (s - j\omega_k)\,(s + j\omega_k)$$

   $$= s^2 + \omega_k^2$$

   For real roots of s_k, all the quotients of s in $s^2 + \omega_k^2$ of the polynomial $P(s)$ are non-negative. So by multiplying all factors in $P(s)$, we find that all quotients $a_0, a_1, \cdots, a_n$ are positive.

3. The real parts of the driving point functions $Z(s)$ and $Y(s)$ are positive, or zero, that is, Re $Z(s) > 0$ or Re $Y(s) > 0$ provided for all Re$(s) > 0$.

   Let $\quad Z(s) = \dfrac{P(s)}{Q(s)}$

   where $P(s)$ and $Q(s)$ are polynomials in s and have real coefficients. Hence, $Z(s)$ is real, when s is real. Further, $P(s)$ and $Q(s)$, are real when s is real. Since the poles and zeros of a network function $Z(s)$ are real, complex zeros must appear in conjugate pairs.

18.3 FREQUENCY RESPONSE OF REACTIVE ONE-PORTS

Based on the locations of zeros and poles, a reactive one-port can have the following four types of frequency response.

1. A frequency response with two external poles is shown in Fig. 18.1(a). In this case the driving point impedance with poles at $\omega = 0$ and $\omega = \infty$ must have an s in the denominator polynomial and one excess term $(s^2 + \omega_n^2)$ in the numerator than in the denominator.

   $$\therefore \qquad Z(s) = \frac{H(s^2 + \omega_1^2)\cdots(s^2 + \omega_n^2)}{s(s^2 + \omega_2^2)\cdots(s^2 + \omega_{n-1}^2)}$$

   The driving point impedance of the one-port is infinite, and it will not

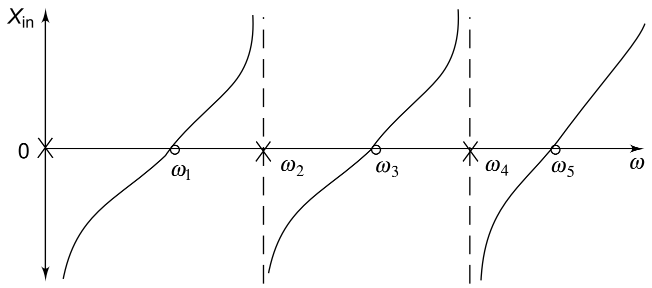


Fig. 18.1 (a)

pass either direct current ($\omega = 0$) or alternating current of an infinitely high frequency.

2. A frequency response with two external zeros is shown in Fig. 18.1(b). In this case the driving point impedance with zeros at $\omega = 0$ and $\omega = \infty$ must have an s term in the numerator and an excess $(s^2 + \omega_n^2)$ term in the denominator polynomial.

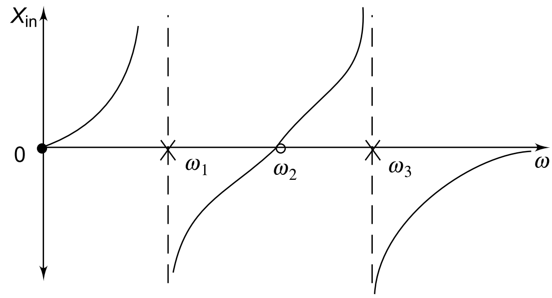


Fig. 18.1 (b)

$$\therefore \qquad Z(s) = \frac{Hs(s^2+\omega_2^2)\cdots(s^2+\omega_{n-1}^2)}{(s^2+\omega_1^2)\cdots(s^2+\omega_n^2)}$$

The driving point impedance of the one-port is zero, and it will pass both direct current and an alternating current of an infinitely high frequency.

3. A frequency response with an external zero at $\omega = 0$ and an external pole at $\omega = \infty$ is shown in Fig. 18.1(c). In this case, the driving point impedance with zero at $\omega = 0$ and pole at $\omega = \infty$ must have a term s in the numerator and equal number of $(s^2 + \omega_n^2)$ type terms in the numerator and the denominator.

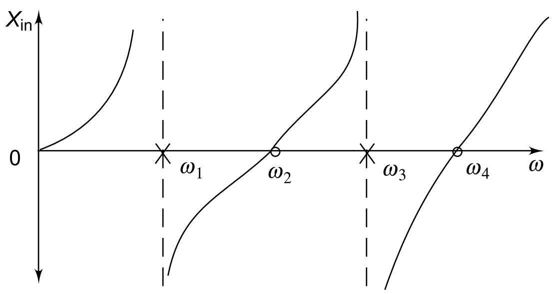


Fig. 18.1 (c)

$$\therefore \quad Z(s) = \frac{Hs(s^2+\omega_2^2)\cdots(s^2+\omega_n^2)}{(s^2+\omega_1^2)\cdots(s^2+\omega_{n-1}^2)}$$

In this case, the one-port will pass direct current and block an alternating current of an infinitely high frequency.

4. A frequency response with an external pole at $\omega = 0$ and an external zero at $\omega = \infty$ is shown in Fig. 18.1 (d). In this case, the driving point impedance with pole at $\omega = 0$ and zero at $\omega = \infty$ must have a term s in the denominator and equal number of $(s^2 + \omega_n^2)$ terms in the numerator and the denominator.

$$\therefore \qquad Z(s) = \frac{H(s^2+\omega_1^2)\cdots(s^2+\omega_{n-1}^2)}{s(s^2+\omega_2^2)\cdots(s^2+\omega_n^2)}$$

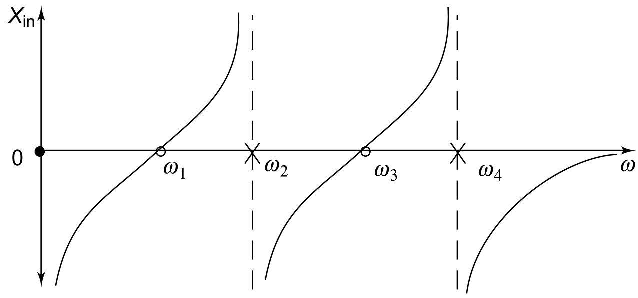


Fig. 18.1 (d)

Here, the one-port will block direct current and pass an alternating current of an infinitely high frequency.

The function of factor H is to fix the scale of the reactance, and hence it is referred to as the *multiplying factor*, or the *scale factor*. It is to be noted that as the number of zeros and poles in X_{in} (ω) increases, there will be an increasing number of reactive one-ports having the same form of frequency response.

18.4 SYNTHESIS OF REACTIVE ONE-PORTS BY FOSTER'S METHOD

The driving point function of a reactive one-port $Z(s)$ is given by

$$Z(s) = \frac{H(s^2+\omega_1^2)(s^2+\omega_3^2)(s^2+\omega_5^2)\cdots}{s(s^2+\omega_2^2)(s^2+\omega_4^2)(s^2+\omega_6^2)\cdots} \tag{18.1}$$

Let us determine the circuit and parameters that implement its frequency response $Z_{in}(j\omega) = jX_{in}(\omega)$. There are two forms of Foster networks for reactive one-ports. One is a series combination of parallel LC circuits with capacitance C_0 and inductance L_∞ as shown in Fig. 18.2 known as first Foster form or Impedance form.

The other form (known as second Foster form or admittance form) is a parallel combination of series LC circuits with inductance L_0 and capacitance C_∞ as shown in Fig. 18.3.

To synthesize the impedance form or first Foster form, we shall write the expression for LC parallel combination in the network of Fig. 18.2

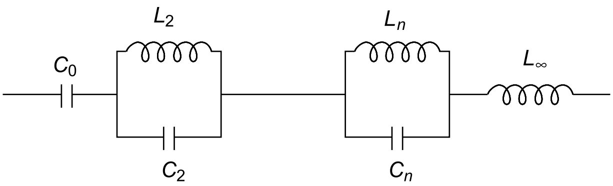


Fig. 18.2

$$Z(s) = \frac{1}{Cs + \frac{1}{Ls}} = \frac{\left(\frac{1}{C}\right)s}{s^2 + \frac{1}{LC}} \tag{18.2}$$

To synthesise the first Foster network, the first step is to express $Z(s)$ as the sum of rational fractions of the form of Eq. 18.2, to which is added the term $1/C_0 s$ and $L_\infty s$

Equation 18.1 can be written as

$$Z(s) = \frac{P_0}{s} + \frac{2P_2\,s}{s^2 + \omega_2^2} + \frac{2P_4 s}{s^2 + \omega_4^2} + \cdots + Hs \tag{18.3}$$

If we divide the total impedance into a series combination of impedances $Z_1(s)$, $Z_2(s)$, $\cdots$, $Z_n(s)$.

$$Z(s) = Z_1(s) + Z_2(s) + Z_3(s) + \cdots + Z_n(s) \tag{18.4}$$

By comparing Eqs 18.3 and 18.4, we have impedance $Z_1(s) = P_0/s$ that represents a capacitor C_0 of value $1/P_0$, and the impedance $Z_n(s) = Hs$ that represents an inductor L_∞ of value H henrys. The remaining intermediate terms represents parallel combination of an inductor and a capacitor. By comparing Eq. 18.2 and the middle terms of Eq. 18.3, we get

$$C_n = \frac{1}{2P_n} \text{ and } L_n = \frac{2P_n}{\omega_n^2}$$

where n refers to the term $2P_n s/s^2 + \omega_n^2$ in Eq. 18.3.

The presence of first element capacitor C_0 and the last element inductor L_∞ depends on the pole-zero configuration. If there is pole at $\omega = 0$, the first element C_0 is present in the network. Similarly, if there is pole at $\omega = \infty$, the last element L_∞ is present in the network.

The second canonical form, known as the second Foster network, is a parallel combination of series LC circuits. Because all branches in the network of Fig. 18.3 are connected in parallel, the network can be simplified by taking the driving point admittance $Y(s)$. Therefore, we have

$$Y(s) = \frac{H}{s}\frac{(s^2 + \omega_1^2)(s^2 + \omega_3^2)}{(s^2 + \omega_2^2)(s^2 + \omega_4^2)} \tag{18.5}$$

To synthesise the parallel Foster network, we shall write the expression for LC series combination in the network of Fig. 18.3.

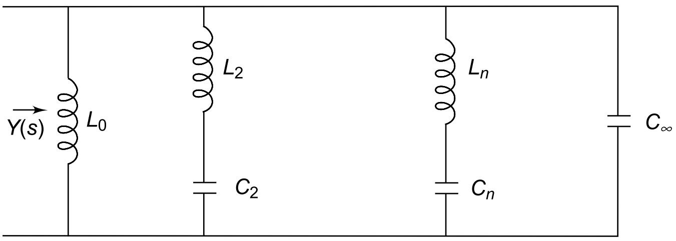


Fig. 18.3

$$Y(s) = \frac{1}{Ls + \frac{1}{Cs}} = \frac{(s/L)}{s^2 + \frac{1}{LC}} \tag{18.6}$$

Now to synthesise the second Foster network, the first step is to express $Y(s)$ as the sum of rational fractions of Eq. 18.6, to which is added the term $C_\infty s$ and $1/L_0 s$.

Equation 18.5 can be written as

$$Y(s) = \frac{P_0}{s} + \frac{2P_2 s}{s^2 + \omega_2^2} + \frac{2P_4 s}{s^2 + \omega_4^2} + \cdots + Hs \tag{18.7}$$

If we divide the total admittance into a parallel combination of admittance $Y_1(s)$, $Y_2(s)$, $\cdots$, $Y_n(s)$

$$\therefore \qquad Y(s) = Y_1(s) + Y_2(s) + \cdots + Y_n(s) \tag{18.8}$$

By comparing Eqs 18.7 and 18.8, we have the admittance

$Y_1(s) = P_0/s$ which represents an inductor L_0 of value $1/P_0$, and the admittance $Y_n(s) = Hs$ which represents a capacitor C_∞ of value H. The remaining intermediate terms represents series combination of an inductor and a capacitor. By comparing Eq. 18.6 and middle terms of Eq. 18.7, we get

$$L_n = \frac{1}{2P_n} \text{ and } C_n = \frac{2P_n}{\omega_n^2}$$

where n refers to the terms $2P_n s/(s^2 + \omega_n^2)$ in Eq. 18.7.

The presence of first element inductor L_0 and the last element capacitor C_∞ depends on the pole-zero configuration. If there is pole at $\omega = 0$, the first element L_0 is present in the network. Similarly, if there is pole at $\omega = \infty$, the last element C_∞ is present in the network.

Example 18.1 *The driving point impedance of a one-port reactive network is given by*

$$Z(s) = 5\frac{(s^2+4)(s^2+25)}{s(s^2+16)}$$

Obtain the first and second Foster networks.

Solution Since, there is an extra term in the numerator compared to the denominator, and also an s term in the denominator, the two poles exists at $\omega = 0$ and at $\omega = \infty$. Therefore, the network consists of first element and last element.

By taking the partial fraction expansion of $Z(s)$, we have

$$Z(s) = \frac{P_0}{s} + \frac{P_2}{s + j4} + \frac{P_2^*}{s - j4} + Hs$$

By applying the Heaviside method, from the above equation we have

$$P_0 = \left.\frac{5(s^2+4)(s^2+25)}{s^2+16}\right|_{s=0}$$

$$= \frac{5 \times 4 \times 25}{16} = \frac{125}{4}$$

$$P_2 = \left.\frac{5(s^2+4)(s^2+25)}{s(s-j4)}\right|_{s=-j4} = \frac{135}{8}$$

By inspection, $H = 5$

Therefore, $C_0 = \frac{1}{P_0} = \frac{4}{125}$ Farad

$$L_\infty = H = 5 \text{ H}$$

$$C_2 = \frac{1}{2P_2} = \frac{8}{2 \times 135} = \frac{8}{270} \text{ F}$$

$$L_2 = \frac{2P_2}{\omega_n^2} = \frac{2 \times 135}{16 \times 8} = \frac{135}{64} \text{ H}$$

The element values in the first Foster form are shown in Fig. 18.4.

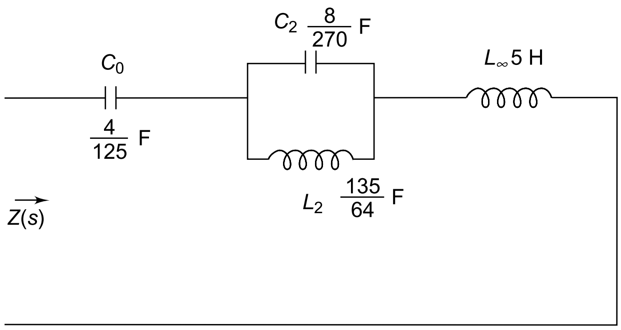


Fig. 18.4

To find the second Foster form, first we have to take the function into admittance form.

$$Y(s) = \frac{s(s^2+16)}{5(s^2+4)(s^2+25)}$$

Since, there is an s term in the numerator and an excess term in the denominator, the two zeros exists at $\omega = 0$ and at $\omega = \infty$. Therefore, the network consists of a series LC combination of parallel elements.

By taking the partial fraction expansion of $Y(s)$, we get

$$Y(s) = \frac{2P_1 s}{s^2+4} + \frac{2P_2 s}{s^2+25}$$

By applying the Heaviside method, we get

$$P_1 = \frac{1}{5}\left.\frac{s(s^2+16)}{(s-j2)(s^2+25)}\right|_{s=-j2} = \frac{2}{35}$$

$$P_2 = \frac{1}{5}\left.\frac{s(s^2+16)}{(s^2-j2)(s^2+25)}\right|_{s=-j2} = \frac{2}{35}$$

Therefore, the elemental values are

$$L_1 = \frac{1}{2P_1} = \frac{35}{4}\text{H}$$

$$C_1 = \frac{2P_1}{\omega_1^2} = \frac{1}{35}\text{F}$$

$$L_2 = \frac{1}{2P_2} = \frac{35}{3}\text{H}$$

$$C_2 = \frac{2P_2}{\omega_2^2} = \frac{2\times 3}{70\times 25} = \frac{3}{875}\text{F}$$

The circuit of second Foster form is shown in Fig. 18.5.

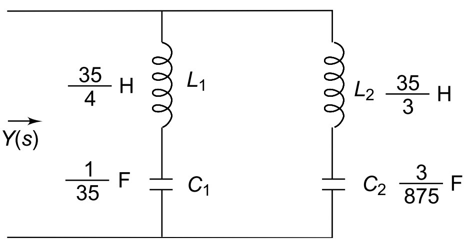


Fig. 18.5

18.5 SYNTHESIS OF REACTIVE ONE-PORTS BY THE CAUER METHOD

In the Cauer method, there are two types of ladder networks to realise the one-port network. In one type of network, the series arms are inductors and the shunt arms are capacitors as shown in Fig. 18.6 (a).

In the other network, the series arms are capacitors and the shunt arms are inductors as shown in Fig. 18.6 (b).

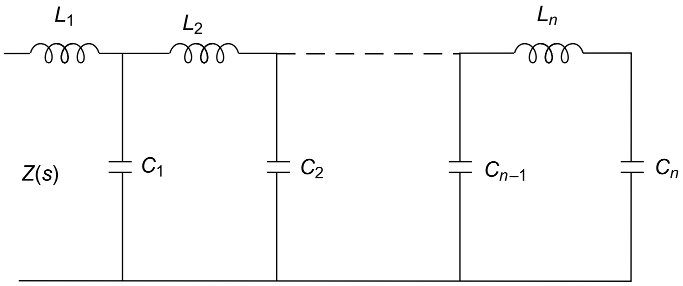


Fig. 18.6 (a)

From the driving point function $Z(s)$ or $Y(s)$, there is always a zero or a pole at $s = \infty$. We can remove this pole or zero by remaining an impedance $Z_1(s)$ or

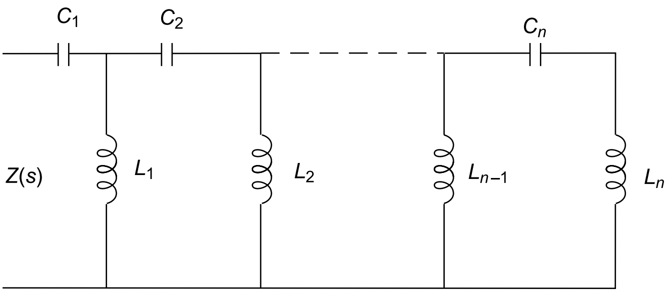


Fig. 18.6 (b)

admittance $Y_1(s)$. Then from each remainder left, an inductor or a capacitor is removed, depending upon the driving point function. It may be an impedance or an admittance function. This process continues until the remainder is zero. From the above, the impedance $Z(s)$ may be written as a continued fraction as under.

$$Z(s) = Z_1(s) + \cfrac{1}{Y_2(s) + \cfrac{1}{Z_3(s) + \cfrac{1}{Y_4(s) + \cfrac{1}{Z_5(s) + \cdots}}}}$$

Let us realise the first Cauer form. Consider a driving point function having a pole at infinity. This implies that the degree of the numerator is greater than that of the denominator. We always remove pole at infinity by inverting the remainder, and dividing. That means an *LC* driving point function can be synthesised by the continued fraction expansion.

If $Z(s)$ is the function to be synthesised, then the continued fraction expansion is as follows.

$$Z(s) = L_1 s + \cfrac{1}{C_1 s + \cfrac{1}{L_2 s + \cfrac{1}{C_2 s + \cdots}}}$$

Therefore, in the first Cauer network shown in Fig. 18.6 (a), the inductors are connected in series and the capacitors are connected in shunt.

If the driving point function, $Z(s)$ has zero at infinite, that is, if the degree of its numerator is less than that of its denominator, the driving point function is inverted. In this case, the continued fraction will give a capacitive admittance as first element, and a series inductance.

Now let us realise the second Cauer network. In this case, the removal of the pole at zero gives the network shown in Fig. 18.6 (b), where the capacitors are connected in series and the inductors are connected in shunt. If $Z(s)$ is the function to be synthesised, then the continued fraction expansion is

$$Z(s) = \frac{1}{C_1 s} + \cfrac{1}{\cfrac{1}{L_1 s} + \cfrac{1}{\cfrac{1}{C_2 s} + \cfrac{1}{\cfrac{1}{L_2 s} + \cdots}}}$$

If the driving point function, $Z(s)$ has a zero at zero, the continued fraction expansion will give an inductive admittance as first element and a series capacitance.

From the above discussion, we can conclude that in the first Cauer network, the first element in a series inductor when the driving point function consists of a pole at infinity, and it is a shunt capacitor when the driving point function consists of zero at infinity. Similarly, the last element is an inductor when the function consists of zero at $\omega = 0$, and it is a capacitor when the function consists of pole at $\omega = 0$.

In case of second Cauer network, the first element is a series capacitor when the driving point function consists of a pole at zero and it is shunt inductance when the function consists of a zero at zero. Similarly, the last element is an inductor when the driving point function consists of a pole at infinity; and it is a capacitor when impedance function consists of zero at infinity.

Example 18.2 *The driving point impedance of an LC network is given by*

$$Z(s) = \frac{2s^5 + 12s^3 + 16s}{s^4 + 4s^2 + 3}$$

Determine the first Cauer form of the network.

Solution By taking continued fraction expansion, we get

$$s^4 + 4s^2 + 3)\, 2s^5 + 12s^3 + 16s\, (2s - L_1$$
$$\underline{2s^5 + 8s^3 + 6s}$$
$$4s^3 + 10s)\, s^4 + 4s^2 + 3\, (\frac{s}{4} - C_2$$
$$\underline{s^4 + \frac{5}{2}s^2}$$
$$\frac{3s^2}{2} + 3)\, 4s^3 + 10s\, (\frac{8}{3}s - L_3$$
$$\underline{4s^3 + 8s}$$
$$2s)\, \frac{3s^2}{2} + 3\, (\frac{3}{4}s - C_3$$
$$\underline{\frac{3s^2}{2}}$$
$$3)\, 2s\, (\frac{2}{3}s - L_5$$
$$\underline{2s}$$
$$0$$

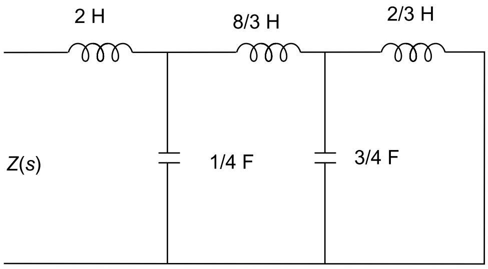


Fig. 18.7

Hence,

$$Z(s) = 2s + \cfrac{1}{\cfrac{s}{4} + \cfrac{1}{\cfrac{8}{3}s + \cfrac{1}{\cfrac{3}{4}s + \cfrac{1}{\cfrac{2}{3}s}}}}$$

The resulting network shown in Fig. 18.7 is called the first Cauer form.

Example 18.3 *The driving point impedance of an LC network is given by*

$$Z(s) = s^4 + 4s^2 + 3/(s^3 + 2s)$$

Determine the second Cauer form of the network.

Solution To obtain the second Cauer form, we have to arrange the numerator and the denominator of given Z(s) in assending powers of s before starting the continued fraction expansion.

By taking continued fraction expansion, we get

$$2s + s^3)\,3 + 4s^2 + s^4\,(\frac{3}{2s} - C_1$$

$$3 + \frac{3}{2}s^2$$

$$\frac{5s^2}{2} + s^4)\,2s + s^3\,(\frac{4}{5s} - L_2$$

$$2s + \frac{4}{5}s^3$$

$$\frac{s^3}{5})\,\frac{5s^2}{2} + s^4\,(\frac{25}{2s} - C_3$$

$$\frac{5s^2}{2}$$

$$s^4)\,\frac{s^3}{5}\,(\frac{1}{5s} - L_4$$

$$\frac{s^3}{5}$$

$$0$$

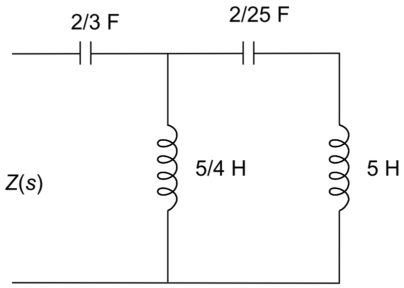


Fig. 18.8

Hence, $$Z(s) = \frac{3}{2s} + \cfrac{1}{\cfrac{4}{5s} + \cfrac{1}{\cfrac{25}{2s} + \cfrac{1}{\cfrac{1}{5s}}}}$$

The resulting network shown in Fig. 18.8 is called the second Cauer form.

18.6 SYNTHESIS OF *R-L* NETWORK BY THE FOSTER METHOD

The driving point impedance function of an *RL* network $Z(s)$ is given by

$$Z(s) = \frac{H(s+\sigma_1)(s+\sigma_3)\cdots}{(s+\sigma_2)(s+\sigma_4)\cdots} \tag{18.9}$$

The first form of the Foster network is shown in Fig. 18.9.

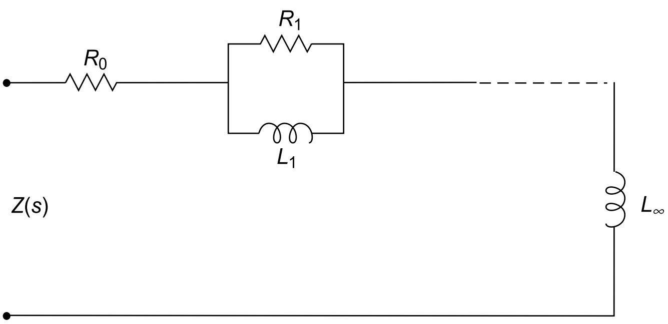


Fig. 18.9

The above impedance function possess the following properties.

(i) The poles and zeros of the *RL* driving point impedance function are located on the negative real axis of the *s*-plane.

(ii) Poles and zeros alternate along the negative real axis.

(iii) The singularity at the origin, or $s = 0$ is a zero.

(iv) The singularity at $s = \infty$ is a pole.

(v) The slope of the impedance curve is positive.

(vi) The impedance at $s = \infty$ is always greater than the impedance at $\omega = 0$.

(vii) The residues at the poles of $Z(s)$ are real and negative. The residues of $Z(s)/s$ are real and positive.

To synthesise the first Foster network, we shall write the expression for the *RL* parallel combination in the network of Fig. 18.9.

$$Z_1(s) = \frac{R_1 s}{s + \sigma_1}$$

where $$\sigma_1 = \frac{R_1}{L_1} \tag{18.10}$$

or $$\frac{Z_1(s)}{s} = \frac{R_1}{s + \sigma_1}$$

We have the another form of the equation as discussed in chapter 15.

$$Z(s) = \frac{a_0 s^n + a_1 s^{n-1} + \cdots + a_n}{b_0 s^m + b_1 s^{m-1} + \cdots + b_m} \tag{18.11}$$

where $n > m$

The degree of the numerator is greater than that of the denominator by one. At $s = 0$

$$Z(s) = \frac{a_n}{b_n} \text{ (when } a_n \neq 0)$$

$$= 0 \text{ (when } a_n = 0)$$

And at $s = \infty$, $$Z(s) = s\left(\frac{a_0}{b_0}\right) \text{ (when } a_0 \neq 0)$$

$$= \frac{a_1}{b_1} \text{ (when } a_0 = 0)$$

By separating the constant term and linear term in Eq. 18.11, the *RL* impedance function can be written as

$$Z(s) = P_0 + \frac{P_i s}{s + \sigma_i} + \cdots + Hs \tag{18.12}$$

If we divide the total impedance into a series combination of impedance $Z_1(s)$, $Z_2(s)$, $\cdots$ $Z_n(s)$

$$Z(s) = Z_1(s) + Z_2(s) + \cdots + Z_n(s) \tag{18.13}$$

By comparing Eqs 18.12 and 18.13, we have the impedance $Z_1(s) = P_0$, which is constant. The term P_0 represents a resistor R_0, and the impedance $Z_n(s) = Hs$ represents L_∞ of value H henrys. The remaining terms represent parallel combination of an inductor and a resistor. By comparing Eq. 18.10 and middle terms of Eq. 18.12, we have

$$P_n = R_n \text{ and } \sigma_n = \frac{R_n}{L_n}$$

where n refers to the term $P_n s/(s + \sigma_n)$ in Eq. 18.12.

Consider a function $$Z(s) = 5\frac{(s+1)(s+4)}{(s+3)(s+5)}$$

$Z(s)$ represents *RL* impedance, because it satisfies all the properties, but the signs of $Z(s)$ at its poles are negative as shown.

$$Z(s)=\frac{5(s+1)(s+4)}{(s+3)(s+5)}=5-\frac{5}{s+3}-\frac{10}{s+5}$$

Therefore, we have to expand $\frac{Z(s)}{s}$

$$\frac{Z(s)}{s}=\frac{5(s+1)(s+4)}{s(s+3)(s+5)}=\frac{4}{3s}+\frac{5}{3(s+3)}+\frac{2}{s+5}$$

If we multiply both sides by s, we get

$$Z(s)=\frac{4}{3}+\frac{5}{3}\frac{s}{s+3}+\frac{2s}{s+5}$$

Hence, the impedance $Z(s)$ can be realised as a series Foster form of RL network shown in Fig. 18.10.

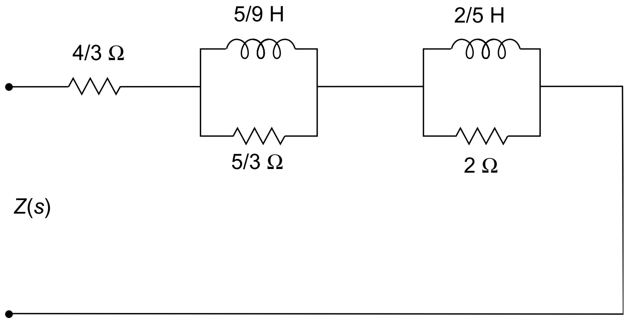


Fig. 18.10

Similarly, the driving point admittance function of the RL network, $Y(s)$ is given by

$$Y(s)=\frac{b_0s^m+b_1s^{m-1}+\cdots+b_m}{a_0s^n+a_1s^{n-1}+\cdots+a_n} \tag{18.14}$$

The second form of the Foster network is shown in Fig. 18.11.

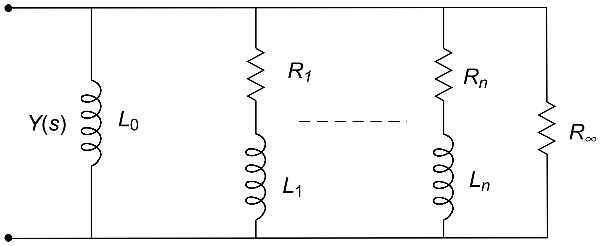


Fig. 18.11

The above admittance function must possess the following properties.

(i) The poles and zeros of the RL driving point admittance function are located on the negative real axis of the s-plane.

(ii) Poles and zeros alternate along the negative real axis.
(iii) The singularity at the origin, or $s = 0$, is a pole.
(iv) The singularity at $s = \infty$ is a zero.
(v) The slope of the admittance curve is negative.
(vi) The admittance at $s = 0$ is always greater than the admittance at $s = \infty$.
(vii) The residues at the poles of $Y(s)$ are real and positive.

The *RL* admittance function can be written as

$$Y(s) = \frac{P_0}{s} + \frac{P_i}{s+\sigma_i} + \cdots + H \tag{18.15}$$

If we observe Eq. 18.15, we have the first term P_0/s representing inductance $L_0 = 1/P_0$, and the last term representing a resistance $R_\infty = H$. The intermediate terms represent admittance function of the series *RL* network. We, therefore, have

$$Y_n = \frac{1}{R_n + sL_n} \tag{18.16}$$

Comparing Eq. 18.16 with the middle terms of Eq. 18.15, we have $R_n = \sigma_n/P_n$ and $L_n = 1/P_n$
where n refers to the nth term of Eq. 18.15, i.e. $P_n/(s + \sigma_n)$,
Consider an admittance function

$$Y(s) = \frac{2s^2 + 16s + 30}{s^2 + 6s + 8}$$

The poles and zeros are positive, real and simple. The poles are at -2, -4, and the zeros are at -3, and -5. For the second Foster form of realisation by partial fraction expansion,

$$Y(s) = 2 + \frac{4s+14}{s^2+6s+8}$$

$$= 2 + \frac{A}{s+2} + \frac{B}{s+4}$$

where $$A = \left.\frac{4s+14}{(s+4)}\right|_{s=-2} = 3$$

$$B = \left.\frac{4s+14}{s+2}\right|_{s=-4} = 1$$

The residues are positive. Hence

$$Y(s) = 2 + \frac{3}{s+2} + \frac{1}{s+4}$$

Comparing with Eq. 18.15, we have $R_\infty = 2$, $R_1 = 2/3\ \Omega$, $L_1 = 1/3$ H and $R_2 = 4\,\Omega$, $L_2 = 1\,$H.

The second Foster form of the *RL* admittance function with various values is shown in Fig. 18.12.

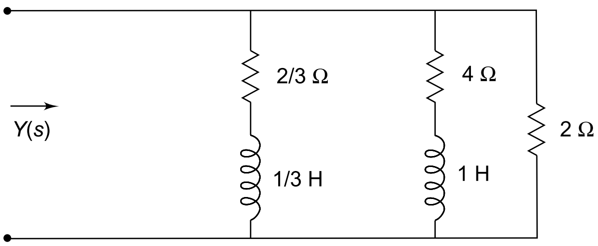


Fig. 18.12

18.7 SYNTHESIS OF *R-L* NETWORK BY CAUER METHOD

To synthesise the *RL* network, the basic step to know is that the impedance function at infinity is always greater than the impedance function at zero. Similarly, the admittance function at zero is always greater than the admittance function at infinity. In case of *RL* network synthesis, we remove the minimum real part from the function $Z(s)$. If the minimum real part is Re $[Z(j\omega)] = Z(0)$, by removing $Z(0)$ from $Z(s)$, the remainder will have a zero at $s = 0$. After inverting the remaining function, we can remove the pole at $s = 0$. By carrying on this process, we obtain a continued fraction expansion. The first form of continued fraction expansion is called the first Cauer form, which is

$$Z(s) = sL_1 + \cfrac{1}{\cfrac{1}{R_1} + \cfrac{1}{sL_2 + \cfrac{1}{\cfrac{1}{R_2} + \cdots}}}$$

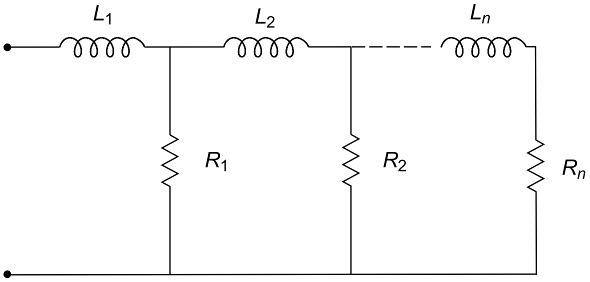


Fig. 18.13

The Cauer network for realising the above function is shown in Fig. 18.13.

In the network shown above, if $Z(s)$ has a pole at $s = \infty$, the first element is L_1. If $Z(s)$ is a constant at $s = \infty$, the first element is R_1. If $Z(s)$ has a zero at $s = 0$, the last element is L_n. If $Z(s)$ is a constant at $s = 0$, the last element is R_n.

The second form of continued fraction expansion is

$$Z(s) = R_1 + \cfrac{1}{\cfrac{1}{sL_1} + \cfrac{1}{R_2 + \cfrac{1}{\cfrac{1}{sL_2} + \cfrac{1}{R_3 + \cdots}}}}$$

The second Cauer form of the network for the above function $Z(s)$ is shown in Fig. 18.14.

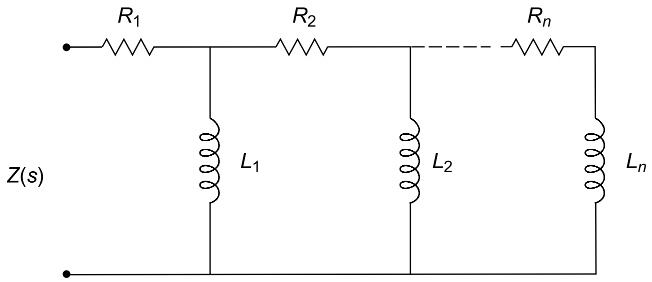


Fig. 18.14

Here also the presence of the first and the last element depends on the characteristics of impedance function, $Z(s)$. If $Z(s)$ has a zero at $s = 0$, the first element is L_1. If $Z(s)$ is a constant at $s = 0$, the first element is R_1. If $Z(s)$ has a pole at $s = \infty$, the last element is L_n. If $Z(s)$ is a constant at $s = \infty$, the last element is R_n.

The first form of the Cauer network can be obtained by continued fraction expansion and arranging the numerator and denominator polynomials of $Z(s)$ in descending powers of s. The second form of the Cauer network can be obtained by continued fraction expansion and arranging the numerator and denominator polynomials of $Z(s)$ in ascending powers of s. Consider a function

$$Z(s) = \frac{(s+4)(s+8)}{(s+2)(s+6)}$$

To find out the first Cauer form, let us take the continued fraction expansion of $Z(s)$.

$$\begin{array}{l}
s^2+8s+12)\, s^2+12s+32\ (1 \\
\qquad\qquad\quad \underline{s^2+8s+12} \\
\qquad 4s+20)\, s^2+8s+12\ (\frac{s}{4} \\
\qquad\qquad\quad \underline{s^2+5s} \\
\qquad 3s+12)\, 4s+20\ (\frac{4}{3} \\
\qquad\qquad\quad \underline{4s+16} \\
\qquad\qquad 4)\, 3s+12\ (\frac{3}{4}s \\
\qquad\qquad\quad \underline{3s} \\
\qquad\qquad\quad 12)\, 4\ (\frac{1}{3} \\
\qquad\qquad\qquad\ \ \underline{4} \\
\qquad\qquad\qquad\ \ 0
\end{array}$$

$$Z(s) = 1 + \cfrac{1}{\cfrac{s}{4} + \cfrac{1}{\cfrac{4}{3} + \cfrac{1}{\cfrac{3}{4}s + \cfrac{1}{\cfrac{1}{3}}}}}$$

Therefore, the impedance function $Z(s)$ can be realised as an RL network as shown in Fig. 18.15.

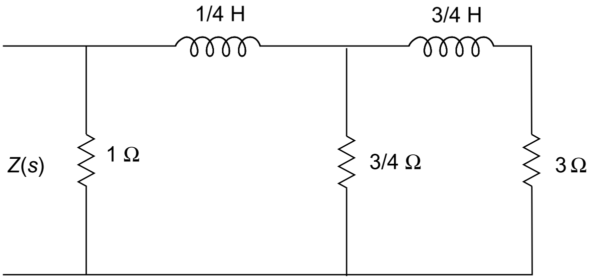


Fig. 18.15

Similarly, consider another function

$$Z(s) = \frac{2s^2 + 8s + 6}{s^2 + 8s + 12}$$

To find out the second Cauer network, we have to write the impedance function in ascending powers. By taking the continued fraction expansion of $Z(s)$, we have

$$12 + 8s + s^2\,)\,6 + 8s + 2s^2\,(\frac{1}{2}$$

$$\underline{6 + 4s + \frac{1}{2}s^2}$$

$$4s + \frac{3}{2}s^2\,)\,12 + 8s + s^2\,(\frac{3}{s}$$

$$\underline{12 + \frac{9}{2}s}$$

$$\frac{7}{2}s + s^2\,)\,4s + \frac{3}{2}s^2\,(\frac{8}{7}$$

$$\underline{4s + \frac{8}{7}s^2}$$

$$\frac{5}{14}s^2\,)\,\frac{7}{2}s + s^2\,(\frac{49}{5s}$$

$$\underline{\frac{7}{2}s}$$

$$s^2\,)\,\frac{5}{14}s^2\,(\frac{5}{14}$$

$$\underline{\frac{5}{14}s^2}$$

$$0$$

$$Z(s) = \frac{1}{2} + \cfrac{1}{\cfrac{3}{s} + \cfrac{1}{\cfrac{8}{7} + \cfrac{1}{\cfrac{49}{5s} + \cfrac{1}{\cfrac{5}{14}}}}}$$

Therefore, the impedance function $Z(s)$ can be realised as an RL network shown in Fig. 18.16.

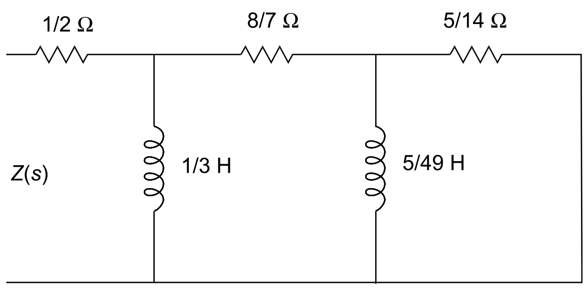


Fig. 18.16

18.8 SYNTHESIS OF RC NETWORK BY FOSTER METHOD

The driving point impedance RC network, $Z(s)$ is given by

$$Z(s)=\frac{H(s+\sigma_1)(s+\sigma_3)\cdots}{s(s+\sigma_2)(s+\sigma_4)\cdots} \tag{18.17}$$

The first form of the RC Foster network is shown in Fig. 18.17.

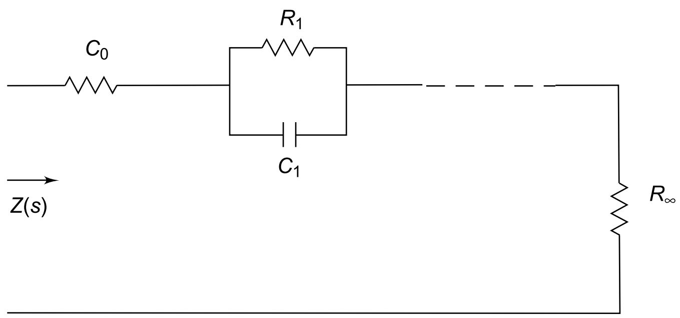


Fig. 18.17

Here, the RC impedance possesses the same properties as the RL admittance function. To synthesise the first Foster form of the RC network, we shall write the expression for the RC parallel combination in the network of Fig. 18.17.

$$Z_1(s)=\frac{\frac{1}{C_1}}{s+\frac{1}{R_1C_1}} \tag{18.18}$$

where $\sigma_1=\dfrac{1}{R_1C_1}\quad P_1=\dfrac{1}{C_1}$

We have the other form of the impedance function

$$Z(s) = \frac{a_0 s^n + a_1 s^{n-1} + \cdots + a_n}{b_0 s^m + b_1 s^{m-1} + \cdots + b_n s} \tag{18.19}$$

Obviously, the degree in s of the numerator polynomial is greater than that of the denominator polynomial by one. The roots of the polynomials are real and negative.

At $s = \infty$, $\quad Z(s) = \frac{a_0}{b_0} = R_\infty, \quad \text{when } a_0 \neq 0$

$\qquad\qquad = 0, \quad \text{when } a_0 = 0$

The total impedance can be written as the combination of impedances $Z_1(s)$, $Z_2(s), \cdots, Z_n(s)$

$$Z(s) = Z_1(s) + Z_2(s) + \cdots + Z_n(s) \tag{18.20}$$

From Fig. 18.18, we have the impedance

$$Z(s) = \frac{P_0}{s} + \frac{P_i}{s + \sigma_i} + \cdots + \text{H} \tag{18.21}$$

By comparing Eqs 18.20 and 18.21, we have the impedance $Z_1(s) = P_0/s$ representing a capacitance term $1/P_0$, and the impedance $Z_n(s) = \text{H}$, a constant term representing resistor R_∞. The remaining terms represent a parallel combination of a capacitor and resistor. By comparing Eq. 18.18 with the middle terms of Eq. 18.21, we have

$$P_n = \frac{1}{C_n}$$

and $\quad \sigma_n = \frac{1}{R_n C_n}$

where n refers to the term $P_n/(s + \sigma_n)$ in Eq. 18.21.

Similarly, the driving point function of an RC network $Y(s)$ is given by

$$Y(s) = \frac{b_0 s^m + b_1 s^{m-1} + \cdots + b_m s}{a_0 s^n + a_1 s^{n-1} + \cdots + a_n} \tag{18.22}$$

The second form of the Foster network is shown in Fig. 18.18.

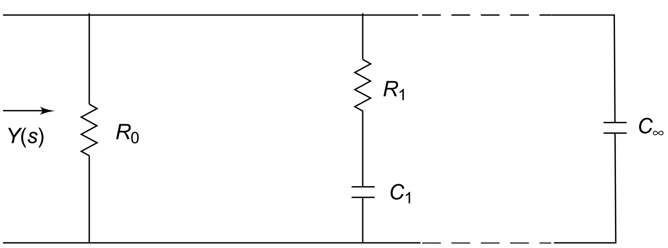


Fig. 18.18

The RC admittance function possesses the same properties as the RL impedance function. By taking the partial fraction expansion of Eq. 18.22 we can write the RC admittance function as

$$Y(s) = P_0 + \frac{P_i s}{s + \sigma_i} + \cdots + \mathrm{H}s \tag{18.23}$$

If we observe Eq. 18.23, we have the first term P_0 representing resistance $R_0 = 1/P_0$, and the last term represents capacitance $C_\infty = \mathrm{H}$ and the intermediate terms representing admittance function of series RC network.

$$Y_n = \frac{1}{R_n + \frac{1}{sC_n}} \tag{18.24}$$

Comparing Eq. 18.24 and the middle terms of Eq. 18.23, we have

$$R_n = \frac{1}{P_n} \text{ and } C_n = \frac{1}{\sigma_n R_n}$$

Consider a function $Z(s) = \dfrac{3(s+2)(s+4)}{(s+1)(s+3)}$

The first Foster form can be realised by taking the partial fraction of $Z(s)$

$$Z(s) = 3 + \frac{6s+15}{s^2+4s+3}$$

$$= 3 + \frac{6s+15}{(s+1)(s+3)} = 3 + \frac{A}{s+1} + \frac{B}{s+3}$$

where $\quad A = \left.\dfrac{6s+15}{s+3}\right|_{s=-1} = \dfrac{9}{2}$

$$B = \left.\frac{6s+15}{s+1}\right|_{s=-3} = \frac{3}{2}$$

The residues are positive, and hence

$$Z(s) = 3 + \frac{9/2}{s+1} + \frac{3/2}{s+3}$$

Comparing with Eq. 18.21, we have

$$R_\infty = 3, R_1 = \frac{9}{2}, \quad C_1 = \frac{2}{9}\,\mathrm{F}$$

and $\quad R_2 = \dfrac{1}{2}, \; C_2 = \dfrac{2}{3}\,\mathrm{F}$

The network with elemental values is shown in Fig. 18.19.

The second Foster form can be realised by taking the reciprocal of the impedance function and partial fraction expansion as

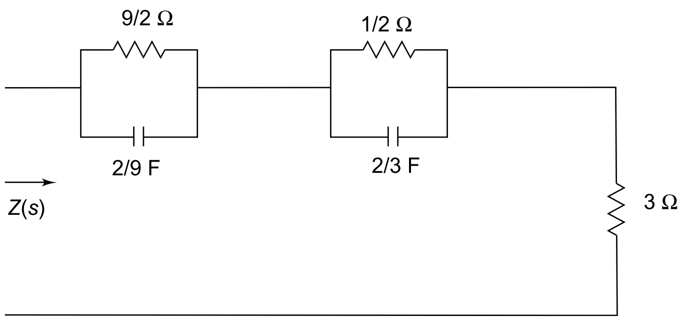


Fig. 18.19

$$Y(s) = \frac{s^2 + 4s + 3}{3s^2 + 18s + 24} = \left(\frac{s^2 + 4s + 3}{3s^2 + 18s + 24} - \frac{1}{8}\right) + \frac{1}{8}$$

$$Y(s) = \frac{s(5s + 14)}{8(3)(s+2)(s+4)} + \frac{1}{8}$$

$$\frac{Y(s)}{s} = \frac{5s + 14}{24(s+2)(s+4)} + \frac{1}{8s} = \frac{1}{8s} + \frac{A}{s+2} + \frac{B}{s+4}$$

$$A = \left.\frac{5s + 14}{24(s+4)}\right|_{s=-2} = \frac{1}{12}$$

$$B = \left.\frac{5s + 14}{24(s+2)}\right|_{s=-4} = \frac{1}{8}$$

$$\therefore \qquad Y(s) = \frac{1}{8} + \frac{(1/12)s}{s+2} + \frac{(1/8)s}{s+4}$$

The network with elemental values is shown in Fig. 18.20.

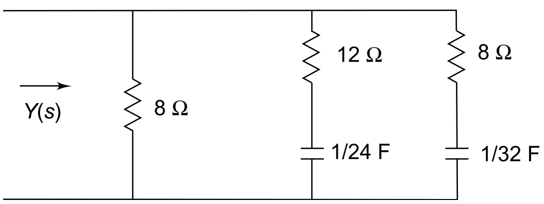


Fig. 18.20

18.9 SYNTHESIS OF R-C NETWORK BY CAUER METHOD

To synthesise the RC network function, the basic step to know is that the impedance function at zero is always greater than the impedance function at infinity. Similarly, the admittance function at infinite is always greater than the admittance function at zero.

To synthesise an RC network, we remove the minimum real part from the function, $Z(s)$. If the minimum real part is Re $[Z(j\omega)] = Z(\infty)$, by removing $Z(\infty)$ from $Z(s)$, the remainder will have a zero at $s = \infty$. After inverting the remaining function, we can remove a pole at $s = \infty$. By carrying on this process, we obtain a continued fraction expansion. The first form of continued fraction expansion is called the first Cauer form, and is given by

$$Z(s) = R_1 + \cfrac{1}{C_1 s + \cfrac{1}{R_2 + \cfrac{1}{C_2 s + \cdots}}}$$

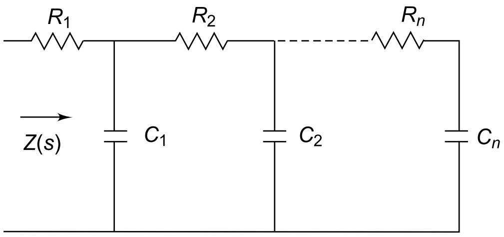


Fig. 18.21

The Cauer network for realising the above function is shown in Fig. 18.21.

In the network shown, if $Z(s)$ has a zero at $s = \infty$, the first element is C_1. If $Z(s)$ is a constant at $s = \infty$, the first element is R_1. If $Z(s)$ has a pole at $s = 0$, the last element is C_n. If $Z(s)$ is constant at $s = 0$, the last element is R_n.

The second form of continued fraction expansion is

$$Z(s) = \frac{1}{C_1 s} + \cfrac{1}{\cfrac{1}{R_1} + \cfrac{1}{\cfrac{1}{C_2 s} + \cfrac{1}{\cfrac{1}{R_2} + \cdots}}}$$

The second Cauer form of network for the above function $Z(s)$ is shown in Fig. 18.22.

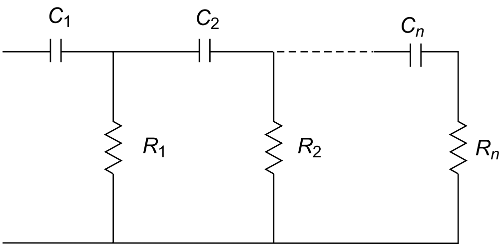


Fig. 18.22

In the network shown in Fig. 18.22, if $Z(s)$ has a pole at $s = 0$, the first element is C_1. If $Z(s)$ is a constant at $s = 0$, the first element is R_2. If $Z(s)$ has a zero at $s = \infty$, the last element is C_n. If $Z(s)$ is constant at $s = \infty$, the last element is R_n.

Consider a function $Z(s) = (s + 2)(s + 4)/s(s + 3)$. To find the first Cauer form, we take the continued fraction expansion by the divide, invert, divide procedure as follows.

$$s^2+3s)\,s^2+6s+8\,(1-R_1$$
$$\underline{s^2+3s}$$
$$3s+8)\,s^2+3s\,(\frac{s}{3}-C_1 s$$
$$\underline{s^2+\frac{8s}{3}}$$
$$\frac{s}{3})\,3s+8\,(9-R_2$$
$$\underline{3s}$$
$$8)\,\frac{s}{3}\,(\frac{s}{24}-C_2 s$$
$$\underline{\frac{s}{3}}$$
$$0$$

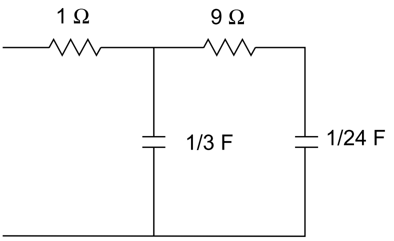


Fig. 18.23

$$Z(s)=1+\cfrac{1}{\cfrac{s}{3}+\cfrac{1}{9+\cfrac{1}{\cfrac{s}{24}}}}$$

Therefore, the impedance function $Z(s)$ can be realised as an RC network shown in Fig. 18.23.

Similarly, the second Cauer network can be obtained by arranging the numerator and denominator polynomials of $Z(s)$ in ascending powers of s. The continued fraction expansion is

$$3s+s^2)\,8+6s+s^2\,(\frac{8}{3s}$$
$$\underline{8+\frac{8s}{3}}$$
$$\frac{10s}{3}+s^2)\,3s+s^2\,(\frac{9}{10}$$
$$\underline{3s+\frac{9s^2}{10}}$$
$$\frac{s^2}{10})\,\frac{10s}{3}+s^2\,(\frac{100}{3s}$$
$$\underline{\frac{10s}{3}}$$
$$s^2)\,\frac{s^2}{10}\,(\frac{1}{10}$$
$$\underline{\frac{s^2}{10}}$$
$$0$$

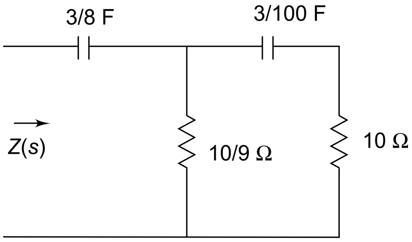


Fig. 18.24

$$Z(s) = \frac{8}{3s} + \cfrac{1}{\cfrac{9}{10} + \cfrac{1}{\cfrac{100}{3s} + \cfrac{1}{\cfrac{1}{10}}}}$$

Therefore, the impedance function $Z(s)$ can be realised as an RC network shown in Fig. 18.24.

Additional Solved Problems

Problem 18.1 *Find the two Foster realisations of the given function.*

$$Z(s) = \frac{2s^3 + 8s}{s^2 + 1}$$

Solution For the first Foster network, we expand $Z(s)$ into partial fractions.

$$Z(s) = 2s + \frac{6s}{s^2 + 1}$$

$$= 2s + \frac{A}{s + j} + \frac{A^*}{s - j}$$

By applying Heaviside method, we get

$$A = \frac{6s}{(s + j)(s - j)}(s + j)\bigg|_{s=-j} = 3$$

$$A^* = \frac{6s}{(s + j)(s - j)}(s - j)\bigg|_{s=j} = 3$$

By inspection, $H = 2$

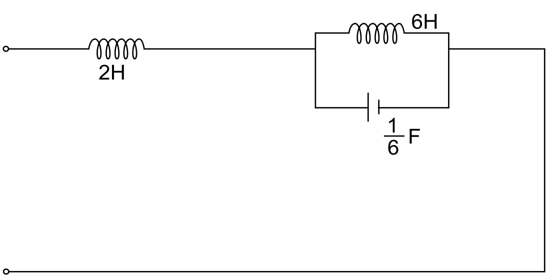


Fig. 18.25

$$\therefore \quad Z(s) = 2s + \frac{3}{s + j} + \frac{3}{s - j}$$

$$\therefore \quad L_2 = \frac{2A}{\omega_n^2} = 6;$$

$$C_2 = \frac{1}{2A} = \frac{1}{6}$$

The First Foster network with elemental values is shown in Fig. 18.25.

Second Foster network can be obtained by taking admittance function

$$Y(s) = \frac{s^2+1}{2s(s+4)}$$

By taking partial fractions, we have

$$Y(s) = \frac{A}{s} + \frac{B}{s+j2} + \frac{B^*}{s-j2}$$

By applying Heaviside method, we get

$$A = \frac{s^2+1}{2s(s+4)} s \bigg|_{s=0} = \frac{1}{8}$$

$$B = \frac{s^2+1}{2s(s+4)} (s+j2) \bigg|_{s=-j2} = \frac{3}{16}$$

$$B^* = \frac{s^2+1}{2s(s+4)} (s-j2) \bigg|_{s=j2} = \frac{3}{16}$$

Therefore, the elemental values are

$$L_0 = \frac{1}{A} = 8\text{ H}$$

$$L_1 = \frac{1}{2B} = \frac{8}{3}\text{ H}$$

$$C_1 = \frac{2B}{\omega_n^2} = \frac{3}{32}\text{ F}$$

The second Foster network with elemental values is shown in Fig. 18.26.

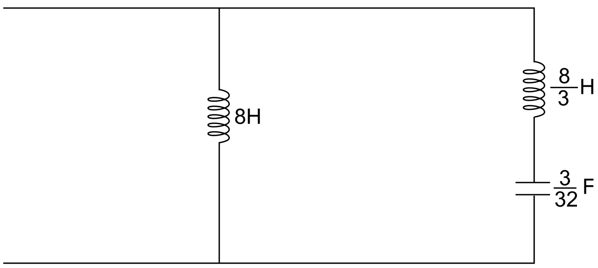


Fig. 18.26

Problem 18.2 *Find the two Foster realisations of the given function.*

$$z(s) = \frac{3(s^2+1)(s^2+16)}{s(s^2+9)}$$

Solution For the First Foster network, we expand $Z(s)$ into partial fractions

$$Z(s) = 3s + \frac{24(s^2+2)}{s(s^2+9)}$$
$$= 3s + \frac{A}{s} + \frac{B}{s+j3} + \frac{B^*}{s-j3}$$

By applying Heaviside method,

$$A = \frac{24(s^2+2)}{s(s^2+9)} s \Bigg|_{s=0} = \frac{16}{3}$$

$$B = \frac{24(s^2+2)}{s(s^2+9)} (s+j3) \Bigg|_{s=-j3} = \frac{28}{3} = B^*$$

$$\therefore \quad Z(s) = 3s + \frac{16}{3s} + \frac{28}{3(s+j3)} + \frac{28}{3(s-j3)}$$

By inspection $H = L_\infty = 3$

$$A = \frac{16}{3} \text{ and } C_0 = \frac{1}{A} = \frac{3}{16} \text{ F}$$

$$C_2 = \frac{1}{2B} = \frac{3}{56} \text{ F} ; L_2 = \frac{2B}{\omega_n^2} = \frac{56}{27}$$

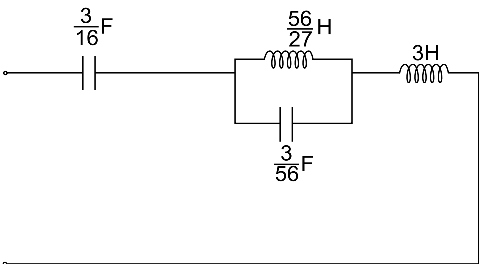


Fig. 18.27

The first Foster network with elemental values is shown in Fig. 18.27.

Second Foster network can be obtained by taking admittance function

$$Y(s) = \frac{s(s^2+9)}{s(s^2+1)(s^2+16)}$$

By taking partial fraction expansion, we have

$$Y(s) = \frac{2AS}{s^2+1} + \frac{2BS}{s^2+16}$$

By applying Heaviside method, we get

$$A = \frac{8}{90} ; B = \frac{7}{90}$$

Therefore, the elemental values are

$$L_1 = \frac{1}{2A} = \frac{90}{16} \text{ H} ; C_1 = \frac{2A}{\omega_1^2} = \frac{8}{45} \text{ F}$$

$$L_2 = \frac{1}{2B} = \frac{90}{14} \text{ H} ; C_2 = \frac{2B}{\omega_1^2} = \frac{7}{720} \text{ F}$$

The second Foster network with elemental values is shown in Fig. 18.28.

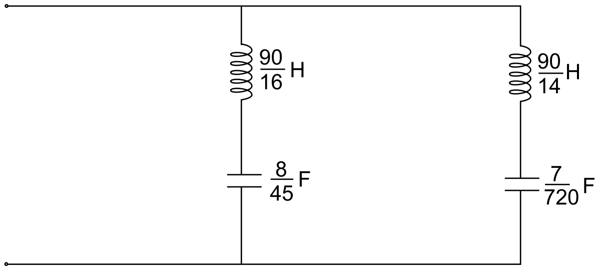


Fig. 18.28

Problem 18.3 *Find the second cauer network of the given function.*

$$z(s) = \frac{s^4 + 6s^2 + 4}{s^3 + 2s}$$

Solution The second cauer network can be realised by arranging the numerator and denominator polynomials of $Z(s)$ in ascending power of s and taking continued fraction expansion, we get

$$2s + s^3)\; 4 + 6s^2 + s^4 \;(\frac{2}{s}$$
$$\underline{4 + 2s^2}$$
$$4s^2 + s^4)\; 2s + s^3 \;(\frac{1}{2s}$$
$$\underline{2s + \frac{s^3}{2}}$$
$$\frac{s^3}{2})\; 4s^2 + s^4 \;(\frac{8}{s}$$
$$\underline{4s^2}$$
$$s^4)\; \frac{s^3}{2} \;(\frac{1}{2s}$$
$$\underline{\frac{s^3}{2}}$$
$$0$$

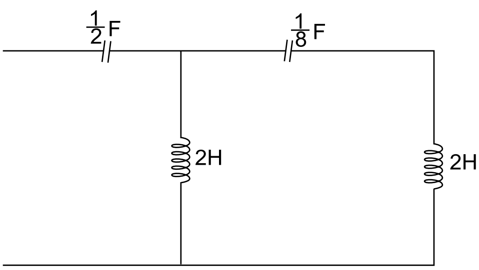


Fig. 18.29

$$Z(s) = \frac{2}{s} + \cfrac{1}{\cfrac{1}{2s} + \cfrac{1}{\cfrac{8}{s} + \cfrac{1}{1/2s}}}$$

Therefore, the impedance function, $Z(s)$, can be realised as the RC network shown in Fig. 18.29.

Problem 18.4 *Find the first and second Cauer forms of the function.*

$$z(s) = \frac{2s^2 + 8s + 6}{s^2 + 2s}$$

Solution The first cauer network can be realised by taking continued fraction expansion

$$s^2 + 2s)\ 2s^2 + 8s + 6\ (2$$
$$\underline{2s^2 + 4s}$$
$$4s + 6)\ s^2 + 2s\ (\frac{s}{4}$$
$$\underline{s^2 + \frac{6s}{4}}$$
$$\frac{s}{2})\ 4s + 6\ (8$$
$$\underline{4s}$$
$$6)\ s/2\ (s/12$$
$$\underline{s/2}$$
$$0$$

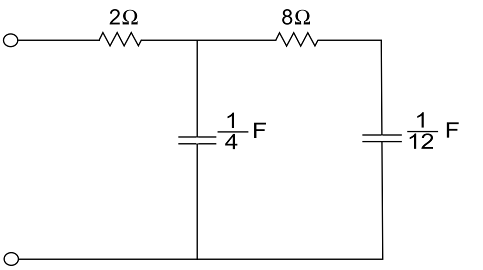


Fig. 18.30

$$Z(s) = 2 + \cfrac{1}{\frac{s}{4} + \cfrac{1}{8 + \cfrac{1}{\frac{s}{12}}}}$$

Therefore, the impedance function $Z(s)$ can be realised as RC network shown in Fig. 18.30.

The second Cauer network can be realised by arranging the numerator and denominator polynomials of $Z(s)$ in ascending power of s and taking continued fraction expansion, we get

$$2s + s^2)\ 6 + 8s + 2s^2\ (\frac{3}{s}$$
$$\underline{6 + 3s}$$
$$5s + 2s^2)\ 2s + s^2\ (\frac{2}{5}$$
$$\underline{2s + \frac{4}{5}s^2}$$
$$\frac{s^2}{5})\ 5s + 2s^2\ (\frac{25}{s}$$
$$\underline{5s}$$
$$2s^2)\ \frac{s^2}{5}\ (\frac{1}{10}$$
$$\underline{\frac{s^2}{5}}$$
$$0$$

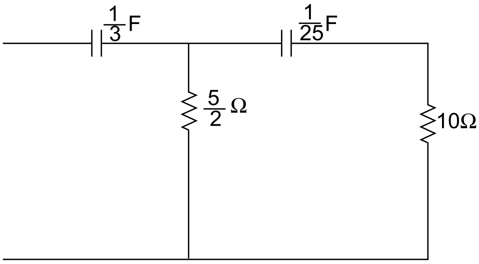


Fig. 18.31

$$Z(s)=\frac{3}{s}+\cfrac{1}{\cfrac{2}{5}+\cfrac{1}{\cfrac{25}{s}+\cfrac{1}{10}}}$$

Therefore, the impedance function, $Z(s)$, can be realised as the RC network shown in Fig. 18.31.

Problem 18.5 *Find the second Foster form and the first Cauer form of the network whose driving point admittance is*

$$Y(s)=\frac{3(s+2)(s+5)}{s(s+3)}$$

Solution By taking partial fraction expansion, we get

$$Y(s)=3+\frac{9s+24}{s^2+3s}=3+\frac{A}{s}+\frac{B}{s+3}$$

By applying Heaviside method, we get

$$A=\frac{9s+24}{s(s+3)}s\bigg|_{s=0}=8$$

$$B=\frac{9s+24}{s^2+3s}(s+3)\bigg|_{s=-3}=1$$

$$\therefore\quad Y(s)=3+\frac{8}{s}+\frac{1}{s+3}$$

Therefore, the elemental values are

$$R=\frac{1}{3}\,\Omega,\quad L=\frac{1}{8}\,\text{H};\ R_1=3\,\Omega;\ L_1=1\,\text{H}$$

Therefore, the second Foster network is shown in Fig. 18.32.

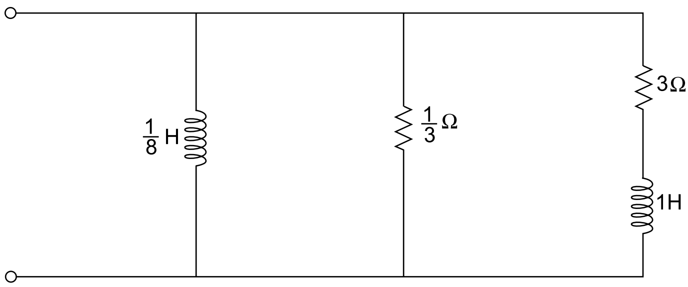


Fig. 18.32

To get the first cauer realisation, we take continued fraction expansion from the expression.

$$Y(s) = \frac{3(s+2)(s+4)}{s(s+3)}$$

$$s^2+3s)\,3s^2+18s+24\,(3$$
$$\underline{3s^2+9s}$$
$$9s+24)\,s^2+3s\,\left(\frac{s}{9}\right.$$
$$\underline{s^2+\frac{8}{3}s}$$
$$\frac{s}{3}\Big)\,9s+24\,(27$$
$$\underline{9s}$$
$$24)\,\frac{s}{3}\left(\frac{s}{72}\right.$$
$$\underline{\frac{s}{3}}$$
$$0$$

$$\therefore \qquad Y(s) = \cfrac{1}{3+\cfrac{1}{\cfrac{s}{9}+\cfrac{1}{27+\cfrac{1}{\cfrac{s}{72}}}}}$$

Therefore, the admittance, $Y(s)$, can be realised as RL network shown in Fig. 18.33.

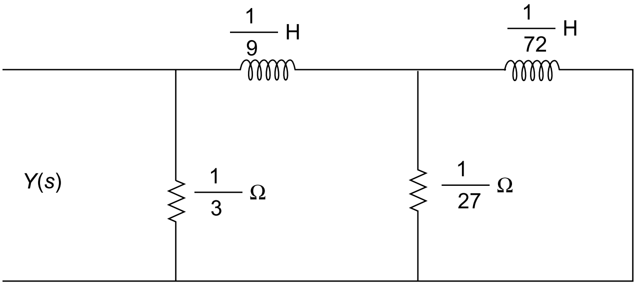


Fig. 18.33

Problem 18.6 *Find the two Foster realisation of*

$$Z(s) = \frac{4(s^2+1)(s^2+16)}{s(s^2+4)}$$

Solution For the first Foster network, we expand $Z(s)$ into partial fractions.

$$Z(s) = \frac{P_0}{s} + \frac{P_2}{s+j2} + \frac{P_2^*}{s-j2} + Hs$$

By applying Heaviside method, from the above equation we have

$$P_0 = \left.\frac{4(s^2+1)(s^2+16)}{s^2+4}\right|_{s=0} = 16$$

$$P_2 = \left.\frac{4(s^2+1)(s^2+16)}{s(s-j2)}\right|_{s=-j2} = -\frac{4\times 3\times 12}{-j4\times -j2} = 18$$

By inspection, $H = 4$

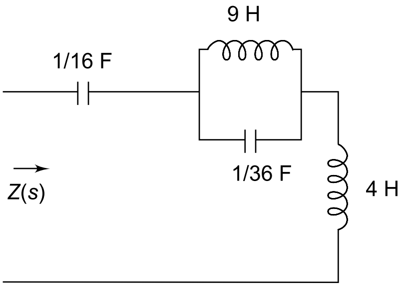


Fig. 18.34 (a)

$$\therefore \quad C_0 = \frac{1}{P_0} = \frac{1}{16}\text{F}; L_\infty = H = 4\text{ H}$$

$$C_2 = \frac{1}{2P_2} = \frac{1}{36}\text{F}$$

$$L_2 = \frac{2P_2}{\omega_n^2} = \frac{2\times 18}{4} = 9\text{H}$$

The first Foster network with elemental values is shown in Fig. 18.34 (a).

Second Foster network can be obtained by taking admittance function

$$Y(s) = \frac{s(s^2+4)}{4(s^2+1)(s^2+16)}$$

Let us take the partial fraction expansion, we have

$$Y(s) = \frac{2P_1 s}{s^2+1} + \frac{2P_2 s}{s^2+16}$$

By applying Heaviside method, we get

$$P_1 = \left.\frac{1}{4}\frac{s(s^2+4)}{(s^2+16)(s+j1)}\right|_{s=-j1} = \frac{1}{4}\frac{(-j1)(3)}{(15)(-j2)} = \frac{1}{40}$$

$$P_2 = \left.\frac{1}{4}\frac{s(s^2+4)}{(s-j4)(s^2+1)}\right|_{s=-j4} = \frac{1}{4}\frac{(-j4)(-12)}{(-j8)(-15)} = \frac{1}{10}$$

Therefore, the element values are

$$L_1 = \frac{1}{2P_1} = 20\text{H}$$

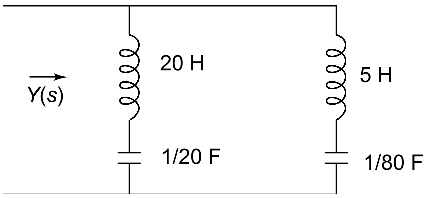


Fig. 18.34 (b)

$$C_1 = \frac{2P_1}{\omega_1^2} = \frac{2\times 1}{40\times 1} = \frac{1}{20}\ \text{F}$$

$$L_2 = \frac{1}{2P_2} = \frac{10}{2} = 5\,\text{H}$$

$$C_2 = \frac{2P_2}{\omega_1^2} = \frac{2}{10\times 16} = \frac{1}{80}\ \text{F}$$

The second Foster network with elemental values is shown in Fig. 18.34 (b).

Problem 18.7 *Find the two Cauer realisations of driving point function given by*

$$Z(s) = \frac{10s^4 + 12s^2 + 1}{2s^3 + 2s}$$

Solution By taking the continued fraction expansion, we get

$$\begin{array}{l}
2s^3 + 2s)\,10s^4 + 12s^2 + 1\,(5s \\
\qquad\qquad \underline{10s^4 + 10s^2} \\
\qquad\qquad\qquad 2s^2 + 1)\,2s^3 + 2s\,(s \\
\qquad\qquad\qquad\qquad\quad \underline{2s^3 + s} \\
\qquad\qquad\qquad\qquad\qquad s)\,2s^2 + 1\,(2s \\
\qquad\qquad\qquad\qquad\qquad\quad \underline{2s^2} \\
\qquad\qquad\qquad\qquad\qquad\qquad 1)\,s\,(s \\
\qquad\qquad\qquad\qquad\qquad\qquad\quad \underline{s} \\
\qquad\qquad\qquad\qquad\qquad\qquad\quad 0
\end{array}$$

Hence $$Z(s) = 5s + \cfrac{1}{s + \cfrac{1}{2s + \cfrac{1}{s}}}$$

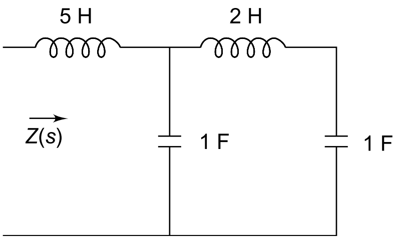


Fig. 18.35 (a)

The resulting network is called the first Cauer form with elemental values shown in Fig. 18.35 (a).

To realise the second Cauer network, we have to take ascending powers of impedance function.

Continued fraction expansion gives

$$
\begin{array}{l}
2s+2s^3)\,1+12s^2+10s^4\,(\frac{1}{2s} \\
\quad\underline{1+s^2} \\
\quad 11s^2+10s^4)\,2s+2s^3\,(\frac{2}{11s} \\
\qquad\qquad \underline{2s+\frac{20s^3}{11}} \\
\qquad\qquad \frac{2s^3}{11})\,11s^2+10s^4\,(\frac{121}{2s} \\
\qquad\qquad\qquad \underline{11s^2} \\
\qquad\qquad\qquad\quad 10s^4)\,\frac{2}{11}s^3\,(\frac{2}{110s} \\
\qquad\qquad\qquad\qquad \underline{\frac{2}{11}s^3} \\
\qquad\qquad\qquad\qquad\quad 0
\end{array}
$$

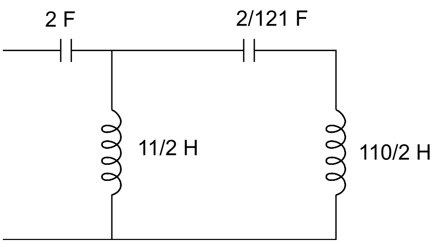


Fig. 18.35 (b)

Hence, $Z(s)=\dfrac{1}{2s}+\cfrac{1}{\cfrac{2}{11s}+\cfrac{1}{\cfrac{121}{2s}+\cfrac{1}{\cfrac{2}{110s}}}}$

The resulting network shown in Fig. 18.35 (b) is called the second Cauer form.

Problem 18.8 *Find the first Foster form of the driving point function of*

$$Z(s)=\frac{2(s+2)\,(s+5)}{(s+4)\,(s+6)}$$

Solution If we take the partial fraction of $Z(s)$, the signs of the function and its poles are negative as shown.

$$Z(s)=2-\frac{2}{s+4}-\frac{4}{s+6}$$

Therefore, we have to expand $Z(s)/s$

$$\frac{Z(s)}{s}=\frac{2(s+2)(s+5)}{s(s+4)(s+6)}$$

By taking partial fractions, we get

$$\frac{2(s+2)(s+5)}{s(s+4)(s+6)}=\frac{A}{s}+\frac{B}{s+4}+\frac{C}{s+6}$$

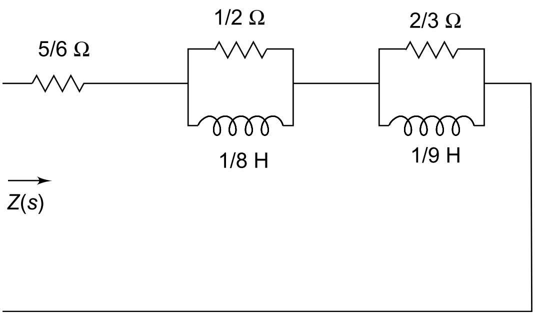


Fig. 18.36

$$= \frac{5}{6s} + \frac{1}{2(s+4)} + \frac{2}{3(s+6)}$$

If we multiply both sides by s, we get

$$Z(s) = \frac{5}{6} + \frac{s}{2(s+4)} + \frac{2s}{3(s+6)}$$

Hence, impedance $Z(s)$ can be realised as a series Foster form of RL network shown in Fig. 18.36.

Problem 18.9 *Find the second Foster form of RL network for the function.*

$$Y(s) = \frac{s^2 + 8s + 15}{s^2 + 5s + 4}$$

Solution By taking partial fraction expansion, we get

$$Y(s) = 1 + \frac{3s + 11}{s^2 + 5s + 4} = 1 + \frac{A}{(s+1)} + \frac{B}{s+4}$$

where $$A = \left.\frac{3s+11}{s+4}\right|_{s=-1} = \frac{8}{3}$$

$$B = \left.\frac{3s+11}{s+1}\right|_{s=-4} = \frac{1}{3}$$

The residues are positive. Hence

$$Y(s) = 1 + \frac{8}{3(s+1)} + \frac{1}{3(s+4)}$$

Therefore $$R = 1\Omega, R_1 = \frac{3}{8}\,\Omega, L_1 = \frac{3}{8}\text{ H}$$

The second Foster form of RL admittance function with various elemental values is shown in Fig. 18.37.

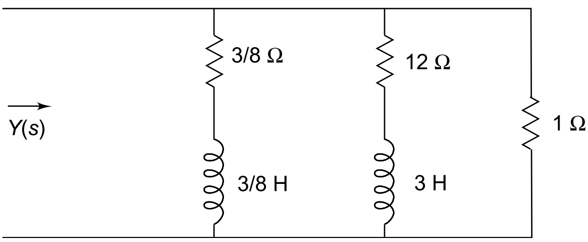


Fig. 18.37

Problem 18.10 *Find the first Cauer form of the function.*

$$Z(s) = \frac{(s+3)(s+7)}{(s+2)(s+4)}$$

Solution By taking continued fraction expansion, we get

$$s^2 + 6s + 8)\, s^2 + 10s + 21\, (1$$
$$\underline{s^2 + 6s + 8}$$
$$4s + 13)\, s^2 + 6s + 8\, (\frac{s}{4}$$
$$\underline{s^2 + \frac{13s}{4}}$$
$$\frac{11s}{4} + 8)\, 4s + 13\, (\frac{16}{11}$$
$$\underline{4s + \frac{128}{11}}$$
$$\frac{15}{11})\, \frac{11s}{4} + 8\, (\frac{121}{60}s$$
$$\underline{\frac{11s}{4}}$$
$$8)\, \frac{15}{11}\, (\frac{15}{88}$$
$$\underline{\frac{15}{11}}$$
$$0$$

$$Z(s) = 1 + \cfrac{1}{\cfrac{s}{4} + \cfrac{1}{\cfrac{1}{16/11} + \cfrac{1}{\cfrac{121}{60}s + \cfrac{1}{\cfrac{15}{88}}}}}$$

Therefore, the impedance function can be realised as RL network shown in Fig. 18.38.

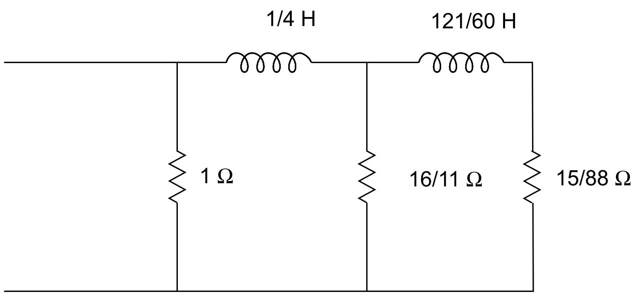


Fig. 18.38

Problem 18.11 *Find the first and second Foster forms of the function.*

$$Z(s) = \frac{(s+1)(s+3)}{s(s+2)}$$

Solution By taking the partial fraction expansion, we get

$$Z(s) = 1 + \frac{2s+3}{s(s+2)} = 1 + \frac{A}{s} + \frac{B}{s+2}$$

$$Z(s) = 1 + \frac{3}{2s} + \frac{1}{2(s+2)}$$

Hence, the impedance function $Z(s)$ can be realised as series Foster form of RC network shown in Fig. 18.39 (a).

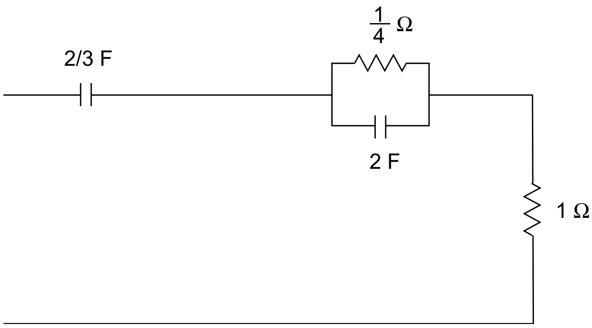


Fig. 18.39 (a)

The second Foster form can be realised by taking $Y(s)$ as under.

$$Y(s) = \frac{s(s+2)}{(s+1)(s+3)}$$

$$= 1 - \frac{2s+3}{(s+1)(s+3)} = 1 - \frac{1}{2(s+1)} - \frac{3}{2(s+3)}$$

Since negative quotients appear, we have to expand $Y(s)/s$ as follows

$$\frac{Y(s)}{s} = \frac{(s+2)}{(s+1)(s+3)}$$

$$= \frac{A}{s+1} + \frac{B}{s+3} = \frac{1/2}{s+2} + \frac{1/2}{2(s+3)}$$

Multiplying both sides by s, we get

$$Y(s) = \frac{s/2}{s+1} + \frac{s/2}{s+3}$$

The network with elemental values are shown in Fig. 18.39 (b).

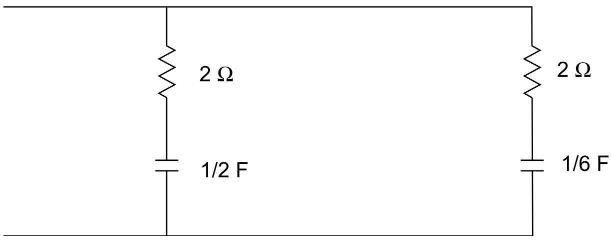


Fig. 18.39 (b)

Problem 18.12 *Find the first and second Cauer forms of the given function.*

$$Z(s) = \frac{(s+1)(s+3)}{s(s+2)}$$

Solution The first Cauer network can be realised by taking continued fraction expansion.

$$s^2 + 2s)\, s^2 + 4s + 3\, (1$$
$$\underline{s^2 + 2s}$$
$$2s + 3)\, s^2 + 2s\, (\frac{s}{2}$$
$$\underline{s^2 + \frac{3s}{2}}$$
$$\frac{s}{2})\, 2s + 3\, (4$$
$$\underline{2s}$$
$$3)\, \frac{s}{2}\, (\frac{1}{6}s$$
$$\underline{\frac{s}{2}}$$
$$0$$

$$Z(s) = 1 + \cfrac{1}{\cfrac{s}{2} + \cfrac{1}{4 + \cfrac{1}{\cfrac{s}{6}}}}$$

Therefore, the impedance function, $Z(s)$, can be realised as RC network shown in Fig. 18.40 (a).

The second Cauer network can be realised by arranging the numerator and denominator polynomials of $Z(s)$ in ascending power of s and taking continued fraction expansion; we get

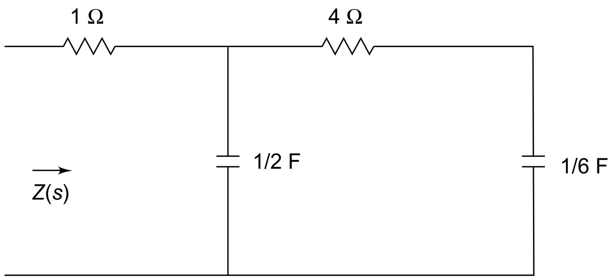


Fig. 18.40 (a)

$$\begin{array}{l}
2s+s^2\,)\,3+4s+s^2\,\left(\frac{3}{2s}\right. \\
\qquad 3+\frac{3}{2}s \\
\qquad \overline{\qquad\quad \frac{5s}{2}+s^2\,)\,2s+s^2\,\left(\frac{4}{5}\right.} \\
\qquad\qquad\qquad 2s+\frac{4}{5}s^2 \\
\qquad\qquad\qquad \overline{\frac{s^2}{5})\,\frac{5s}{2}+s^2\,\left(\frac{25}{2s}\right.} \\
\qquad\qquad\qquad\quad \frac{5s}{2} \\
\qquad\qquad\qquad\quad \overline{\qquad\quad s^2\,)\,\frac{s^2}{5}\,\left(\frac{1}{5}\right.} \\
\qquad\qquad\qquad\qquad\qquad \frac{s^2}{5} \\
\qquad\qquad\qquad\qquad\qquad \overline{\;0\;}
\end{array}$$

$$Z(s)=\frac{3}{2s}+\cfrac{1}{\frac{4}{5}+\cfrac{1}{\frac{25}{2s}+\cfrac{1}{\frac{1}{5}}}}$$

Therefore, the impedance function, $Z(s)$, can be realised as the RC network shown in Fig. 18.40 (b).

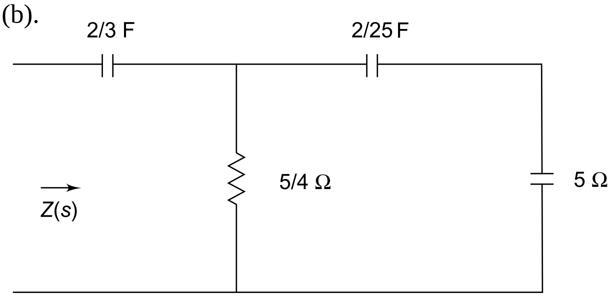


Fig. 18.40 (b)

Practice Problems

18.1 Test whether the following polynomials are Hurwitz.

(a) $P(s) = s^3 + 2s^2 + 4s + 2$

(b) $P(s) = s^4 + s^3 + 4s^2 + 2s + 3$

(c) $P(s) = s^4 + 2s^3 + 2s^2 + 6s + 10$

18.2 Check the positive realness of the following functions.

(a) $\dfrac{(2s+4)}{s+5}$

(b) $\dfrac{s^2+2s+4}{(s+3)(s+1)}$

(c) $(s^2 + 2s)/(s^2 + 1)$

18.3 Investigate if the following partially factored driving point impedance function is a minimum positive real function.

$$Z(s) = \frac{2s^4+3s^3+5s^2+5s+1}{(s^2+1)(2s^2+2s+1)}$$

18.4 Find the two canonical Foster networks with elements for the impedance function $Z(s)$ given by

$$Z(s) = \frac{(s+1)(s+3)}{s(s+2)}$$

18.5 Find the first, and second Cauer networks of the given functions.

$$Z_1(s) = \frac{2s^3+8s}{s^2+1}$$

$$Z_2(s) = \frac{s^3+4s}{2s^4+20s^2+18}$$

18.6 An impedance function has the pole-zero diagram as shown in Fig. 18.41. Find the impedance function to $z(-4) = \dfrac{3}{8}$ and realise it in Cauer form.

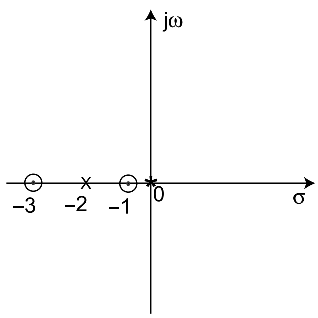


Fig. 18.41 *Pole zero diagram*

18.7 Find the first Foster form and the second Cauer form of the function

$$Z(s) = \frac{2(s+1)(s+3)}{(s+2)(s+6)}$$

18.8 Find the first and second Foster forms of the function

$$Z(s) = \frac{10^9 s^3 + 16\times10^{21} s}{s^4 + 37\times10^{12} s^2 + 36\times10^2 s}$$

18.9 Synthesize first and second Foster form of LC network for the impedance

$$Z(s) = \frac{(s^2+1^2)(s^2+3^2)}{(s^2)(s^2+2^2)}$$

18.10 Find the second Cauer form of the function

$$Z(s) = \frac{s^2+4s+3}{s^2+8s+12}$$

18.11 Find the first Foster form and the second Cauer form after synthesising the impedance function given by

$$Z(s) = \frac{2(s+1)(s+3)}{(s+2)(s+6)}$$

18.12 For the given function

$$Z(s) = \frac{(s+1)(s+3)(s+5)}{s(s+2)(s+4)(s+6)}$$

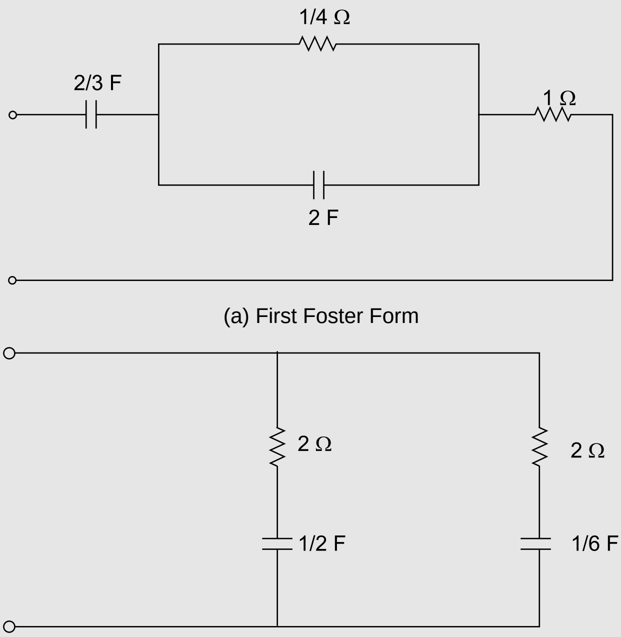
determine the first and second Foster forms of realisation, and the Cauer, first and second forms of realisation.

Answers to Practice Problems

18.1 (a) Hurwitz (b) Hurwitz (c) Not Hurwitz

18.3 The function is a minimum positive real function

18.4



(b) Second Foster Form

Fig. 18.42

18.5

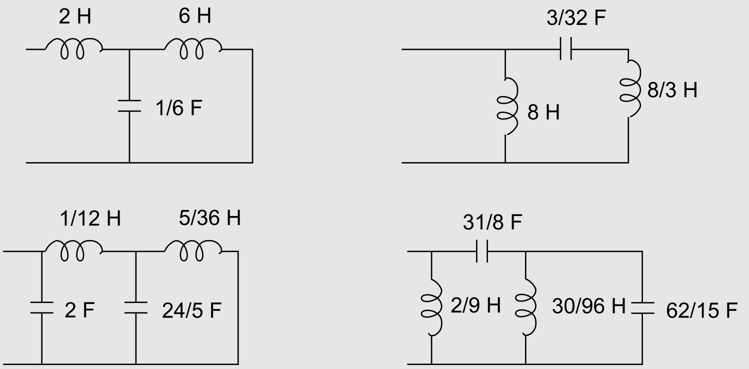
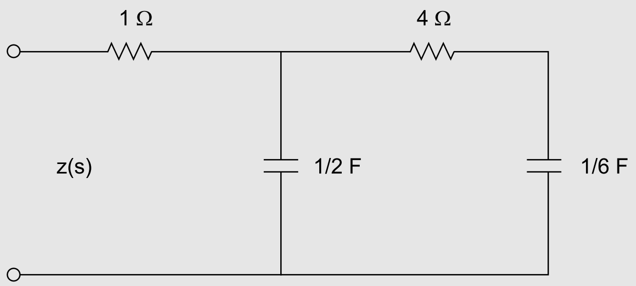
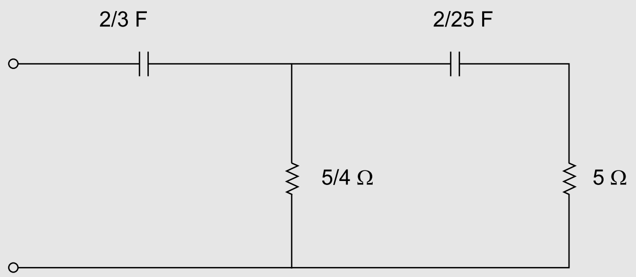


Fig. 18.43

18.6 $Z(s) = \dfrac{(s+1)(s+3)}{s(s+2)}$



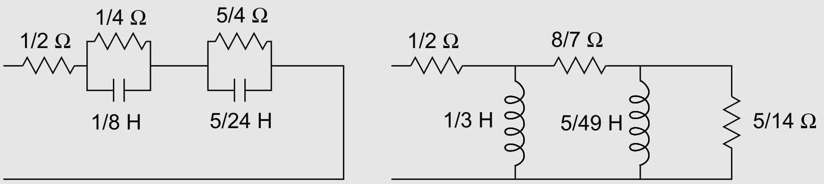
(a) First Cauer Form



(b) Second Cauer Form

Fig. 18.44

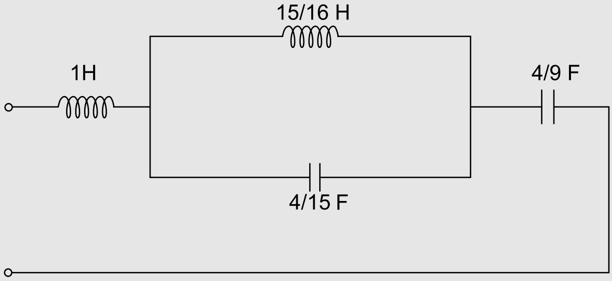
18.7.



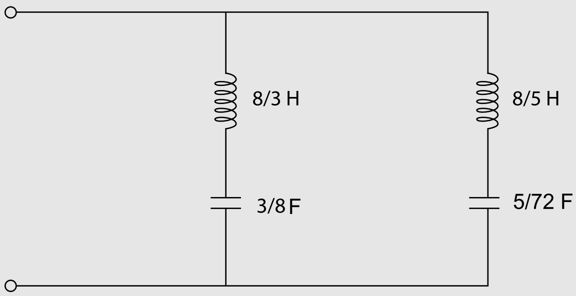
I Foster Form   II Cauer Form

Fig. 18.45

18.9



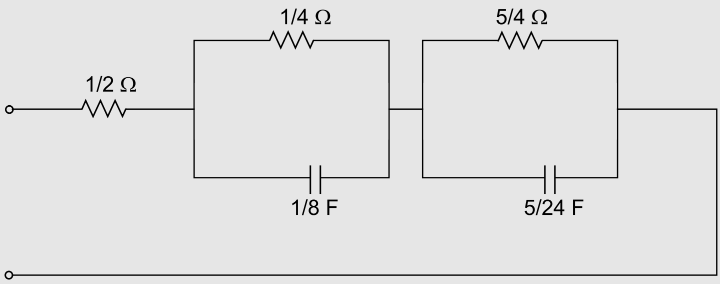
(a) First Foster Form



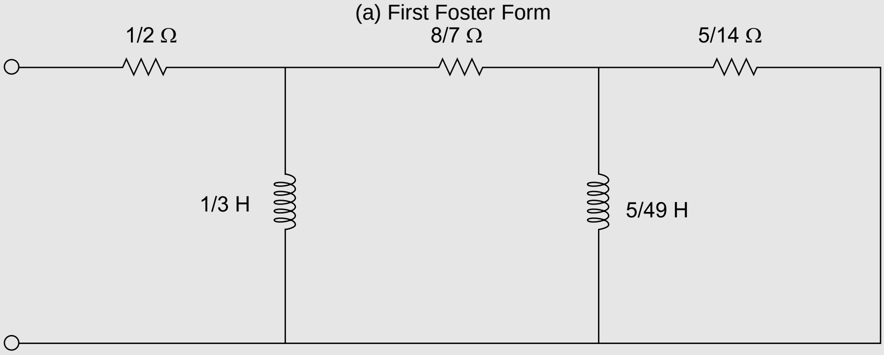
(b) Second Foster Form

Fig. 18.46

18.11



(a) First Foster Form



(b) Second Cauer Form

Fig. 18.47

Objective-Type Questions

18.1 A polynomial must satisfy the condition that

(a) $Z(s)$ is a real function
(b) all the roots of $P(s)$ have zero real parts, or negative real parts
(c) both (a) and (b)
(d) none of the above

18.2 Hurwitz polynomial possesses one of the conditions that

(a) all the quotients in the polynomial $P(s)$ must be positive
(b) the roots of $P(s)$ must lie on the right half of the S-plane
(c) The ratio of $P(s)$ and $P'(s)$ gives negative quotients
(d) $P(s)$ may have missing terms

18.3 The function is said to be positive real, when

(a) the poles and zeros lie on the right half of the S-plane
(b) the poles and zeros lie on the left half of the S-plane
(c) the poles and zeros are simple and lie on the imaginary axis
(d) both (b) and (c)

18.4 The driving point impedance with poles at $\omega = 0$ and $\omega = \infty$ must have the

(a) s terms in the denominator and an excess term in the numerator
(b) s term in the numerator and an excess term in the denominator
(c) s term in the numerator and equal number of terms in the numerator and the denominator
(d) s term in the denominator and equal number of terms in the numerator and the denominator

18.5 In the first Foster form, the presence of first element capacitor C_0 indicates

(a) pole at $\omega = 0$
(b) pole at $\omega = \infty$
(c) zero at $\omega = 0$
(d) zero at $\omega = \infty$

18.6 In the first Foster form, the presence of last element inductor L_∞ indicates

(a) pole at $\omega = 0$
(b) pole at $\omega = \infty$
(c) zero at $\omega = 0$
(d) zero at $\omega = \infty$

18.7 Pole at infinity indicates that the

(a) degree of numerator is greater than that of denominator
(b) degree of denominator is greater than that of numerator
(c) degree of numerator is equal to the degree of denominator
(d) none of the above

18.8 In the first Cauer LC network, the first element is a series inductor when the driving point function consists of

(a) pole at $\omega = \infty$
(b) zero at $\omega = \infty$
(c) pole at $\omega = 0$
(d) zero at $\omega = 0$

18.9 In the second Cauer LC network, the last element is an inductor, when the driving point function consists of

(a) pole at $\omega = 0$
(b) pole at $\omega = \infty$
(c) zero at $\omega = \infty$
(d) zero at $\omega = 0$

Answers to Objective-Type Questions

18.1	(c)	**18.2**	(a)	**18.3**	(d)	**18.4**	(a)	**18.5**	(a)
18.6	(b)	**18.7**	(a)	**18.8**	(a)	**18.9**	(b)		

An Introduction to PSpice

CHAPTER 19

19.1 INTRODUCTION

SPICE is a universal standard simulator used to simulate the operation of various electric circuits and devices. PSpice is one of the many commercial derivatives of SPICE. PSpice helps to simulate electrical circuit design before they are set up. This allows the designer to decide if changes are needed, without touching any hardware. PSpice also helps to check the design and response of the network. In short PSpice is a simulated lab bench on which the test circuit can be created and measurement can be made.

SPICE stands for Simulation Program with Integrated Circuit Emphasis. PSpice is a member of the spice family of circuit simulators, developed at the University of California, Berkeley. PSpice is a commercial product developed by Microsim Corporation.

19.2 WHAT IS PSPICE?

In 1968, a junior faculty member at the University of California, Berkeley started a course on circuit simulation, hoping to develop a new circuit simulator for his work in circuit optimisation. He along with a few students assembled a non-linear circuit simulator which was to become the foundation for SPICE. The first simulator was named CANCER (Computer Analysis of Non-linear Circuits Excluding Radiation). But its capability was limited as it could not handle more components and/or circuit nodes.

During the 1970s, improvements in CANCER continued. In 1971, an improved version of CANCER named SPICE 1 (Simulation Program with Integrated Circuit Emphasis 1) was released. The next major breakthrough was in 1975 with the introduction of SPICE 2. From 1975 through 1983, Berkeley continued improving and upgrading the SPICE 2 program. In 1983, SPICE 2G.6 version was released. All these versions were written in FORTRAN source code. Later it was rewritten in C. The new C version of the program was known as SPICE 3. SPICE 3 offers several technical advantages as compared to SPICE 2. Several vendor-offered versions of SPICE are there in the market. Some of the better-known simulators include Meta-Software's HSPICE, Intusoft's IS-SPICE, Spectrum Software's MICRO-CAP and Microsim's PSpice. All these were developed from the original SPICE 2. Although many other SPICE-based programs exist, these four represent the best known simulators. Majority of the SPICE-like simulators are still based on SPICE 2G.6, that is a SPICE 2 version. PSpice, which uses the same algorithms as SPICE 2 (and confirms to its output syntax), shares this emphasis on micro circuit technology. However, the electrical concepts are general and are useful for all sizes of circuits and a wide range of applications.

19.3 GETTING STARTED WITH PSPICE

SPICE is widely used in the academic and industrial worlds to simulate the operation of various electric circuits and devices. In order to use the educational version of PSpice from Microsim or elsewhere, the minimum requirement for any PC are PC/XT/AT with atleast 512 KB of RAM, a fixed disk, MS-DOS version 3.0 or later and a monochrome or colour graphic monitor with a 20 MB hard disc. PSpice was developed by Microsim Corporation in California and made available in 1984, and later by ORCAD. PSpice has been made available in different operating systems such as DOS; WINDOWS or UNIX, etc. Though the Windows version of PSpice is becoming more and more popular, a general description is presented in this chapter. PSpice can analyse upto roughly 125 elements and over 100 nodes. It is capable of performing dc analysis transient analysis and ac analysis. In addition it can also perform transfer function analysis, Fourier analysis and operating point analysis. The circuit may contain resistors, inductors, capacitors, independent and dependent sources, OP amps, transformers, transmission lines and semiconductor devices.

Make sure that the operating system and PSpice is already installed in your P.C. with the necessary configuration. The best way to learn a circuit simulator is to do simulations. Running this simulation involves the following main steps.

(i) create the input file or circuit file. It is also called a program for the simulator
(ii) run the simulator
(iii) find where the output is available
(iv) check the output. A text editor is required to create the input file, then the PSpice program can be run specifying the input file. If everything works, PSpice will read the input file executes and place the results in an output file. This output file may also be directed to a printer to get a print out.

Though PSpice is a powerful program that can carry out many different procedures, a brief introduction for the elementary types of dc, ac and transient analysis is presented in this chapter. The procedure described in this chapter is general, many advanced versions of spice packages are now available, students are advised to consult the user's guide supplied by the vendor for a specific PSpice simulation and design.

19.4 SIMULATION STEPS

As a first step in simulation, an input file must be created for the given circuit which is also called the circuit file. Always begin with a complete sketch of the circuit. Label the nodes using distinct markings. There must be always a zero (0) node, which will be the reference node. The other nodes can have either numerical or alphabetical designations.

Title or Comment Line

The input file must be given a name (title or description of the file). Any line beginning with an asterisk (*) will be printed or displayed with the program, but will otherwise be ignored by the computer. Any line may be a title line, by starting it with a "*" in the first column. It is always better to include a statement for every element in the circuit. PSpice allows the user to insert comments or statements on any line by starting the comment with a ";" (semicolon). Everything on the line after the ";" is ignored. PSpice always expects the first line of the circuit file to be a title line. If it describes an element, it will be ignored. Any statement that begins with a "." (period) is called a control statement. The last statement must be the .END statement which completes the description of the entire circuit. After the .END statement, PSpice will let you start another completely different circuit simulation. Upper and lowercase alphabetic characters may be used in PSpice; RSHUNT, Rshunt, refer to the same device.

19.5 COMPONENT VALUES

While representing either large or small component values, the following letters with corresponding scale factors are to be used in PSpice.

Table 19.1 Letters used in PSpice

Symbol	*Meaning*	*Value*	*Exponential form*
F	Femto	10^{-15}	IE-15
P	Pico	10^{-12}	IE-12
N	Nano	10^{-9}	IE-9
U	Micro	10^{-6}	IE-6
M	Milli	10^{-3}	IE-3
K	Kilo	10^{3}	IE 3
MEG	Mega	10^{6}	IE 6
G	Giga	10^{9}	IE 9
T	Tera	10^{12}	IE 12

The symbolic form may be written either using upper or lower case letters. For example M or m indicates milli or 10^{-3}; mega or 10^{6} is written by MEG or meg. All the quantities, or values, in PSpice may be expressed as decimal or floating point values as used by all computer programs. The symbols in Table 19.1, when used as suffixes multiply the number they follow by a power of ten as an example 25N indicates the value of $25 \times 10^{-9} = 0.025\text{E-6}$.

19.6 DC ANALYSIS AND CONTROL STATEMENTS

In dc circuit for spice, only seven circuit elements are used. These are the resistors—two independent sources and four dependent sources. Let us consider the voltage divider circuit shown in Fig. 19.1 (a) to investigate using PSpice.

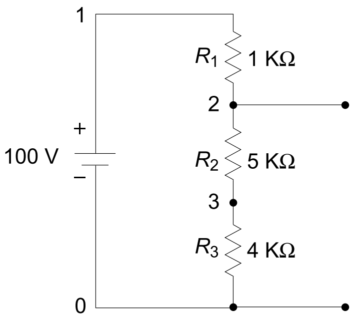


Fig. 19.1 (a)

Figure 19.1 (a) shows a series circuit with a dc voltage source and 3 resistors R1, R2 and R3. To specify the device or element in the circuit file we have to include the name of the element; the location of the element i.e. the nodes between which the element is connected and its value. PSpice uses the basic electrical units for voltage (volts); current (amps) and also uses, ohms, Farads and Henrys. We can specify the elements merely by using appropriate letter as the first letter of the device name as R for resistor, L for inductor C for capacitor, V for independent voltage source and I for independent current source. Now let us write the input file or circuit file for the circuit shown in Fig. 19.1 (a).

```
*voltage divider circuit
 VIN     1     0     100V
 R1      1     2     1K
 R2      2     3     5K
 R3      3     0     4K
 .OP
 .END
```

An editor such as the MS-DOS editor note pad or MS word from MS Office is to be used to enter the circuit file. The file might be suitably named. After running the PSpice program, the result would appear in the output file as follows

Simulation result of Fig. 19.1 (a).

```
NODE   VOLTAGE    NODE    VOLTAGE    NODE    VOLTAGE
(1)      100.0     (2)     90.000     (3)      40.000
VOLTAGE SOURCE CURRENTS
NAME          CURRENT
VIN          –1.00E – 02
TOTAL POWER DISSIPATION 1.00E + 00W
Total job time 1.05.
```

Let us examine the statements in the input file. There is a statement for each element of the circuit. Each line of the input file is a statement. The first line in the program indicates the title of the file. Four lines are used to describe four elements in the circuit. The second line describes the independent voltage source. It is identified by using the first letter of the source (It can be followed by any combination of seven additional letters or numbers). The name (VIN) is followed by a blank, the node (1) to which the positive reference to the source is connected, another blank, and then the node (0) at which the negative terminal is located. Another blank precedes the numerical value of the voltage in volts. 3rd, 4th and 5th lines describes the three resistances in the circuit. A resistor is identified by its first letter (It can be followed by another seven additional letters or integers), the name R1 is followed by one or more blanks, followed by the first node (1), followed by one or more blanks and then the second node (2) and one more blank precedes the value (1 kΩ) of R1. The last two lines in the input file are called control statements. After incorporating all the circuit data in the program, it is necessary to specify the operations that are to be performed. This is done by control statements.

.OP Statement

The .OP Statement is a control statement which instructs the computer to calculate the dc voltage between each node and the reference node.

.END Statement

The .END statement is the another most important control statement which must be used as the last line in every input file program. The .END statement marks the end of the circuit. All the data and commands must come before it. When the .END statement is reached, PSpice does all the specified analysis on the circuit.

There may be more than one circuits in an input file. Each circuit and its commands are marked by a .END statement. PSpice processes all the analysis for each circuit before going on to the next one. Everything is reset at the beginning of each circuit. Having several circuits in one file gives the same results as having them in separate files and running each one separately. Having finished with the file, exit the editor and run the PSpice program. If there are no errors the output analysis of the circuit will be available in the output file.

The control statement .OP gives maximum amount of information. It produces detailed bias point information, that is the voltage of all nodes, the currents and power dissipation of all the voltage sources. If the number of nodes are more, the computer generates a lot of output data that we may not really need.

.PRINT Statement

Instead of the .OP statement, we can use another control statement the .PRINT, for specific outputs. The print control statement consists of .PRINT followed by a space and DC, another space, and the desired node voltage or node voltages separated by at least one space. For example the following statement indicates the voltage at node 2 and node 3 with reference to zero node.

PRINT DC V(2) V(3)

In addition, the voltage between two nodes current values may be specified by .PRINT statement as .PRINT DC V(1, 3) I(R1). The above statement indicates in the output file, the voltage between nodes 1 and 3, and the current through resistor R1. One important point is that the .PRINT command does not result in printing of any value on paper. It is merely made available in the output file in computer memory. If the printer is connected to the system, then the appropriate command will produce a printed output. The currents through the branches can be measured in PSpice. If an independent voltage source exists in that branch. Thus, if, we want to calculate the value of current in some branch of a circuit that does not contain an independent voltage source, then we have to insert a voltage source with a value of 0 volts in the branch. Let us consider Fig. 19.1 (b). It is required to write the input file to calculate the current through 3 Ω resistor with the indicated direction where no voltage source exists. The four nodes and the reference node have been numbered, the current through 3 Ω is desired, we shall therefore insert (V_3) a 0 volt voltage source in the branch as shown in Fig. 19.1 (c). In SPICE a voltage source current is positive if it were directed from plus to minus through V. Hence, the assumed polarities for V_3 are correct.

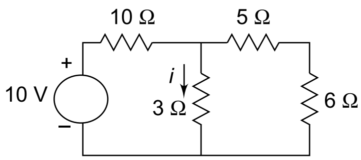


Fig. 19.1 (b)

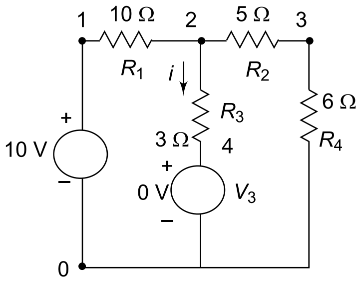


Fig. 19.1 (c)

The data for the circuit file is given by

*current measurement

VIN	1	0	10 V
R_1	1	2	10
R_2	2	3	5
R_3	2	4	3
R_4	3	0	6
V_3	4	0	

.PRINT DC I (R3)

.END

The result of this program is as follows

Simulation result of Fig. 19.1 (*c*)

```
NODE   VOLTAGE     N      V     N      V     N      V
(1)    10.0000    (2)  1.9075  (3)  1.040  (4)  0.0000
VOLTAGE SOURCE CURRENTS
VIN – 8.092 E − 01
V3   6.358 E − 01
```

Total power dissipation 8.09E + 00 WATTS

As mentioned earlier .PRINT statement can be used for several outputs in one table, and mix voltages and currents. The output values you can print are basically node voltages and device (also source) currents. Node voltages can be printed relative to ground (0 node) or relative to another node. Examine the following statements for the circuit shown in Fig. 19.1 (c).

```
.PRINT DC V(1) to print voltage at node 1 (i.e. source voltage 10V)
.PRINT DC V(1,2) to print voltage between node 1 & 2.
```

```
.PRINT DC V(R4) to print voltage across R4
.PRINT DC V(2) V(3) I(R1) to print voltage at node 2,
node 3 and current through R1.
```

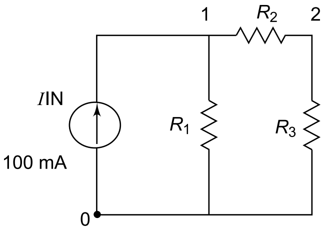


Fig. 19.1 (d)

Current Source

If a current source is present in the circuit, the first node listed in the SPICE statement is the one at the tail of the current arrow and the rest of the listing is similar to the voltage source representation. As an example consider the circuit in Fig. 19.1 (d).

Now the statement for the independent current source in the input file is I I N 0 1 100 M.

19.7 DEPENDENT SOURCES

In the seven circuit elements/devices mentioned in dc analysis we have discussed only three; they are resistor, independent current and voltage sources. The other four elements are dependent sources. They are (VCVS) voltage-controlled voltage sources, (CCCS) current-controlled current source, (VCCS) voltage-controlled current source and (CCVS) current-controlled voltage source. These sources are described in the input file in a way that is similar to the passive devices. The names of VCVS; and VCCS are identified in the circuit file with a name beginning with letters E and G respectively followed by the connecting nodes, control nodes and gain factor in the order mentioned, of course with blanks in between. The following examples illustrate the description of the dependent sources in the file. Let us consider the current in Fig. 19.2 (a) where we have one

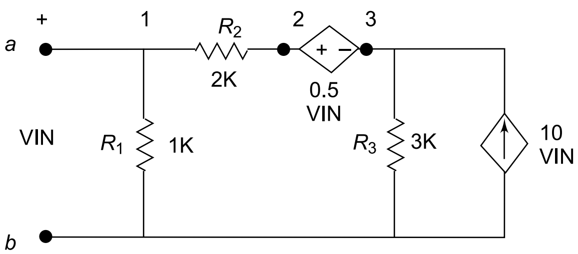


Fig. 19.2 (a)

The VCVS in the circuit of Fig. 19.2 (a) is written in the input file as

E 2 3 1 0 0.5

where, E–Device name

2–Positive node of the device

3–Negative node of the device

1–Controlled voltage positive node

0–Controlled voltage negative node

0.5–Gain factor of the voltage

Similarly, VCCS in the circuit of Fig. 19.2 (a) is written in the input file as

G 0 3 1 0 10

G–Device name

0–node at the tail of the arrow in current source
3–node at the head of the arrow in current source
1–Controlled voltage positive node
0–Controlled voltage 2nd node
10–Gain factor.

Let us consider the current in Fig. 19.2 (b) where we have a CCCS and a CCVS.

The statement for current controlled sources has a name beginning with F and H for CCCS and CCVS respectively, followed by the two nodes defining the direction of the current flow through the dependent source and the name of the V-device that

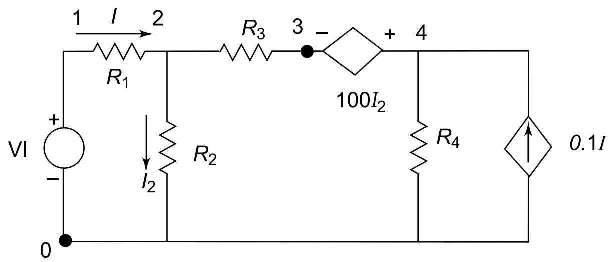


Fig. 19.2 (b)

has the controlling current, as PSpice measures the currents through voltage sources only. In the circuit shown the controlling current I_2 is in the branch R_2 which does not have an independent voltage source, that is, no V-type element. Hence, a slight modification is required in the above circuit. Insert a zero volt independent voltage source in the branch R_2 and name this as VO, and change the above circuit to the circuit shown in Fig. 19.2 (c).

Now the CCVS in the circuit of Fig. 19.2 (c) is written in the circuit file as

H 4 3 VO 100

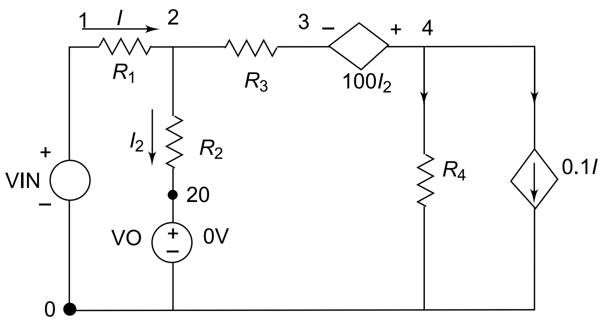


Fig. 19.2 (c)

where, H–Device name
4–Positive node of the device
2–2nd node (negative node) of the device
VO–The name of the zero volt source in the control branch
100–gain factor of the controlling current.

Similarly, CCCS of Fig. 19.2 (c) in the circuit file is listed as

F 4 0 VIN –0.1

where, F–Device name

4–Node at the tail of the arrow of the CCCS

0–Node at the head of the arrow of the CCCS

VIN–Name of the independent voltage source through which the controlling current is carried.

– 0.1–Gain factor of the controlling current. The reason for using minus sign is that the controlling current *I* is directed from minus to plus through VIN in the circuit.

Example 19.1 *Write a SPICE program for the circuit shown in Fig. 19.2 (a) to determine the voltages at node 2 and 3, if VIN = 10 volts dc.*

Solution *voltage dependent sources

```
VIN    1    0    10
R1     1    0    1K
R2     1    2    2K
R3     3    0    3K
E      2    3    1    0    0.5
G      0    3    1    0    10
.OP
.END
```

Simulation result of Example 19.1

```
NODE   VOLTAGE    N     V               N     V
(1)       10      (2)   12.01EW + 03    (3)   12.00E + 03
VOLTAGE SOURCE    CURRENTS
VIN               5.989E + 00
POWER DISSIPATION − 5.99E + 01W; Time 2.43.
```

Example 19.2 *Write a spice program for the circuit shown in Fig. 19.2 (c) to determine the voltages of all nodes, and the power dissipation of all sources. Assume VIN = 100 volts;* $R_1 = R_2 = 2\ \text{k}\Omega$; $R_3 = R_4 = 0.5\ \text{k}\Omega$.

Solution * Current dependent source

```
VIN   1    0    100
R1    1    2    2K
R2    2    20   2K
R3    2    3    0.5K
R4    4    0    0.5K
VO    20   0
H     4    3    VO     100
F     4    0    VIN    −0.1
.OP
.END
```

Simulation result of Example 19.2

NODE	VOLTAGE	N	V	N	V	N	V	N	V
(1)	100	(2)	18.4810	(3)	4.2208	(4)	5.1948	(20)	0.000

Voltage source currents

NAME	CURRENT
VIN	−4.026E − 0.2
VO	9.740E − 03

Total power dissipation 4.03 W.

Current controlled sources

NAME	F
I SOURCE	2.013E − 02

Current controlled voltage sources

NAME	
V-SOURCE	9.740E − 01
I-SOURCE	−3.052E − 02

Total job time 2.32.

19.8 DC SWEEP

In the calculations so far the values for sources maintained fixed values. But when the PSpice analysis is used with a range of input voltages which is called dc sweep, where the sources vary, though the analysis will still calculate quiescent operation. Using this analysis allows to look at the results from many .OP analysis in a single simulation run. That is, when you sweep a source the simulator starts with one value for a source (voltage or current), calculate the DC bias point as it does for the .OP statement, then increments the value and does another dc bias point calculation and so on until the last source value has been analysed.

.DC Statement

The dc sweep analysis is controlled with a .DC statement. The .DC statement gives a range to voltages/currents. This is called a sweep of voltage/current. This statement specifies the values used during the dc sweep. The statement says which source value is to be swept, the starting value, the end value and the amount of increment in each step. Let us insert the .DC statement in the circuit file of Fig. 19.1 (a) and rewrite the file.

```
*voltage divider circuit
VIN    1    0    100V
R1     1    2    1K
R2     2    3    5K
R3     3    0    4K
.OP
.DC   VIN  0  100  10
.END
```

While adding a .DC statement to the circuit file, the other lines used to describe the circuit need not be changed. Adding .DC statement will override the fixed value indicated by the independent source VIN during DC sweep analysis. After running the PSpice program, the output file contains the following simulation result.

Small signal bias solution

```
 N   V       N     V     N     V
(1)100     (2)    90    (3)    40
 VOLTAGE SOURCE CURRENTS
 NAME    CURRENT
 VIN     21.00E − 02
 Total power dissipation 1.00 + 0.0watt
 Time 1.66.
```

The .DC statement followed by name of the source VIN whose voltage is to be swept, the next two values 0 and 100, are for the start and stop voltage values of the sweep, and the last value 10 is the increment.

.PROBE Statement

In PSpice, we have a facility called PROBE which provides us the powerful graphic capability of PSpice. To use the above statement we must instruct PSPICE to create a data file for probe which is done by including the .PROBE statement in the input file. This statement is similar to the .PRINT with the .PROBE you may select node voltages and device currents to be output from the simulation. The .PROBE statement writes the results from DC, AC and transient analysis to a data file named PROBE. DAT for graphics analysis by post-processor. The general forms of the .PROBE statement are

```
.PROBE
.PROBE   V (1)   V (4 3)   I (R4)
```

The first form without any output variable writes all the node voltages and all the device currents to the data file. The second form writes the following output variables to the data file. The voltage of node 1, voltage between node 4 and 3 and the current through R_4. Another important difference between .PRINT and .PROBE statement is that the analysis name (DC, AC or transient) is absent before the output variable in .PROBE statement.

19.9 AC ANALYSIS AND CONTROL STATEMENTS

Another important application of the PSpice simulator is to verify the frequency response of various devices and circuits. The response calculates all the ac node voltages and branch currents over a specified range of frequencies. PSpice calculates the dc node voltage without any special requirements, but in ac analysis, we must specifically ask for it.

Let us consider the circuit shown in Fig. 19.3 (a) which is a series *RLC* circuit with a voltage source of 100 V at an angle of 15°. Each independent voltage and current source in ac analysis is characterised by its amplitude and phase with the source statement in the file. The source frequency is specified in a control statement. (.AC statement). Let us write a suitable input file for the circuit shown in Fig. 19.3 (a).

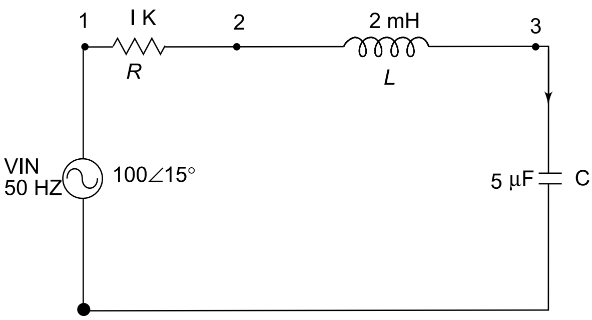


Fig. 19.3 (a)

```
*RLC SERIES CIRCUIT
 VIN    1      0     AC      100    15
 R      1      2     1K
 L      2      3     2mH
 C      3      0     5mF
 .AC    LIN    1     50      50
 .PRINT AC IM (R) IR(R) II(R) IP(R)
 .END
```

The statement VIN 1 0 AC 100 15 indicates the ac source which is connected between nodes 1 and zero with an amplitude of 100 V and a phase angle of 15º. If the phase angle is 0º, the last term in the statement can be omitted.

Simulation result

FREQ	IM(R)	IR(R)	II(R)	IP(R)
5.00E + 0.1	8.438×10^{-2}	5.705×10^{-2}	6.217×10^{-2}	4.746×10^{1}

Time 2.28.

.AC Statement

This statement begins with the specification .AC and continues with four additional terms. The first term indicates the type of frequency sweep (linear, octave and decade). The second term indictes the number of points in the sweep and the 3rd and 4th terms indicate the beginning and ending frequencies. Hence, .AC LIN 1 50 50 statement gives a linear sweep for one frequency only with beginning and ending value of 50 Hz that is it selects only one frequency. If it is required to seep 10 frequencies linearly from 5 Hz to 50 Hz then the ac statement would be

(.AC LIN 10 5 50)

This statement would provide results at the starting and stopping frequencies and eight intermediate frequencies that are uniformly spaced (5, 10, 15, 20, 25, 30, 35, 40, 45, 50).

.PRINT AC Statement

Output from ac analysis may be generated by .PRINT statement, just as in DC analysis. The phase AC replaces DC. The output values that can be printed are node voltages and device currents (source currents) with some special considerations for AC analysis. The voltages and currents may be specified as magnitude, phase, real part, imaginary part or magnitude in dB by adding M, P, R, I and DB respectively as a suffix to "V" (Voltage magnitude) or "I" (Current magnitude). Thus the statement.

```
(.PRINT  AC  IM(R)  IR(R)  II(R)  IP(R))
```

would yield the magnitude, the real component, the imaginary component and the phase angle in degree of the current through R. This is shown in the input file and its simulation result of Fig. 19.3 (a). As mentioned earlier, the frequency sweep can be done in octave and decade also. Their syntax is similar, only thing required is, .AC DEC is used for decade sweep and .AC OCT is used for octave sweep.

19.10 TRANSIENT ANALYSIS

PSpice can be effectively used for transient or time domain analysis. It is used very often for circuit simulation, because this analysis is the tedious and difficult analysis as it involves lengthy integro-differential equations with boundary conditions.

In PSpice we can investigate the circuit transient response for various types of input waveforms, like exponential (EXP), pulse (PULSE); piecewise-linear (PWL); frequency modulated wave (SFFM) and for sinusoidal wave (SIN) forms. Hence, the independent voltage and current sources may be specified in any of the above time-varying waveforms by giving a proper format. The following are the General formats of the statements used to describe the applied voltages (waveforms) in transient analysis.

PWL (T1, V1 T2, V2 … TNVN) describes a piecewise linear waveform. The arguments in parenthesis represent time voltage pairs at the corners of the waveform.

EXP (V1 V2 TD1 TR1 TD2 TR2) describes the exponential waveform initial voltage V1 upto a delay time of TD1 seconds. V2 is the peak voltage with a fall delay time of TD2, TR1 and TR2 are the rise time constant and fall time constants respectively.

PULSE (V1 V2 TD TR TF PW PER) describes the pulse form of voltage with initial voltage (V1); peak value of pulse (V2); delay time (TD); rise time (TR); fall time (TF); width of the pulse (PW); and period of the pulse (PER).

SFFM (VO VA FC MD FS) describes the single frequencies modulated wave, with offset voltage VO peak amplitude (VA); carrier frequency (FC); modulation index (MD) and single frequency (FS).

SIN (VO VA FREQ TD DF PHASE) describes the sinusoidal waveform with an offset voltage of VO, peak value of VA, frequency FREQ, delay time td, damping factor DF and a phase angle. The SIN waveform format is only for transient analysis only.

.TRAN Statement

.TRAN statement specifies the time interval over which the transient analysis takes place. This statement is followed by two values. The first value indicates the print-step (interval) value and the second value indicates the final value of time (length of the time for the analysis). Observe the following .TRAN statement.

```
.TRAN   2ms   20ms
```

wherein the time interval (time step) is 2 ms and the maximum value of time limit is 20 ms. Apart from the time step and final time some more options like starting time (default value is zero), max time for analysis and initial conditions can also be used along with .TRAN statement. Output from transient analysis may be generated by .PRINT statement just as in dc and ac analysis. Hence, transient analysis requires a .PRINT command similar to dc or ac analysis except that the term dc/ac is replaced by TRAN. The statement form is .PRINT TRAN (Any of the eight output variables).

As an example of transient analysis, let us calculate the voltage at node 2 in the circuit shown in Fig. 19.4 (a).

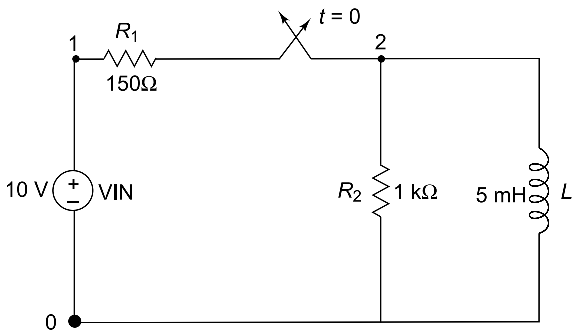


Fig. 19.4 (a)

Let us apply piecewise linear

```
*RL  TRANSIENT
 VIN      1       0       PWL     (0, 0   10 µs,   1V   10 ms,   10 V)
 R1       1       2       150
 R2       2       0       1K
 L        2       0       5M      IC = 0
 TRAN     1ms     10ms
 .PRINT   TRAN    V(2)
 .END
```

IC = 0 in the above file indicates zero initial current in the inductor. After running the PSpice analysis we obtain the following simulation result.

Simulation result

```
N          V         N        V
(1)        0.40      (2)      0.40
Voltage source currents
           V                  0.000E + 00
Total power dissipation       0.00E + 00W
  Transient analysis        Temperature 27 DEG C
```

Time	V(2)
0.000E + 00	0.000E + 00
1.000E − 03	3.002E − 02
2.000E − 03	3.003E − 02
3.000E − 03	3.003E − 02
4.000E − 03	3.003E − 02
5.000E − 03	3.003E − 02
6.000E − 03	3.003E − 02
7.000E − 03	3.003E − 02
8.000E − 03	3.003E − 02
9.000E − 03	3.003E − 02
1.000E − 03	3.003E − 02

Total job time 0.08 sec.

.PROBE Statement

Using probe with transient analysis is identical to what we have done with dc and ac analysis. Include a .PROBE statement to the circuit file. Try the above example with .PROBE statement, and verify on the graph the voltage variation.

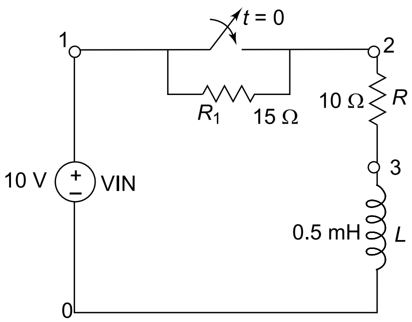


Fig. 19.4 (b)

Consider another example with non-zero initial current as shown in Fig. 19.4 (b). The switch is closed at $t = 0$.

The switch is opened before $t = 0$, the initial current before the closure of the switch is $i(o) = 10/25 = 0.4$A (through inductor). After the switch is closed, the current rises exponentially. The following is the input file for the transient analysis of the circuit in Fig. 19.4 (b).

```
*Transient analysis with I.C.
 VIN      1      0      PWL    (0, 4V   1 µs,   10V   1 ms,   10V)
 R        2      3      10
 L        3      0      0.5M    IC = 0.5A
 .TRAN    10 µs  1 ms
 .PROBE
 .END
```

Notice that, the 1st time voltage pair in PWL parenthesis is written as 0, 4. This is because when the switch is closed at t (o^+) the voltage 4V will appear across R.(0.5 × 10 = 4 V). Run the PSpice program and verify the result.

Sometimes, capacitors have initial voltages, if a 0.5 mF capacitor connected between nodes 3 and 4, carrying an initial voltage of 50 V may be specified in input with the following description.

```
C   3   4   0.5 µF   IC = 50
```

We can also use sinusoidal excitation in transient analysis to verify the frequency response of the circuit. If the input source is a simple sinusoidal voltage source without any off-set values and time delays with a maximum value of 215 V and frequency of 50 Hz. It can be represented in input file as

VIN 0 1 Sin (0 215 50 Hz).

Additional Solved Problems

Problem 19.1 *For the circuit shown in Fig. 19.5 write the input file, run the PSpice program and obtain the current through R_1; R_5, voltage at node 2 and voltage between node 2 and 3.*

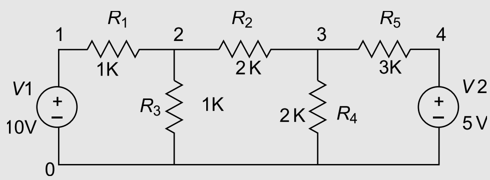


Fig. 19.5

Solution *TWO VOLTAGE SOURCES

```
V1      1    0      10
R1      1    2      1K
R2      2    3      2K
R3      2    0      1K
R4      3    0      2K
R5      3    4      3K
V2      4    0      5
.OP
.DC     VI   50     50      5
.PRINT  DC   1(R1)  1(R5)   V(2)   V(2,3)
.ENDs
```

(.DC is a sweep statement it allows to sweep through a set of voltage of source V1. Though we are not interested in sweep in this problem, it is required for the next (.PRINT) statement. Without .DC statement .PRINT is not valid).

The order in which the elements in the input file are listed makes no difference in the PSpice analysis.

Simulation result

DC transfer curves

VI	I(R1)	I(R5)	V(2)	V(2,3)
5.0×109	2.811×10^{-2}	1.486×10^{-3}	2.189×109	1.243×109

Small signal bias solution i

N	V	N	V	N	V	N	V
(1)	10	(2)	4.5946	(3)	2.9730	(4)	5.000

Voltage source currents

V1	-5.405×10^{-3}
V2	-6.757×10^{-4}

```
Total power dissipation 5.74 × 10^-2 watts
Total job time 2.24.
```

Problem 19.2 *For the circuit shown in Fig. 19.6 write the input file to obtain voltage across R_L and current through R_1 when the input voltage varies from 0 to 100 V.*

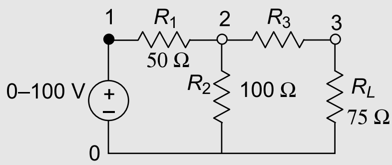


Fig. 19.6

Solution

```
*DC SWEEP
VIN      1     0      100
R1       1     2      50
R2       2     0      100
R3       2     3      25
RL       3     0      75
.OP
.DC VIN  0     100    10
.PRINT   DC    V(RL)  I(RI)
.END
```

The variation of the source voltage from 0 to 100 is set with 10 V increment.

After running the PSpice programme the output file consists a table; showing the relation between VIN V(RL) and I(RI).

Following is the simulation result.

DC Transfer curves

VIN	V(RL)	I(RI)
0	0	0
10	3.750	1×10^{-1}
20	7.5	2×10^{-1}
30	1.125×10^{1}	3×10^{-1}
40	1.5×10^{1}	4×10^{-1}
50	1.875×10^{1}	5×10^{-1}
60	2.25×10^{1}	6×10^{-1}
70	2.625×10^{1}	7×10^{-1}
80	3×10^{1}	8×10^{-1}
90	3.375×10^{1}	9×10^{-1}
100	3.75×10^{1}	1.00

Small signal bias solution

N	V	N	V	N	V
(1)	100	(2)	50	(3)	37.5

VOLTAGE SOURCE CURRENTS

VIN –1.00

Total power dissipation 1×10^{2} W

Time 1.47.

Problem 19.3 *Obtain SPICE solution for the voltages at all nodes for the circuit shown in Fig. 19.7.*

Assume $V_{IN} = 100$ *V;* $R_1 = 1$ *K;* $R_2 = 500\ \Omega$*;* $R_3 = 100\ \Omega$*;* $R_4 = 2$ *K;* $I_1 = 20$ *mA and* $I_2 = 25$ *mA.*

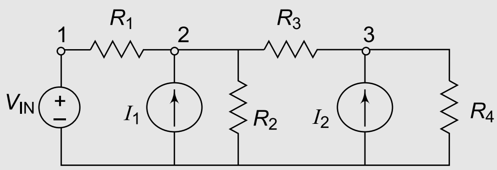


Fig. 19.7

Solution

```
*CURRENT SOURCES
VIN   1   0    100
R1    1   2    1K
R2    2   0    500
R3    2   3    100
R4    3   0    2K
I1    0   2    20M
I2    0   3    25M
.OP
.END
```

Run the PSpice analysis and the output of the result is as follows.

```
Small signal bias solution
N      V      N      V      N      V
(1)    100    (2)    4.37   (3)  41.7810
Voltage source currents
VIN   -5.863 × 10^-2
```

Total power dissipation 5.86 W

Total time 1.31.

Problem 19.4 *For the circuit shown in Fig. 19.8 (a). Find the current, I and voltage at node 3.*

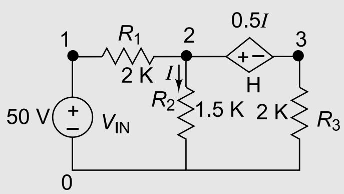


Fig. 19.8 (a)

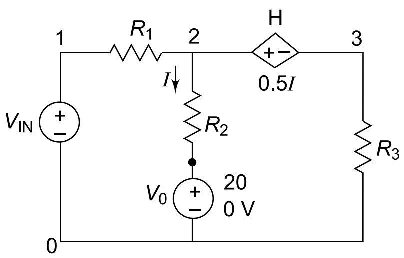


Fig. 19.8 (b)

Solution Since the branch in which the current is to be found, does not have an independent voltage source, assume = 0-volt voltage source with proper polarities as shown in Fig. 19.8 (b).

```
*CURRENT DEPENDENT SOURCE
     VIN    1     0     50
     R1     1     2     2K
     R2     2    20   1.5K
```

```
R3     3     0     2K
VO     20    0
H      2     3     VO 0.5
.OP
.END
```

Simulation result

Small signal bias solution

N	V	N	V	N	V	N	V
(1)	50	(2)	15	(3)	14.997	20	0.0

Voltage source currents

VI -1.75×10^{-2}

VO 1×10^{-2}

Total power dissipation 8.75×10^{-1}

Current-controlled voltage sources

Name	H
V-SOURCE	5×10^{-3}
I-SOURCE	7.498×10^{-3}

Time 5.12.

Problem 19.5 *Find the magnitude of the current, its real, imaginary components and its phase with respect to source in the series R-L circuit shown in Fig. 19.9.*

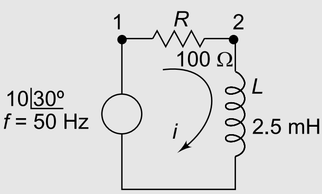


Fig. 19.9

Solution

```
*AC RL series circuit
V_IN     1     0      AC     10     30
R        1     2      100
L        2     0      2.5M
.AC      LIN   1      50     50
.PRINT   AC    IM(R)  IR(R)  II(R)  IP(R)
.END
```

Simulation result

Small signal bias solution

In dc bias calculations, all node voltages; source currents and powers are zero.

AC Analysis

FREQ	IM(R)	IR(R)	II(R)	IP(R)
5.00E + 01	1.000E − 01	8.699E − 02	4.932E − 02	2.955E + 01

Total job time 0.96.

Problem 19.6 *For the given series RLC circuit shown in Fig. 19.10 find the resonant condition and plot the graphs using PSpice program.*

Assume $R = 25\ \Omega$; $L = 10$ mH and $C = 100\ \mu F$; $V = 100$ V.

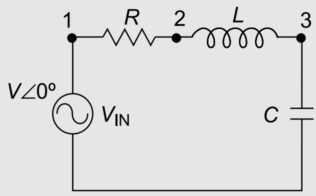


Fig. 19.10

Solution The resonance frequencies $f_r = \dfrac{1}{2\pi\sqrt{10\times10^{-3}\times100\times10^{-6}}}$

$= 159$ Hz

It is observed that the resonance frequency is 159 Hz. Hence, to plot a wide range of frequencies in ac sweep, the .AC statement calls for a linear sweep. Let us fix the starting frequency as 5 Hz and stop frequency as 1000 Hz in 100 steps. Now the input file is given by

```
*RLC series resonance
 VIN      1     0       AC      100
 R        1     2       25
 L        2     3       10M
 C        3     0       100u
 .AC      LIN           100     5     1000
 .PROBE
 .PRINT                 AC      I(R)
 .END
```

Run the PSpice analysis and see the PROBE screen display. There are many variables that can be displayed. These are all menu driven and can be easily learned on screen. You can simultaneously display many quantities on the same graph by incorporating .PROBE statement.

In the AC analysis you will find a linear sweep of 100 frequencies starting from 5 Hz to 1000 Hz. The maximum current 3.999 A is observed at 1.558×10^2 Hz.

Total job time is 1.31.

Problem 19.7 *For the coupled circuit shown in Fig. 19.11, the coefficient of coupling is 0.5. Use SPICE programme to find currents in L_1, L_2. Take $R_1 = R_2 = 10\ \Omega$; $L_1 = L_2 = 20$ mH; $C = 5\ \mu F$ and $R_L = 50\ \Omega$.*

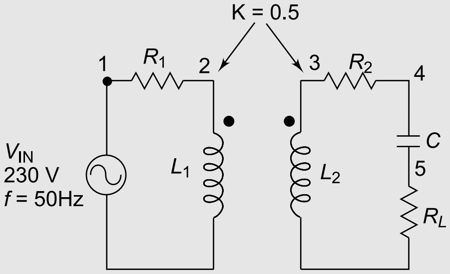


Fig. 19.11

Solution Coupled coils may also be specified in spice input file. The coefficient of coupling is always greater than zero and maximum value is one. If two coils with self inductances of L_1 and L_2 are mutually coupled with a mutual inductance of M, then the coefficient of coupling is given by $K = \frac{M}{\sqrt{L_1 L_2}}$

```
*Coupled coils
 VIN    1     0      AC     230
 R1     1     2      10
 R2     3     4      10
 L1     2     0      20M
 L2     3     0      20M
 RL     5     0      50
 C      4     5      5u
 K      L1    L2     0.5
 .AC    LIN   1      50Hz   50Hz
 .PRINT       AC     I(R1)  I(R2)
 .END
```

Simulation output

```
Small signal DC biasing values will be zero.
AC analysis
FREQ           I(R1)        I(R2)
5.000E + 01 1.946E + 01  9.655E − 02
Time 1.66.
```

Problem 19.8 *For the circuit shown in Fig. 19.12 determine the voltage at node 1 and node 2, when the switch is closed at time t = 0. Use piecewise linear function. Use $R_1 = R_2 = 50\ \Omega$ and $L = 10$ mH.*

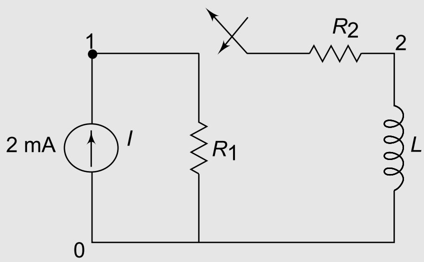


Fig. 19.12

Solution As already mentioned in transient analysis, the .TRAN statement should specify time interval (TSTEP) and length of time. Generally, PSpice employs a variable time interval which is larger when the output is relatively constant, and smaller when the output changes more rapidly. Generally for a circuit having a time constant of τ, we can use TSTEP as 0.1 τ and a maximum time of 10 τ or use the step time as one tenth of the length of the time for analysis.

$$\tau = \frac{L}{R_{eq}} = \frac{10 \times 10^{-3}}{100} = 100\ \mu\text{ sec}$$

Therefore let us use TSTEP as 10 μ sec and max time as 100 μ sec. Now the input file for the transient analysis is given by

```
*RL TRANSIENT
 I     0    1    PWL (0,0   10µs,   2M   100µs,   2M)
 R1    1    0    50
 R2    1    2    50
```

```
L          2     0      10M
.TRAN                   10 µs    100 µs
.PRINT                  TRAN     v(1)     v(2)
.END
```

TRANSIENT ANALYSIS

TIME	V(1)	V(2)
0	0	0
1×10^{-5}	9.758×10^{-2}	9.516×10^{-2}
2×10^{-5}	9.306×10^{-2}	8.611×10^{-2}
3×10^{-5}	8.896×10^{-2}	7.792×10^{-2}
4×10^{-5}	8.525×10^{-2}	6.379×10^{-2}
5×10^{-5}	8.19×1^{-2}	5.772×10^{-2}
6×10^{-5}	7.886×10^{-2}	5.772×10^{-2}
7×10^{-5}	7.611×10^{-2}	5.223×10^{-2}
8×10^{-5}	7.363×10^{-2}	4.726×10^{-2}
9×10^{-5}	7.138×10^{-2}	4.276×10^{-2}
10×10^{-5}	6.935×10^{-2}	3.869×10^{-2}

Time 2.53 sec.

Problem 19.9 *For the source free circuit shown in Fig. 19.13 (a) the initial current in the inductor is 50 mA and the switch is closed at time $t = 0$. Obtain the growth of the current through L. Assume $R = 50$; $L = 5$ H.*

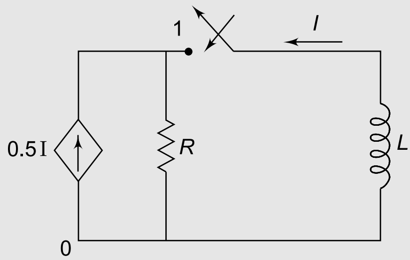


Fig. 19.13 (a)

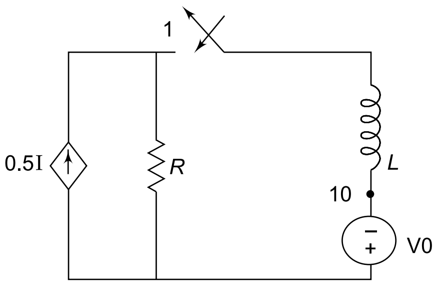


Fig. 19.13 (b)

Solution To find I in L where the source is absent, we assume a zero volt voltage source with proper polarities as shown in Fig. 19.13 (b).

```
*SOURCE   FREE   RL    CIRCUIT
VO        O      10
L         10     1     5    IC = 50M
R         1      0     50
F         0      1     VO   0.5
.TRAN            IM    300M UIC
.PRINT           TRAN  I(L)
.END
```

UIC indicates the use initial condition. The transient analysis results are given upto 10 milli seconds. The current through inductor will be zero at ≈ 275 m sec. By incorporating .PROBE statement the variation of $I(L)$ can be observed in the graphic display.