

**BHARTIYA INSTITUTE OF ENGINEERING & TECHNOLOGY,  
SIKAR**

**Lab File**

**(MODELLING AND SIMULATION LAB)**

**(6EE4-24)**

**Semester: 6<sup>th</sup>**



**Faculty: Mr. Megh Raj**

**(Asst. Professor)**

**Department of Electrical Engineering**

**6EE4-24: MODELLING AND SIMULATION LAB****Credit: 1****ETE:20)****Max. Marks: 50(IA:30,****OL+OT+2P****Hours****End Term Exam: 2**

1. Simulate Swing Equation in Simulink (MATLAB)
2. Modeling of Synchronous Machine.
3. Modeling of Induction Machine.
4. Modeling of DC Machine.
5. Simulate simple circuits.
6. (a) Modeling of Synchronous Machine with PSS (b) Simulation of Synchronous Machine with FACTS device.
7. (a) Modeling of Synchronous Machine with FACTS device (b) Simulation of Synchronous Machine with FACTS devices.
8. FACTS Controller designs with FACT devices for SMIB system.

## EXPERIMENT NO:1

**Aim:** To solve the swing equation of the given problem by using point-by-point method and write a MATLAB program to verify the result.

**PROBLEM:** A 20 MVA, 50Hz generator delivers 18MW over a double circuit line to an infinite bus. The generator has KE of 2.52MJ/MVA at rated speed. The generator transient reactance is  $X_d=0.35$ p.u. Each transmission circuit has  $R=0$  and a reactance of 0.2pu on 20 MVA Base.  $|E|=1.1$  p.u and infinite bus voltage  $V=1.0$ . A three phase short circuit occurs at the midpoint of one of the transmission lines. Plot swing curves with fault cleared by simultaneous opening of breakers at both ends of the line at 6.25 cycles after the occurrence of fault. Also plot the swing curve over the period of 0.5 s if the fault sustained.

### THEORY:

### THEORETICAL SOLUTION:

### MATLAB PROGRAM:

Program 1: Save this part in another m-file with name swing.m

```
%Defining the function swing
function[time ang]=swing(tc)
k=0;v=1;E=1.1;pm=0.9;T=0.5;delT=0.05;ddelta=0;time(1)=0;ang(1)=21.64;xdf=1
.25;xaf=0.55;t=0;
delta=21.64*pi/180;i=2; m=2.52/(180*50);
while t<T
if t<tc
x=xdf;
else x=xaf;
end
pmax=(E*v)/x;
pa=pm-pmax*sin(delta);
ddelta=ddelta+(delT^2*(pa/m));
delta=(delta*180/pi+ddelta)*(pi/180);
deltadeg=delta*180/pi; t=t+delT;
```

```

time(i)=t;
ang(i)=deltadeg; i=i+1;
end
end

```

**Program 2:** Main program that is dependent on swing.m

```
% solution of Swing equation by point-by-point method
```

```
Clc
```

```
clear all
```

```
close all
```

```
for i=1:2
```

```
tc=input('enter the value of clearing time:\n');
```

```
[time,ang]=swing(tc)
```

```
t(:,1)=time;
```

```
a(:,i)=ang;
```

```
end
```

```
plot(t,a(:,1),'*-',t,a(:,2),'d-')
```

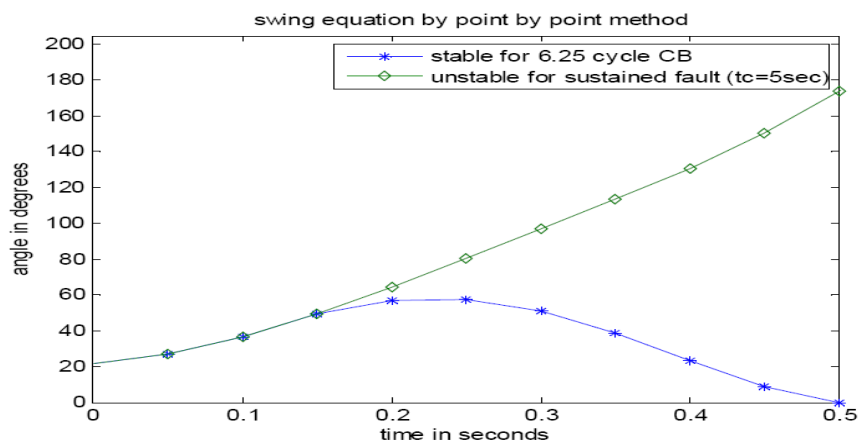
```
axis([0 0.5 0 inf])
```

```
t,a
```

**Inputs to main program:**

Enter the value of clearing time as 0.25 sec, and 5 sec

**EXPECTED OUTPUT:**



**RESULT:**

## EXPERIMENT NO:2

**AIM:** - Modeling of Synchronous Machine.

**Keypoints:-**MATLAB Software

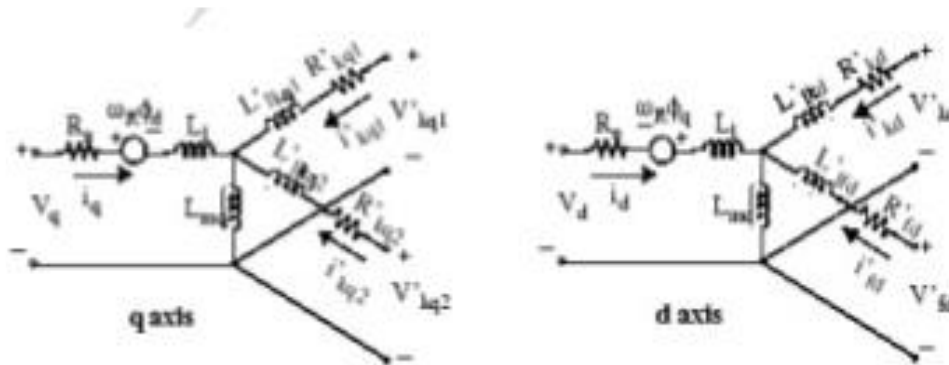
### THEORY:

The Synchronous Machine block operates in generator or motor modes. The operating mode is dictated by the sign of the mechanical power (positive for generator mode, negative for motor mode). The electrical part of the machine is represented by a sixth-order state-space model and the mechanical part is the same as in the Simplified Synchronous Machine block.

The model takes into account the dynamics of the stator, field, and damper windings. The equivalent circuit of the model is represented in the rotor reference frame (qd frame). All rotor parameters and electrical quantities are viewed from the stator. They are identified by primed variables. The subscripts used are defined as follows:

- $d,q$ : d and q axis quantity
- $R,s$ : Rotor and stator quantity
- $l,m$ : Leakage and magnetizing inductance
- $f,k$ : Field and damper winding quantity

The electrical model of the machine is



with the following equations.

$$\begin{aligned}
 V_d &= R_s i_d + \frac{d}{dt} \varphi_d - \omega_R \varphi_q \\
 V_q &= R_s i_q + \frac{d}{dt} \varphi_q + \omega_R \varphi_d \\
 V'_{fd} &= R'_{fd} i'_{fd} + \frac{d}{dt} \varphi'_{fd} \\
 V'_{kd} &= R'_{kd} i'_{kd} + \frac{d}{dt} \varphi'_{kd} \\
 V'_{kq1} &= R'_{kq1} i'_{kq1} + \frac{d}{dt} \varphi'_{kq1} \\
 V'_{kq2} &= R'_{kq2} i'_{kq2} + \frac{d}{dt} \varphi'_{kq2} \\
 \varphi_d &= L_d i_d + L_{md} (i'_{fd} + i'_{kd}) \\
 \varphi_q &= L_q i_q + L_{mq} i'_{kq} \\
 \varphi'_{fd} &= L'_{fd} i'_{fd} + L_{md} (i_d + i'_{kd}) \\
 \varphi'_{kd} &= L'_{kd} i'_{kd} + L_{md} (i_d + i'_{fd}) \\
 \varphi'_{kq1} &= L'_{kq1} i'_{kq1} + L_{mq} i_q \\
 \varphi'_{kq2} &= L'_{kq2} i'_{kq2} + L_{mq} i_q
 \end{aligned}$$

This model assumes currents flowing into the stator windings. The measured stator currents returned by the Synchronous Machine block (Ia, Ib, Ic, Id, Iq) are the currents flowing out of the machine.

**Dialog Box and Parameters:** In the Fundamental Blocks (powerlib) library you can choose between three Synchronous Machine blocks to specify the parameters of the model. They simulate exactly the same synchronous machine model; the only difference is the way that you enter the parameter units in the Parameters tab.

### Preset model

Provides a set of predetermined electrical and mechanical parameters for various synchronous machine ratings of power (kVA), phase-to-phase voltage (V), frequency (Hz), and rated speed (rpm).

Select one of the preset models to load the corresponding electrical and mechanical parameters in the entries of the dialog box. Select No if you do not want to use a preset model, or if you want to modify some of the parameters of a preset model, as described below.

When you select a preset model, the electrical and mechanical parameters in the Parameters tab of the dialog box become nonmodifiable (unavailable). To start from a given preset model and then modify machine parameters, do the following:

Select the preset model that you want to initialize the parameters.

Change the Preset model parameter value to No. This action does not change the machine parameters. By doing so, you just break the connection with the particular preset model.

Modify the machine parameters as you want, then click Apply.

### Mechanical input

Allows you to select the mechanical power applied to the shaft or the rotor speed as a Simulink® input of the block, or to represent the machine shaft by a Simscape™ rotational mechanical port.

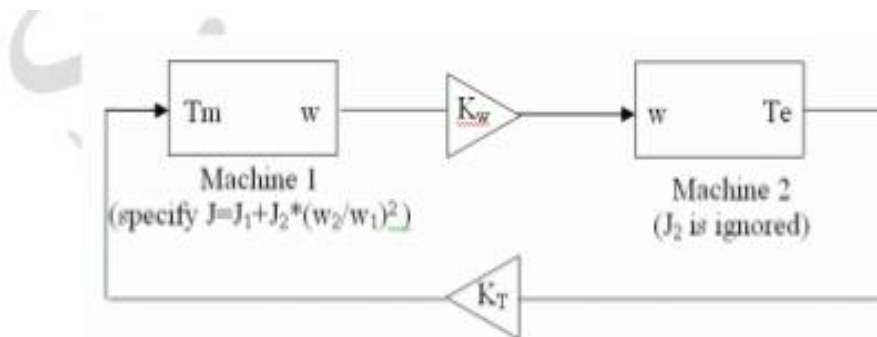
Select Mechanical power Pm to specify a mechanical power input, in W or in pu, and change labeling of the block input to Pm. The machine speed is determined by the machine Inertia J (or inertia constant H for the pu machine) and by the difference between the mechanical torque Tm, resulting from the applied mechanical power Pm, and the internal electromagnetic torque Te. The sign

convention for the mechanical power is when the speed is positive, a positive mechanical power signal indicates generator mode and a negative signal indicates motor mode.

Select Speed  $w$  to specify a speed input, in rad/s or in pu, and change labeling of the block input to  $w$ . The machine speed is imposed and the mechanical part of the model (inertia constant  $H$ ) is ignored. Using the speed as the mechanical input allows modeling a mechanical coupling between two machines.

The next figure indicates how to model a stiff shaft interconnection in a motor-generator set, where both machines are synchronous machines.

The speed output of machine 1 (motor) is connected to the speed input of machine 2 (generator). In this figure friction torque is ignored in machine 2. Therefore, its electromagnetic torque output  $T_e$  corresponds to the mechanical torque  $T_m$  applied to the shaft of machine 1. The corresponding mechanical input power of machine 1 is computed as  $P_m = T_m * w$ . The  $K_w$  factor takes into account speed units of both machines (pu or rad/s) and gear box ratio  $w_2/w_1$ . The  $K_T$  factor takes into account torque units of both machines (pu or N.m) and machine ratings. Also, as the inertia  $J_2$  is ignored in machine 2,  $J_2$  referred to machine 1 speed must be added to machine 1 inertia  $J_1$ .



Select Mechanical rotational port to add to the block a Simscape mechanical rotational port that allows connection of the machine shaft with another machine shaft or with other Simscape blocks having mechanical rotational ports. The Simulink input representing the mechanical power  $P_m$  or the speed  $w$  of the machine is then removed from the block.

#### Waveforms:

**Result:** We have successfully modeled the synchronous machine.

#### Precautions:

- ✓ Do not temper with the settings of the software
- ✓ Study all the observations very carefully.

## **EXPERIMENT NO:3**

**AIM:** - Modeling of Induction Machine.

**Keypoints:-**MATLAB Software

### **THEORY:**

The Asynchronous Machine block implements a three-phase asynchronous machine (wound rotor, single squirrel-cage, or double squirrel-cage). It operates in either generator or motor mode. The mode of operation is dictated by the sign of the mechanical torque:

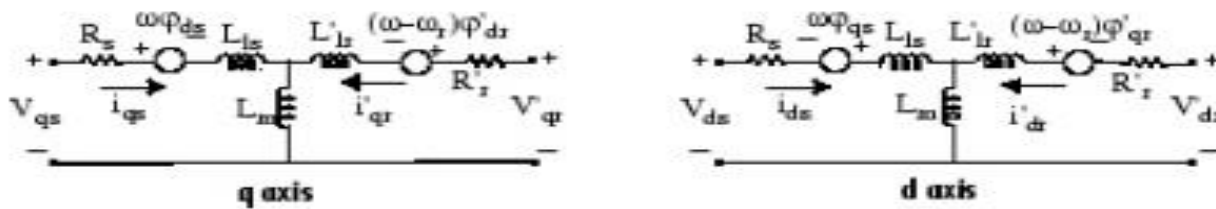
- If  $T_m$  is positive, the machine acts as a motor.
- If  $T_m$  is negative, the machine acts as a generator.

The electrical part of the machine is represented by a fourth-order (or sixth-order for the double squirrel-cage machine) state-space model, and the mechanical part by a second-order system. All electrical variables and parameters are referred to the stator, indicated by the prime signs in the following machine equations. All stator and rotor quantities are in the arbitrary two-axis reference frame (dq frame). The subscripts used are defined in this table.

Subscript	Definition
d	d axis quantity
q	q axis quantity
r	Rotor quantity (wound-rotor or single-cage)
r1	Cage 1 rotor quantity (double-cage)
r2	Cage 2 rotor quantity (double-cage)
s	Stator quantity
l	Leakage inductance
m	Magnetizing inductance



Electrical System of the Wound-Rotor or Squirrel-Cage Machine



$$V_{qs} = R_s i_{qs} + \frac{d}{dt} \phi_{qs} + \omega \phi_{ds}$$

$$V_{ds} = R_s i_{ds} + \frac{d}{dt} \phi_{ds} - \omega \phi_{qs}$$

$$V'_{qr} = R'_r i'_{qr} + \frac{d}{dt} \phi'_{qr} + (\omega - \omega_r) \phi'_{dr}$$

$$V'_{dr} = R'_r i'_{dr} + \frac{d}{dt} \phi'_{dr} - (\omega - \omega_r) \phi'_{qr}$$

$$T_e = 1.5 p (\phi_{ds} i_{qs} - \phi_{qs} i_{ds})$$

$$\phi_{qs} = L_s i_{qs} + L_m i'_{qr}$$

$$\phi_{ds} = L_s i_{ds} + L_m i'_{dr}$$

$$\phi'_{qr} = L'_r i'_{qr} + L_m i_{qs}$$

$$\phi'_{dr} = L'_r i'_{dr} + L_m i_{ds}$$

$$L_s = L_{ls} + L_m$$

$$L'_r = L'_{lr} + L_m$$

Mechanical System

$$\frac{d}{dt} \omega_m = \frac{1}{2H} (T_e - F \omega_m - T_m)$$

$$\frac{d}{dt} \theta_m = \omega_m$$

The Asynchronous Machine block parameters are defined as follows (all quantities are referred to the stator).

Parameters Common to All Models	Definition
$R_s, L_{ls}$	Stator resistance and leakage inductance
$L_m$	Magnetizing inductance
$L_s$	Total stator inductance
$V_{qs}, i_{qs}$	q axis stator voltage and current
$V_{ds}, i_{ds}$	d axis stator voltage and current
$\phi_{qs}, \phi_{ds}$	Stator q and d axis fluxes
$\omega_m$	Angular velocity of the rotor
$\theta_m$	Rotor angular position
$p$	Number of pole pairs
$\omega_r$	Electrical angular velocity ( $\omega_m \times p$ )
$\theta_r$	Electrical rotor angular position ( $\theta_m \times p$ )
$T_e$	Electromagnetic torque
$T_m$	Shaft mechanical torque

Parameters Common to All Models	Definition
J	Combined rotor and load inertia coefficient. Set to infinite to simulate locked
H	Combined rotor and load inertia constant. Set to infinite to simulate locked ro
F	Combined rotor and load viscous friction coefficient

## Dialog Box and Parameters

You can choose between two Asynchronous Machine blocks to specify the electrical and mechanical parameters of the model, by using the pu Units dialog box or the SI dialog box. Both blocks are modeling the same asynchronous machine model.

### Configuration Tab



### Mechanical input

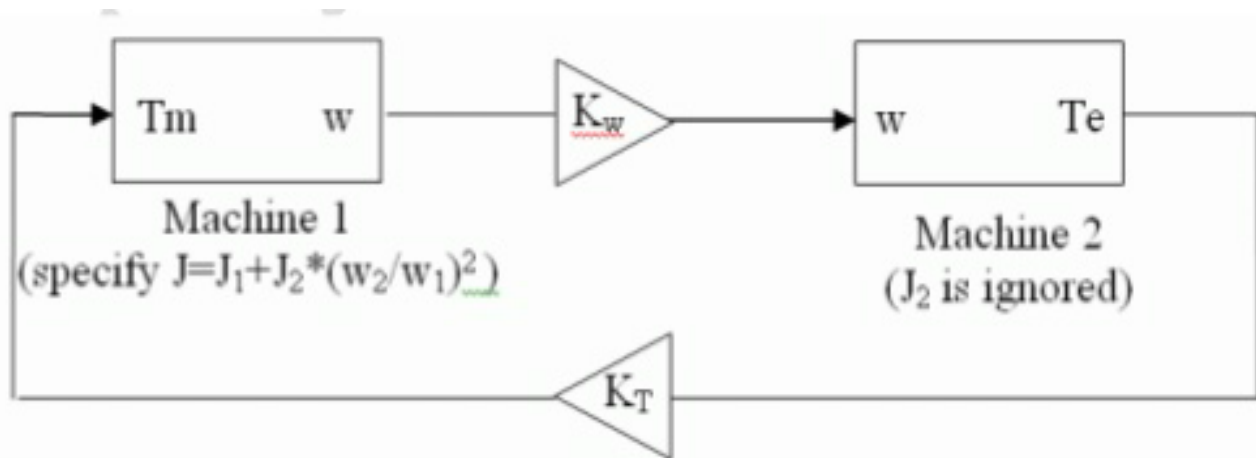
Select the torque applied to the shaft or the rotor speed as a Simulink® input of the block, or to represent the machine shaft by a Simscape™ rotational mechanical port.

Select Torque  $T_m$  to specify a torque input, in N.m or in pu, and change labeling of the block input to  $T_m$ . The machine speed is determined by the machine Inertia  $J$  (or inertia constant  $H$  for the pu machine) and by the difference between the applied mechanical torque  $T_m$  and the internal electromagnetic torque  $T_e$ . The sign convention for the mechanical torque is: when the speed is positive, a positive torque signal indicates motor mode and a negative signal indicates generator mode.

Select Speed  $w$  to specify a speed input, in rad/s or in pu, and change labeling of the block input to  $w$ . The machine speed is imposed and the mechanical part of the model (Inertia  $J$ ) is ignored. Using the speed as the mechanical input allows modeling a mechanical coupling between two machines.

The next figure indicates how to model a stiff shaft interconnection in a motor-generator set when friction torque is ignored in machine 2. The speed output of machine 1 (motor) is connected to the speed input of machine 2 (generator), while machine 2 electromagnetic torque output  $T_e$  is applied to

the mechanical torque input  $T_m$  of machine 1. The  $K_w$  factor takes into account speed units of both machines (pu or rad/s) and gear box ratio  $w_2/w_1$ . The  $K_T$  factor takes into account torque units of both machines (pu or N.m) and machine ratings. Also, as the inertia  $J_2$  is ignored in machine 2,  $J_2$  referred to machine 1 speed must be added to machine 1 inertia  $J_1$ .



### Reference frame

Specifies the reference frame that is used to convert input voltages (abc reference frame) to the dq reference frame, and output currents (dq reference frame) to the abc reference frame. You can choose among the following reference frame transformations:

Rotor (Park transformation)

Stationary (Clarke or  $\alpha\beta$  transformation)

Synchronous

The following relationships describe the abc-to-dq reference frame transformations applied to the Asynchronous Machine phase-to-phase voltages.

$$\begin{bmatrix} V_{qs} \\ V_{ds} \end{bmatrix} = \frac{1}{3} \begin{bmatrix} 2 \cos \theta & \cos \theta + \sqrt{3} \sin \theta \\ 2 \sin \theta & \sin \theta - \sqrt{3} \cos \theta \end{bmatrix} \begin{bmatrix} V_{abs} \\ V_{bcs} \end{bmatrix}$$

$$\begin{bmatrix} V'_{qr} \\ V'_{dr} \end{bmatrix} = \frac{1}{3} \begin{bmatrix} 2 \cos \beta & \cos \beta + \sqrt{3} \sin \beta \\ 2 \sin \beta & \sin \beta - \sqrt{3} \cos \beta \end{bmatrix} \begin{bmatrix} V'_{abr} \\ V'_{bcr} \end{bmatrix}$$

In the preceding equations,  $\Theta$  is the angular position of the reference frame, while  $\beta = \theta - \theta_r$  is the difference between the position of the reference frame and the position (electrical) of the rotor. Because the machine windings are connected in a three-wire Y configuration, there is no homopolar (0) component. This configuration also justifies that two line-to-line input voltages are used inside the model instead of three line-to-neutral voltages. The following relationships describe the dq-to-abc reference frame transformations applied to the Asynchronous Machine phase currents.

$$\begin{bmatrix} i_{as} \\ i_{bs} \end{bmatrix} = \begin{bmatrix} \cos\theta & \sin\theta \\ -\frac{\cos\theta + \sqrt{3}\sin\theta}{2} & -\frac{\sqrt{3}\cos\theta - \sin\theta}{2} \end{bmatrix} \begin{bmatrix} i_{qs} \\ i_{ds} \end{bmatrix}$$

$$\begin{bmatrix} i'_{ar} \\ i'_{br} \end{bmatrix} = \begin{bmatrix} \cos\beta & \sin\beta \\ -\frac{\cos\beta + \sqrt{3}\sin\beta}{2} & -\frac{\sqrt{3}\cos\beta - \sin\beta}{2} \end{bmatrix} \begin{bmatrix} i'_{qr} \\ i'_{dr} \end{bmatrix}$$

$$i_{cs} = -i_{as} - i_{bs}$$

$$i'_{cr} = -i'_{ar} - i'_{br}$$

The following table shows the values taken by  $\Theta$  and  $\beta$  in each reference frame ( $\Theta_e$  is the position of the synchronously rotating reference frame).

Reference Frame	$\Theta$
Rotor	$\Theta_r$
Stationary	0
Synchronous	$\Theta_e$

The choice of reference frame affects the waveforms of all dq variables. It also affects the simulation speed and in certain cases the accuracy of the results. The following guidelines are suggested in [1]:

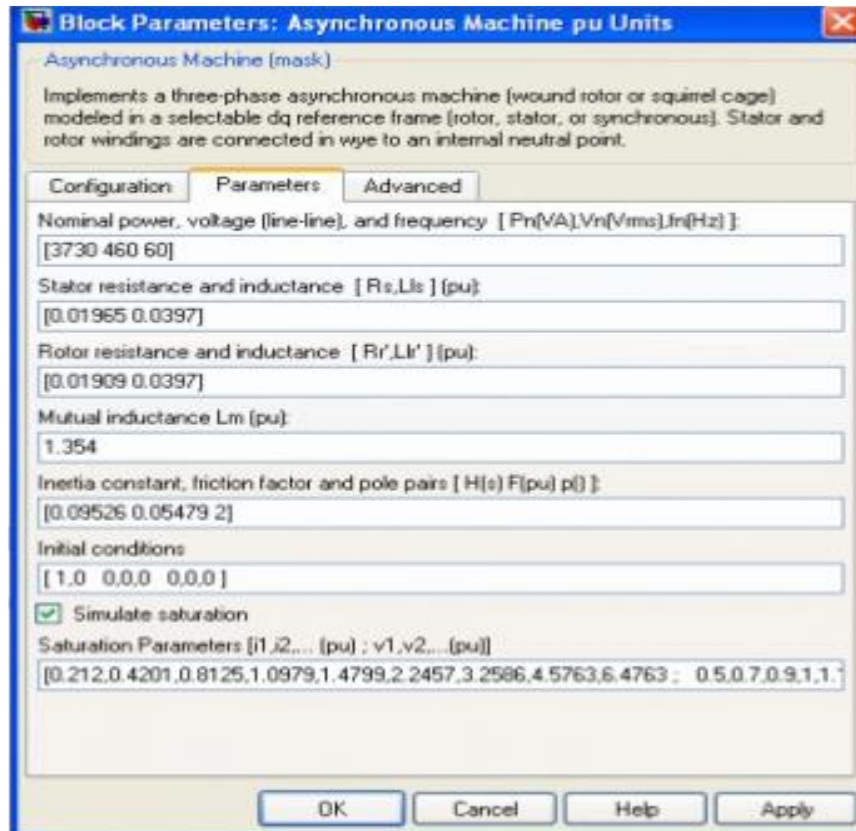
- Use the stationary reference frame if the stator voltages are either unbalanced or discontinuous and the rotor voltages are balanced (or 0).
- Use the rotor reference frame if the rotor voltages are either unbalanced or discontinuous and the stator voltages are balanced.
- Use either the stationary or synchronous reference frames if all voltages are balanced and continuous.

#### Use signal names to identify bus labels

When this check box is selected, the measurement output uses the signal names to identify the bus labels. Select this option for applications that require bus signal labels to have only alphanumeric characters.

When this check box is cleared, the measurement output uses the signal definition to identify the bus labels. The labels contain nonalphanumeric characters that are incompatible with some Simulink applications.

#### Parameters Tab



### Inputs and Outputs

- The stator terminals of the Asynchronous Machine block are identified by the letters A, B, and C. The rotor terminals are identified by the letters a, b, and c. The neutral connections of the stator and rotor windings are not available; three-wire Y connections are assumed.
- $T_m$
- The Simulink input of the block is the mechanical torque at the machine's shaft. When the input is a positive Simulink signal, the asynchronous machine behaves as a motor. When the input is a negative signal, the asynchronous machine behaves as a generator.
- When you use the SI parameters mask, the input is a signal in N.m, otherwise it is in pu.
- $\omega$
- The alternative block input (depending on the value of the Mechanical input parameter) is the machine speed. When you use the SI parameters mask, the input is a signal in rad/s or in pu.
- $m$
- The Simulink output of the block is a vector containing measurement signals. You can demultiplex these signals by using the Bus Selector block provided in the Simulink library. Depending on the type of mask that you use, the units are in SI or in pu. The cage 2 rotor signals return null signal when the Rotor type parameter on the Configuration tab is set to Wound or Squirrel-cage.

Name	Definition
iar	Rotor current ir_a
ibr	Rotor current ir_b
icr	Rotor current ir_c

Name	Definition
iqr	Rotor current iq
idr	Rotor current id
phiqr	Rotor flux phir_q
phidr	Rotor flux phir_d
vqr	Rotor voltage Vr_q
vdr	Rotor voltage Vr_d
iar2	Cage 2 rotor current ir_a
ibr2	Cage 2 rotor current ir_b
icr2	Cage 2 rotor current ir_c
iqr2	Cage 2 rotor current iq
idr2	Cage 2 rotor current id
phiqr2	Cage 2 rotor flux phir_q
phidr2	Cage 2 rotor flux phir_d
ias	Stator current is_a
ibs	Stator current is_b
ics	Stator current is_c
iqs	Stator current is_q
ids	Stator current is_d
phiqs	Stator flux phis_q
phids	Stator flux phis_d
vqs	Stator voltage vs_q
vds	Stator voltage vs_d
w	Rotor speed
Te	Electromagnetic torque Te
theta	Rotor angle thetam

### Limitations

- The Asynchronous Machine block does not include a representation of the saturation of leakage fluxes. You must be careful when you connect ideal sources to the machine's stator. If you choose to supply the stator via a three-phase Y-connected infinite voltage source, you must use three sources connected in Y. However, if you choose to simulate a delta source connection, you must use only two sources connected in series.

**Result:** We have successfully modeled the synchronous machine.

**Precautions:**

- ✓ Do not temper with the settings of the software
- ✓ Study all the observations very carefully.

## EXPERIMENT NO: 4

**AIM:** - Simulate simple circuit using circuit Maker

**Apparatus Required:** MATLAB Software

### **THEORY:**

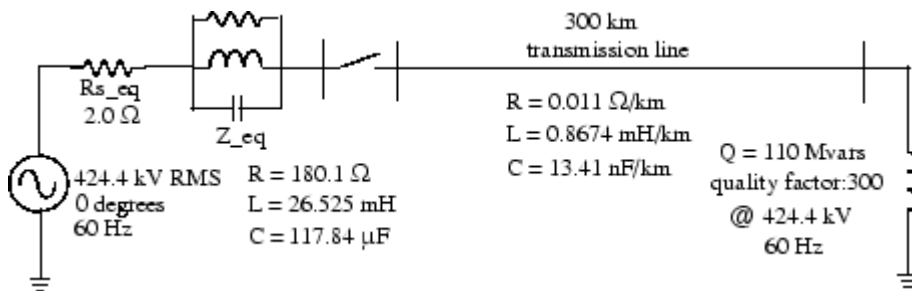
#### **Introduction**

Simscape™ Power Systems™ Specialized Technology allows you to build and simulate electrical circuits containing linear and nonlinear elements.

In this section you

- Explore the Simscape Power Systems Specialized Technology library
- Learn how to build a simple circuit from the Simscape Power Systems Specialized Technology library
- Interconnect Simulink® blocks with your circuit

The circuit below represents an equivalent power system feeding a 300 km transmission line. The line is compensated by a shunt inductor at its receiving end. A circuit breaker allows energizing and de-energizing of the line. To simplify matters, only one of the three phases is represented. The parameters shown in the figure are typical of a 735 kV power system.



#### Building the Electrical Circuit with the Simscape Power Systems Specialized Technology Library

The graphical user interface makes use of the Simulink functionality to interconnect various electrical components. The electrical components are grouped in the Simscape Power Systems Specialized Technology library.

1. Open the Simscape Power Systems main library by entering the following command at the MATLAB® prompt.

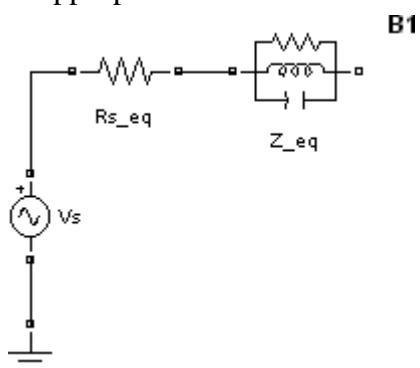
[simscapepowersystems\\_ST](#)

2. From the File menu of the [simscapepowersystems\\_ST](#) window, open a new window to contain your first circuit and save it as circuit1.
3. Open the Fundamental Blocks > Electrical Sources library and copy the [AC Voltage Source](#) block into the circuit1 window.
4. Open the AC Voltage Source dialog box by double-clicking the icon and enter the Amplitude, Phase, and Frequency parameters according to the values shown in [Circuit to Be Modeled](#).



Note that the amplitude to be specified for a sinusoidal source is its peak value ( $424.4e3*\sqrt{2}$  volts in this case).

5. Change the name of this block from AC Voltage Source to Vs.
6. Copy the [Parallel RLC Branch](#) block, which can be found in the Fundamental Blocks/Elements library, set its parameters as shown in [Circuit to Be Modeled](#), and name it Z\_eq.
7. The resistance Rs\_eq of the circuit can be obtained from the Parallel RLC Branch block. Duplicate the Parallel RLC Branch block, which is already in your circuit1 window. Select R for the Branch Type parameter and set the R parameter according to [Circuit to Be Modeled](#).  
Once the dialog box is closed, notice that the L and C components have disappeared so that the icon now shows a single resistor.
8. Name this block Rs\_eq.
9. Resize the various components and interconnect blocks by dragging lines from outputs to inputs of appropriate blocks.



10. To complete the circuit of [Circuit to Be Modeled](#), you need to add a transmission line and a shunt reactor. You add the circuit breaker later in [Simulating Transients](#).

The model of a line with uniformly distributed R, L, and C parameters normally consists of a delay equal to the wave propagation time along the line. This model cannot be simulated as a linear system because a delay corresponds to an infinite number of states. However, a good approximation of the line with a finite number of states can be obtained by cascading several PI circuits, each representing a small section of the line.

A PI section consists of a series R-L branch and two shunt C branches. The model accuracy depends on the number of PI sections used for the model. Copy the [PI Section Line](#) block from the Elements library into the **circuit1** window, set its parameters as shown in [Circuit to Be Modeled](#), and specify one line section.

11. The shunt reactor is modeled by a resistor in series with an inductor. You could use a [Series RLC Branch](#) block to model the shunt reactor, but then you would have to manually calculate and set the R and L values from the quality factor and reactive power specified in [Circuit to Be Modeled](#).

Therefore, you might find it more convenient to use a [Series RLC Load](#) block that allows you to specify directly the active and reactive powers absorbed by the shunt reactor.

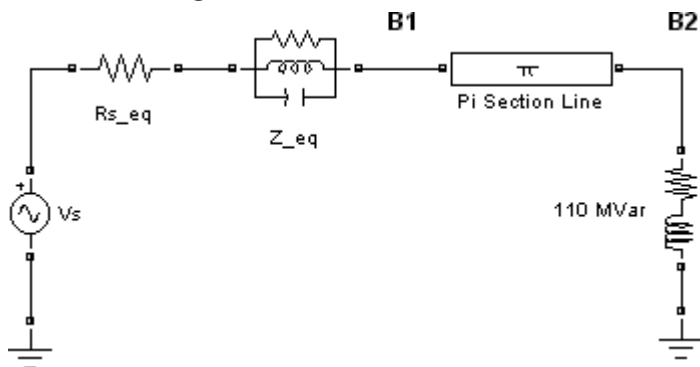
Copy the Series RLC Load block, which can be found in the Fundamental Blocks/Elements library. Name this block 110 Mvar. Set its parameters as follows:

<b>Vn</b>	424.4e3 V
<b>fn</b>	60 Hz
<b>P</b>	110e6/300 W (quality factor = 300)
<b>QL</b>	110e6 vars

Qc

0

Note that, as no reactive capacitive power is specified, the capacitor disappears on the block icon when the dialog box is closed. Interconnect the new blocks as shown.

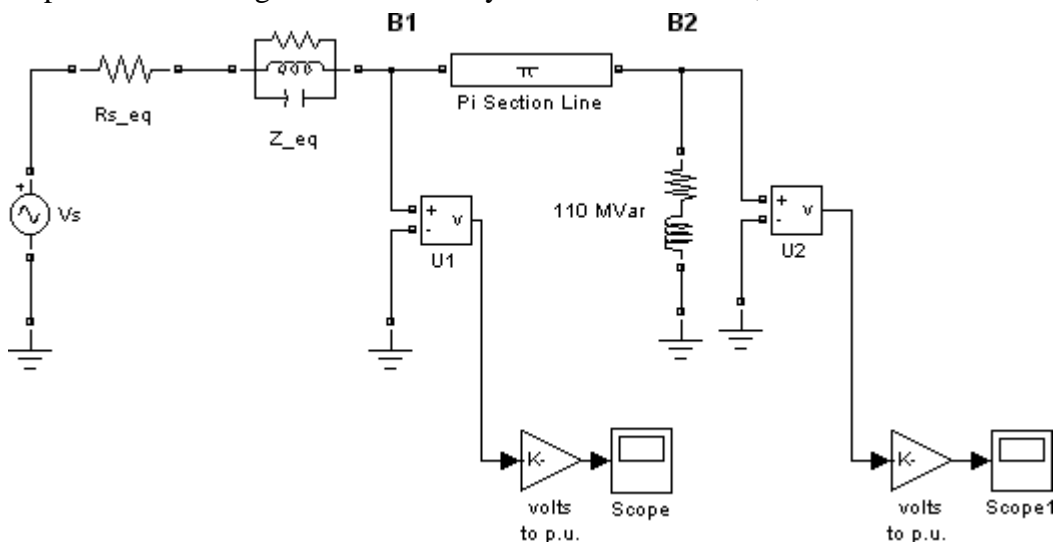


12. You need a Voltage Measurement block to measure the voltage at node B1. This block is found in the Fundamental Blocks/Measurements library. Copy it and name it U1. Connect its positive input to the node B1 and its negative input to a new [Ground](#) block.
13. To observe the voltage measured by the Voltage Measurement block named U1, a display system is needed. This can be any device found in the Simulink Sinks library.

Open the Sinks library and copy the [Scope](#) block into your **circuit1** window. If the scope were connected directly at the output of the voltage measurement, it would display the voltage in volts. However, electrical engineers in power systems are used to working with normalized quantities (per unit system). The voltage is normalized by dividing the value in volts by a base voltage corresponding to the peak value of the system nominal voltage. In this case the scaling factor  $K$  is

$$K = \frac{3}{1424.4 \times 10} G2$$

14. Copy a [Gain](#) block from the Simulink library and set its gain as above. Connect its output to the Scope block and connect the output of the Voltage Measurement block to the Gain block. Duplicate this voltage measurement system at the node B2, as shown below.



15. Add a Powergui block to your model. The purpose of this block is discussed in [Using the Powergui Block to Simulate Simscape Power Systems Models](#).
16. Select **Simulation > Run**.
17. Open the Scope blocks and observe the voltages at nodes B1 and B2.

18. While the simulation is running, open the Vs block dialog box and modify the amplitude. Observe the effect on the two scopes. You can also modify the frequency and the phase. You can zoom in on the waveforms in the scope windows by drawing a box around the region of interest with the left mouse button.

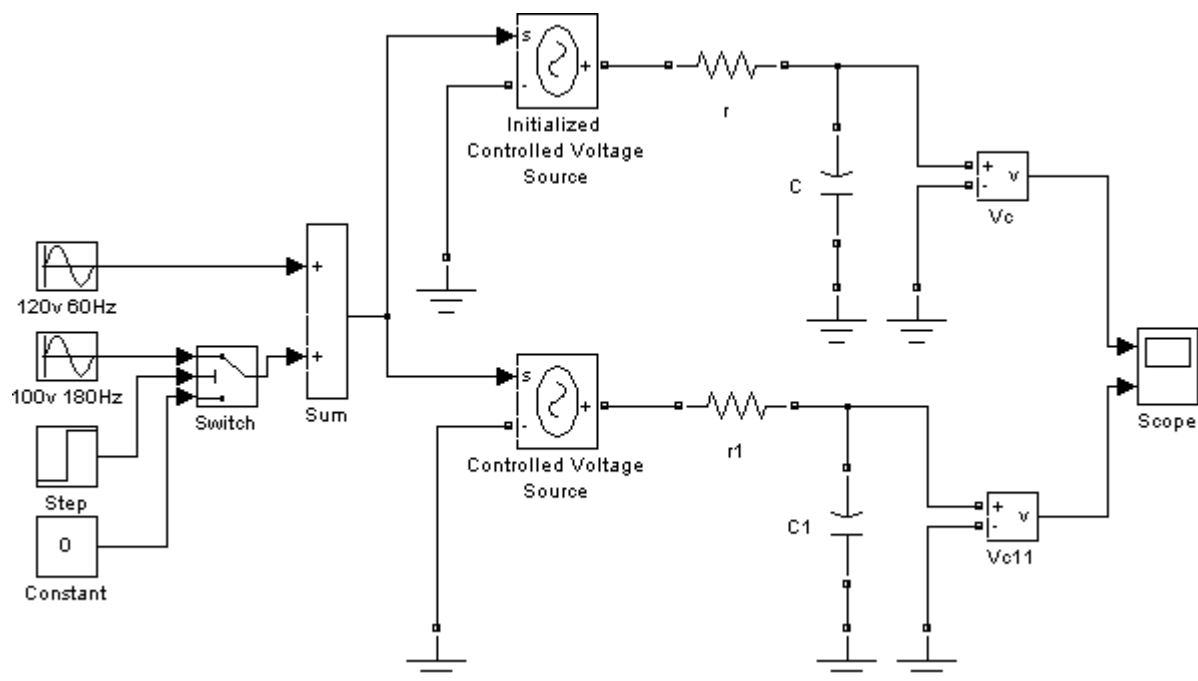
To simulate this circuit, the default integration algorithm (ode45) was used. However, for most Simscape Power Systems applications, your circuits contain switches and other nonlinear models. In such a case, you must specify a different integration algorithm. This is discussed in Simulating Transients, where a circuit breaker is added to your circuit.

### Interfacing the Electrical Circuit with Other Simulink Blocks

The Voltage Measurement block acts as an interface between the Simscape Power Systems blocks and the Simulink blocks. For the system shown above, you implemented such an interface from the electrical system to the Simulink system. The Voltage Measurement block converts the measured voltages into Simulink signals.

Similarly, the [Current Measurement](#) block from the Fundamental Blocks/Measurements library can be used to convert any measured current into a Simulink signal.

You can also interface from Simulink blocks to the electrical system. For example, you can use the [Controlled Voltage Source](#) block to inject a voltage in an electrical circuit, as shown in the following figure.



### Measuring Voltages and Currents

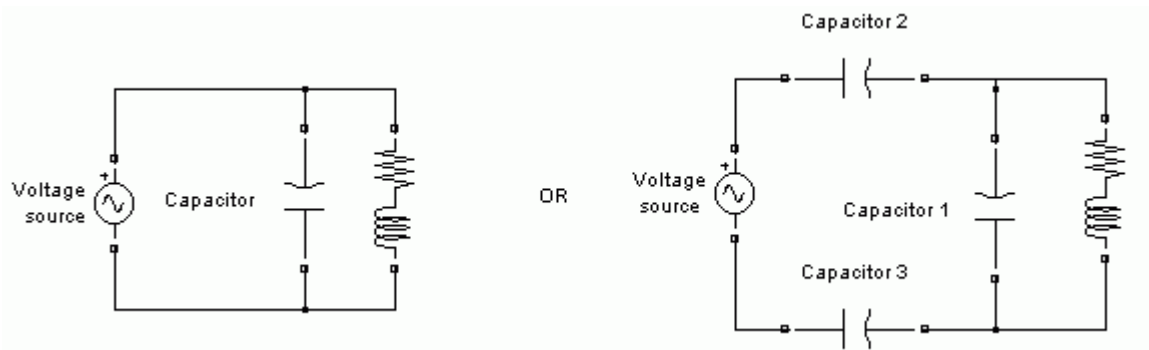
When you measure a current using a [Current Measurement](#) block, the positive direction of current is indicated on the block icon (positive current flowing from + terminal to – terminal). Similarly, when you measure a voltage using a [Voltage Measurement](#) block, the measured voltage is the voltage of the + terminal with respect to the – terminal.

### Basic Principles of Connecting Capacitors and Inductors

You have to pay particular attention when you connect capacitor elements together with voltage sources, or inductor elements in series with current sources. When you start the simulation, the

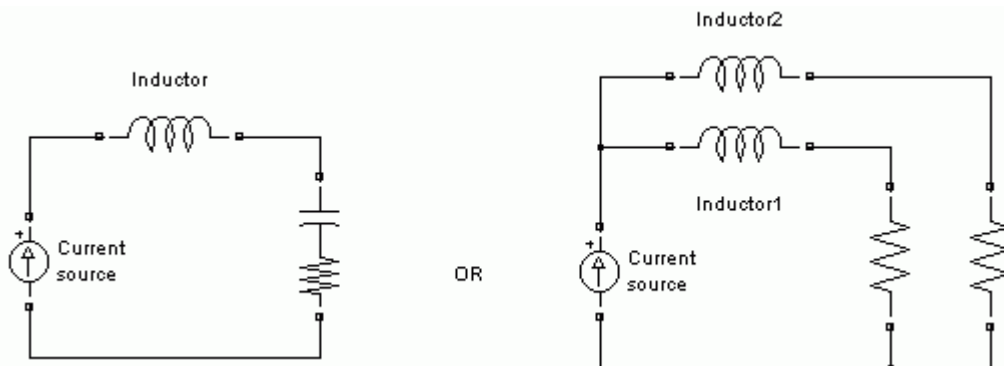
software displays an error message if one of the following two connection errors are present in your diagram:

1. You have connected a voltage source in parallel with a capacitor, or a series of capacitor elements in series, like in the two examples below.



To fix this problem, you can add a small resistance in series between the voltage source and the capacitors.

2. You have connected a current source in series with an inductor, or a series of inductors connected in parallel, like in the example below.



To fix this problem, you can add a large resistance in parallel with the inductor.

### Using the Powergui Block to Simulate Simscape Power Systems Models

The Powergui block is necessary for simulation of any Simulink model containing Simscape Power Systems blocks. It is used to store the equivalent Simulink circuit that represents the state-space equations of the Simscape Power Systems blocks.

**Results:** We have successfully simulated simple circuits.

#### Precautions:

- ✓ Do not temper with the settings of software.
- ✓ Study all the observations very carefully.

## EXPERIMENT NO: 5

**AIM:** - (a) Modeling of Synchronous Machine with PSS  
(b) Simulation of Synchronous Machine with FACTS device.

**Apparatus Required:** MATLAB Software

### **THEORY:**

UPFC (Detailed Model)

### **Model Description**

A Unified Power Flow Controller (UPFC) is used to control the power flow in a 500 kV transmission system. The UPFC located at the left end of the 75-km line L2, between the 500 kV buses B1 and B2, is used to control the active and reactive powers flowing through bus B2 while controlling voltage at bus B1. It consists of two 100-MVA, three-level, 48-pulse GTO-based converters, one connected in shunt at bus B1 and one connected in series between buses B1 and B2. The shunt and series converters can exchange power through a DC bus. The series converter can inject a maximum of 10% of nominal line-to-ground voltage (28.87 kV) in series with line L2.

This pair of converters can be operated in three modes:

- **Unified Power Flow Controller (UPFC)** mode, when the shunt and series converters are interconnected through the DC bus. When the disconnect switches between the DC buses of the shunt and series converter are opened, two additional modes are available:
- Shunt converter operating as a **Static Synchronous Compensator (STATCOM)** controlling voltage at bus B1
- Series converter operating as a **Static Synchronous Series Capacitor (SSSC)** controlling injected voltage, while keeping injected voltage in quadrature with current.

The mode of operation as well as the reference voltage and reference power values can be changed by means of the "UPFC GUI" block.

The principle of operation of the harmonic neutralized converters is explained in another example entitled "Three-phase 48-pulse GTO converter". This power\_48pulsegtoconverter model is accessible in the Power Electronics Models library of examples. When the two converters are operated in UPFC mode, the shunt converter operates as a STATCOM. It controls the bus B1 voltage by controlling the absorbed or generated reactive power while also allowing active power transfer to the series converter through the DC bus. The reactive power variation is obtained by varying the DC bus voltage. The four three-level shunt converters operate at a constant conduction angle ( $\text{Sigma} = 180 - 7.5 = 172.5$  degrees), thus generating a quasi-sinusoidal 48-step voltage waveform. The first significant harmonics are the 47th and the 49th.

When operating in UPFC mode, the magnitude of the series injected voltage is varied by varying the Sigma conduction angle, therefore generating higher harmonic contents than the shunt converter. As illustrated in this example, when the series converter operates in SSSC mode it generates a "true" 48-pulse waveform.

The natural power flow through bus B2 when zero voltage is generated by the series converter (zero voltage on converter side of the four converter transformers) is  $P = +870$  MW and  $Q = -70$  Mvar. In UPFC mode, both the magnitude and phase angle and the series injected voltage can be varied, thus allowing control of P and Q. The UPFC controllable region is obtained by keeping the injected

voltage to its maximum value (0.1 pu) and varying its phase angle from zero to 360 degrees. To see the resulting P-Q trajectory, double click the "Show UPFC Controllable Region". Any point located inside the PQ elliptic region can be obtained in UPFC mode.

## Demonstration

### 1. Power control in UPFC mode

Open the UPFC GUI block menu. The GUI allows you to choose the operation mode (UPFC, STATCOM or SSSC) as well as the Pref/Qref reference powers and/or Vref reference voltage settings. Also, in order to observe the dynamic response of the control system, the GUI allows you to specify a step change of any reference value at a specific time.

Make sure that the operation mode is set to "UPFC (Power Flow Control)". The reference active and reactive powers are specified in the last two lines of the GUI menu. Initially, Pref= +8.7 pu/100MVA (+870 MW) and Qref=-0.6 pu/100MVA (-60 Mvar). At t=0.25 sec Pref is changed to +10 pu (+1000MW). Then, at t=0.5 sec, Qref is changed to +0.7 pu (+70 Mvar). The reference voltage of the shunt converter (specified in the 2nd line of the GUI) will be kept constant at Vref=1 pu during the whole simulation (Step Time=0.3\*100> Simulation stop time (0.8 sec). When the UPFC is in power control mode, the changes in STATCOM reference reactive power and in SSSC injected voltage (specified respectively in 1st and 3rd line of the GUI) as are not used.

Run the simulation for 0.8 sec. Open the "Show Scopes" subsystem. Observe on traces 1 and 2 of the UPFC scope the variations of P and Q. After a transient period lasting approximately 0.15 sec, the steady state is reached (P=+8.7 pu; Q=-0.6 pu). Then P and Q are ramped to the new settings (P=+10 pu Q=+0.7 pu). Observe on traces 3 and 4 the resulting changes in P Q on the three transmission lines. The performance of the shunt and series converters can be observed respectively on the STATCOM and SSSC scopes. If you zoom on the first trace of the STATCOM scope, you can observe the 48-step voltage waveform Vs generated on the secondary side of the shunt converter transformers (yellow trace) superimposed with the primary voltage Vp (magenta) and the primary current Ip (cyan). The dc bus voltage (trace 2) varies in the 19kV-21kV range. If you zoom on the first trace of the SSSC scope, you can observe the injected voltage waveforms Vinj measured between buses B1 and B2.

### 2. Var control in STATCOM mode

In the GUI block menu, change the operation mode to "STATCOM (Var Control)". Make sure that the STATCOM references values (1st line of parameters, [T1 T2 Q1 Q2]) are set to [0.3 0.5 +0.8 -0.8]. In this mode, the STATCOM is operated as a variable source of reactive power. Initially, Q is set to zero, then at T1=0.3 sec Q is increased to +0.8 pu (STATCOM absorbing reactive power) and at T2=0.5 sec, Q is reversed to -0.8 pu (STATCOM generating reactive power).

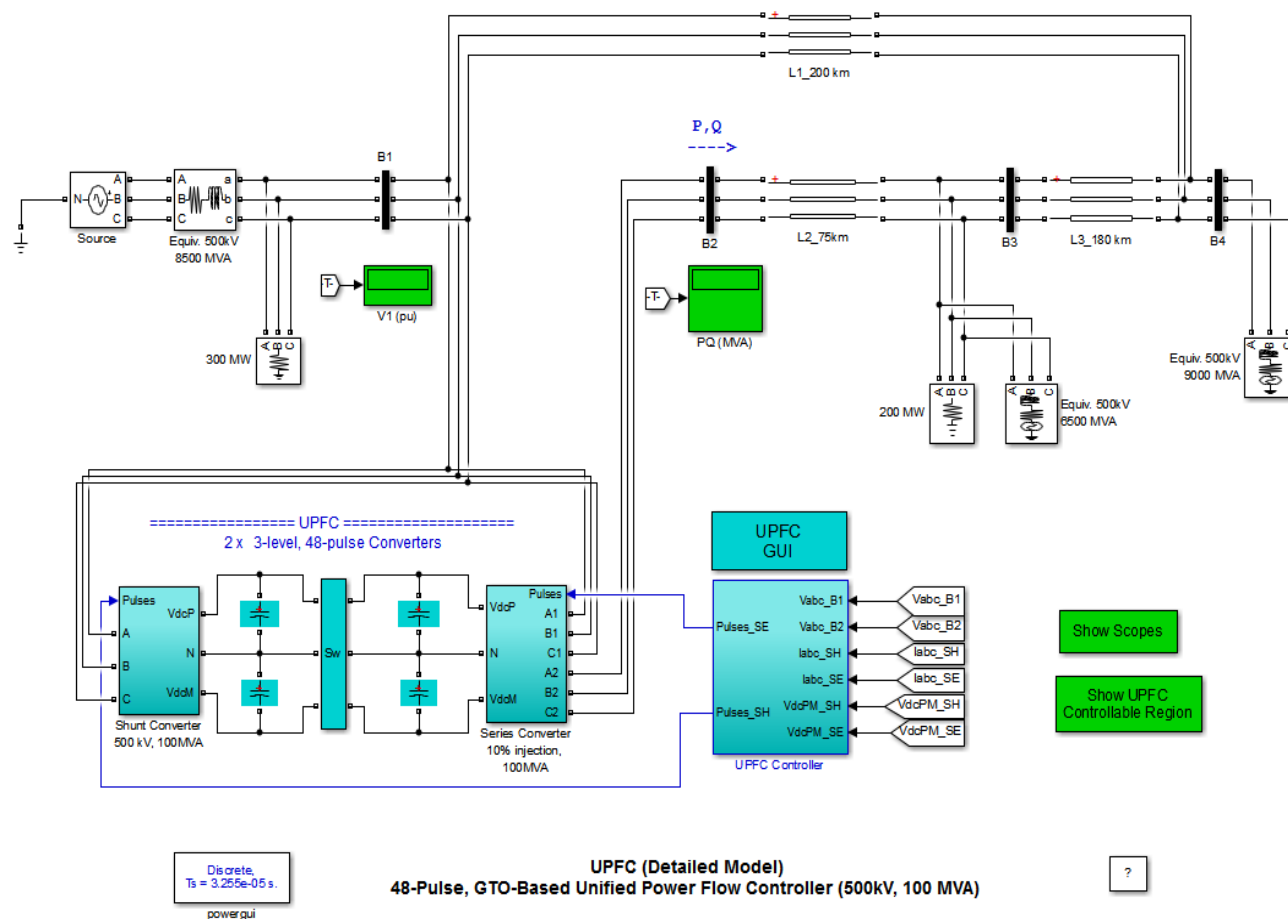
Run the simulation and observe on the STATCOM scope the dynamic response of the STATCOM. Zoom on the first trace around t=0.5 sec when Q is changed from +0.8 pu to -0.8 pu. When Q=+0.8 pu, the current flowing into the STATCOM (cyan trace) is lagging voltage (magenta trace), indicating that STATCOM is absorbing reactive power. When Qref is changed from +0.8 to -0.8, the current phase shift with respect to voltage changes from 90 degrees lagging to 90 degrees leading within one cycle. This control of reactive power is obtained by varying the magnitude of the secondary voltage Vs generated by the shunt converter while keeping it in phase with the bus B1 voltage Vp. This change of Vs magnitude is performed by controlling the dc bus voltage. When Q is changing from +0.8 pu to -0.8 pu, Vdc (trace 3) increases from 17.5 kV to 21 kV.

### 3. Series voltage injection in SSSC mode

In the GUI block menu change the operation mode to "SSSC (Voltage injection)". Make sure that the SSSC references values (3rd line of parameters) [Vinj\_InitialVinj\_FinalStepTime] ) are set to [0.0 0.08 0.3 ]. The initial voltage is set to 0 pu, then at t=0.3 sec it will be ramped to 0.8 pu.

Run the simulation and observe on the SSSC scope the impact of injected voltage on P and Q flowing in the 3 transmission lines. Contrary to the UPFC mode, in SSSC mode the series inverter operates with a constant conduction angle ( $\sigma = 172.5$  degrees). The magnitude of the injected voltage is controlled by varying the dc voltage which is proportional to  $V_{inj}$  (3rd trace). Also, observe the waveforms of injected voltages (1st trace) and currents flowing through the SSSC (2nd trace). Voltages and currents stay in quadrature so that the SSSC operates as a variable inductance or capacitance.

Circuit Diagram :



**Results:** We have successfully demonstrated Modeled Synchronous Machine with PSS and simulated Synchronous Machine with FACTS device.

**Precautions:**

- ✓ Do not temper with the settings of software.
- ✓ Study all the observations very carefully.

### EXPERIMENT NO: 6

**AIM:** - FACTS Controller designs with FACT devices for SMIB system.

**Apparatus Required:** MATLAB Software

### **THEORY:**

A 100-Mvar STATCOM regulates voltage on a three-bus 500-kV system. The 48-pulse STATCOM uses a Voltage-Sourced Converter (VSC) built of four 12-pulse three-level GTO inverters. Look inside the STATCOM block to see how the VSC inverter is built. The four sets of three-phase voltages obtained at the output of the four three-level inverters are applied to the secondary windings of four phase-shifting transformers (-15 deg., -7.5 deg., 7.5 deg., +7.5 deg. phase shifts). The fundamental components of voltages obtained on the 500 kV side of the transformers are added in phase by the serial connection of primary windings. Please refer to the "power\_48pulsegtoconverter" example to get details on the operation of the VSC.

During steady-state operation the STATCOM control system keeps the fundamental component of the VSC voltage in phase with the system voltage. If the voltage generated by the VSC is higher (or lower) than the system voltage, the STATCOM generates (or absorbs) reactive power. The amount of reactive power depends on the VSC voltage magnitude and on the transformer leakage reactances. The fundamental component of VSC voltage is controlled by varying the DC bus voltage. In order to vary the DC voltage, and therefore the reactive power, the VSC voltage angle (alpha) which is normally kept close to zero is temporarily phase shifted. This VSC voltage lag or lead produces a temporary flow of active power which results in an increase or decrease of capacitor voltages.

One of the three voltage sources used in the 500 kV system equivalents can be varied in order to observe the STATCOM dynamic response to changes in system voltage. Open the "Programmable Voltage Source" menu and look at the sequence of voltage steps which are programmed.

### **SIMULATION**

#### **DYNAMIC RESPONSE OF THE STATCOM**

Run the simulation and observe waveforms on the STATCOM scope block. The STATCOM is in voltage control mode and its reference voltage is set to  $V_{ref}=1.0$  pu. The voltage droop of the regulator is 0.03 pu/100 VA. Therefore when the STATCOM operating point changes from fully capacitive (+100 Mvar) to fully inductive (-100 Mvar) the STATCOM voltage varies between  $1-0.03=0.97$  pu and  $1+0.03=1.03$  pu.

Initially the programmable voltage source is set at 1.0491 pu, resulting in a 1.0 pu voltage at SVC terminals when the STATCOM is out of service. As the reference voltage  $V_{ref}$  is set to 1.0 pu, the STATCOM is initially floating (zero current). The DC voltage is 19.3 kV. At  $t=0.1$ s, voltage is suddenly decreased by 4.5 % (0.955 pu of nominal voltage). The SVC reacts by generating reactive power ( $Q=+70$  Mvar) in order to keep voltage at 0.979 pu. The 95% settling time is approximately 47 ms. At this point the DC voltage has increased to 20.4 kV. Then, at  $t=0.2$  s the source voltage is increased to 1.045 pu of its nominal value. The SVC reacts by changing its operating point from capacitive to inductive in order to keep voltage at 1.021 pu. At this point the STATCOM absorbs 72 Mvar and the DC voltage has been lowered to 18.2 kV. Observe on the first trace showing the STATCOM primary voltage and current that the current is changing from capacitive to inductive in approximately one cycle. Finally, at  $t=0.3$  s the source voltage is set back to its nominal value and the STATCOM operating point comes back to zero Mvar.



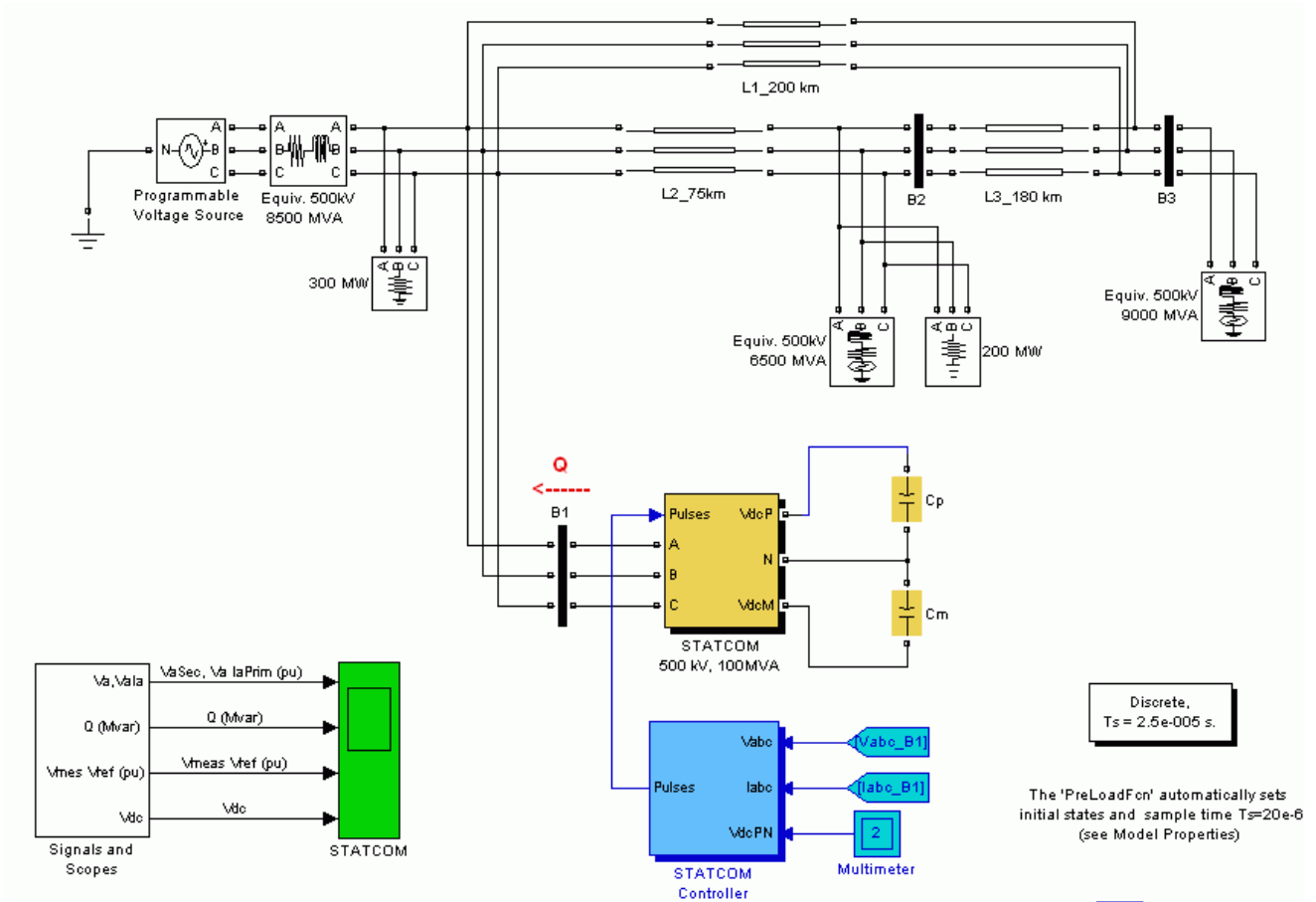
If you look inside the "Signals and Scopes" subsystem you will have access to other control signals. Notice the transient changes on alpha angle when the DC voltage is increased or decreased in order to vary reactive power. The steady state value of alpha (0.5 degrees) is the phase shift required to maintain a small active power flow compensating transformer and converter losses.

### **HOW TO REGENERATE INITIAL CONDITIONS**

The initial states required to start this example in steady state have been saved in the "power\_statcom\_gto48p.mat" file. When you open this example, the InitFcn callback (in the Model Properties/Callbacks) automatically loads into your workspace the contents of this .mat file ("xInitial" variable).

If you modify this model, or change parameter values of power components, the initial conditions stored in the "xInitial" variable will no longer be valid and Simulink will issue an error message. To regenerate the initial conditions for your modified model, follow the steps listed below:

1. In the Simulation/Configuration Parameters/Data Import/Export Parameters menu, uncheck the "Initial state" parameter and check the "Final states" parameter.
2. In the Programmable Voltage Source menu, disable the source voltage steps by setting the "Time variation of " parameter to "none".
3. Make sure that the Simulation Stop Time is 0.4 second. Note that in order to generate initial conditions coherent with the 60 Hz voltage source phase angles, the Stop Time must be an integer number of 60 Hz cycles.
4. Start simulation. When simulation is completed, verify that steady state has been reached by looking at waveforms displayed on the scope. The final states which have been saved in the "xFinal" structure with time can be used as initial states for future simulations. Executing the next two commands copies these final conditions in "xInitial" and saves this variable in a new file (myModel\_init.mat). \* >>xInitial=xFinal; \* >> save myModel\_initxInitial
5. In the File/Model Properties/Callbacks/InitFcn window, change the name of the initialization file from "power\_statcom\_gto48p.mat" to "myModel\_init.mat". Next time you open this model, the variable xInitial saved in the myModel\_init.mat file will be loaded in your workspace.
6. In the Simulation/Configuration Parameters menu, check "Initial state".
7. Start simulation and verify that your model starts in steady-state.
8. In the Programmable Voltage Source menu, set the "Time variation of" parameter back to "Amplitude".
9. Save your Model.



**Results:** We have successfully demonstrated FACT devices for SMIB systems.

**Precautions:**

- ✓ Do not temper with the settings of software.
- ✓ Study all the observations very carefully.