LIST OF EXPERIMENTS

1. Develop a Simulink model for transient stability analysis of transmission line.
2. Develop a Simulink model for transmission system protection using surge arrester.
3. Time and frequency domain analysis of single phase transmission system
4. Develop a Simulink model for single phase rectifier circuit.
5. Develop a Simulink model for Ideal Switching Device solution method to simulate a full wave rectifier using ideal diodes.
6. Effect of compensation on voltage profile in transmission line Using MATLAB-SIMULINK
7. Transmission system behavior before and after fault condition Using MATLAB-SIMULINK
8. Evaluate the value of voltages for a 4-BUS system using node equations in MATLAB
9. Develop a Simulink model for hybrid Stepper Motor Drive

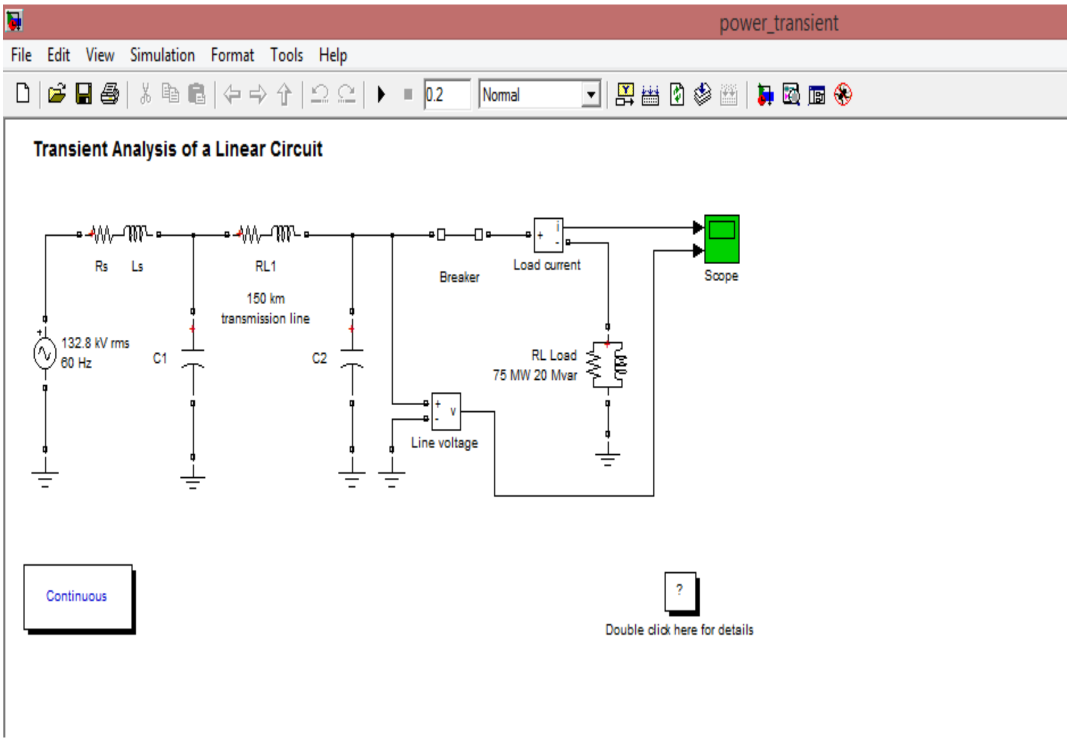
EXPERIMENT NO-1

AIM- Develop a Simulink model for transient stability analysis of transmission line.

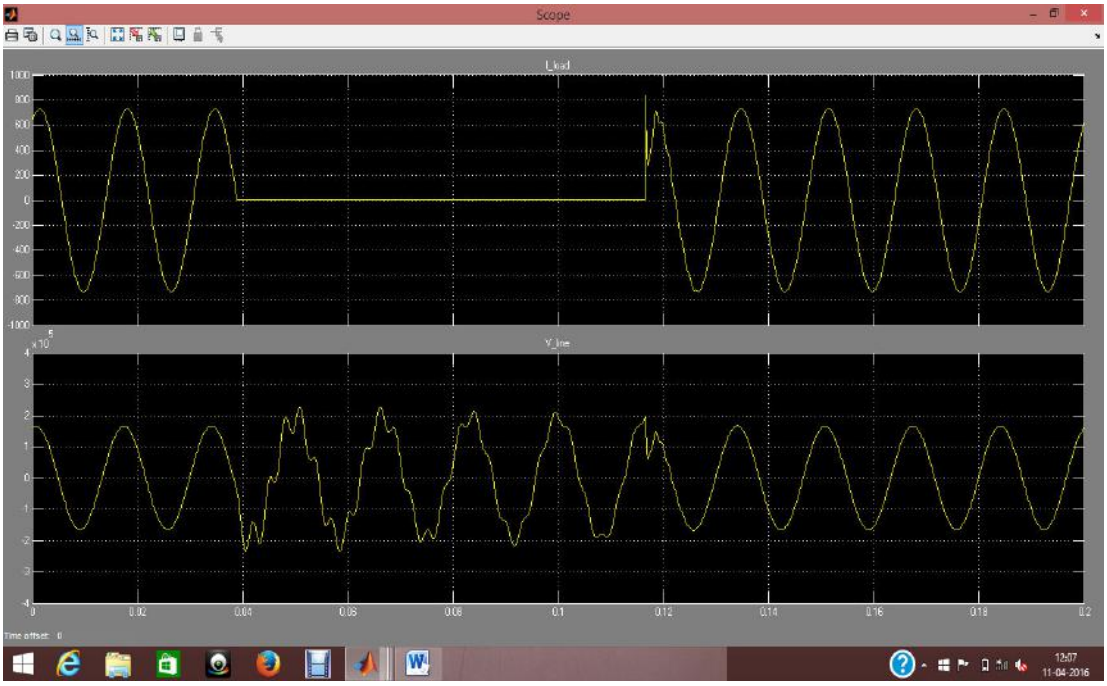
Circuit Description

This circuit is a simplified model of a 230 kV three-phase power system. Only one phase of the transmission system is represented. The equivalent source is modeled by a voltage source (230 kV rms/sqrt(3) or 187.8 kV peak, 60 Hz) in series with its internal impedance (RsLs) corresponding to a 3-phase 2000 MVA short circuit level and X/R = 10. (X = 230e3^2/2000e6 = 26.45 ohms or L = 0.0702 H, R = X/10 = 2.645 ohms). The source feeds a RL load through a 150 km transmission line. The line distributed parameters (R = 0.035ohm/km, L = 0.92 mH/km, C = 12.9 nF/km) are modeled by a single pi section (RL1 branch 5.2 ohm; 138 mH and two shunt capacitances C1 and C2 of 0.967 uF). The load (75 MW - 20 Mvar per phase) is modeled by a parallel RLC load block

Simulink Diagram



Results



Conclusion

EXPERIMENT NO-2

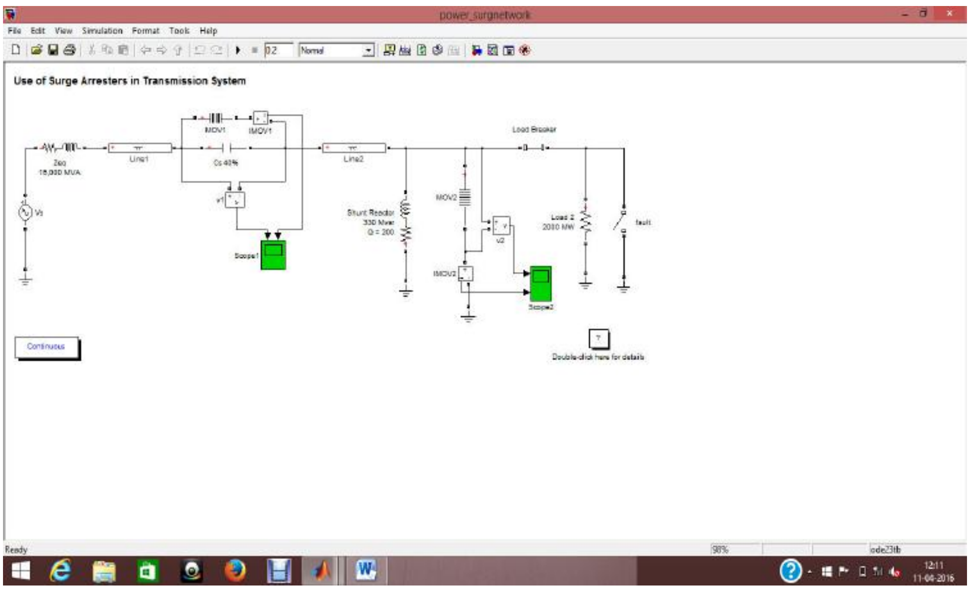
Aim –Develop a Simulink model for transmission system protection using surge arrester.

Circuit Description

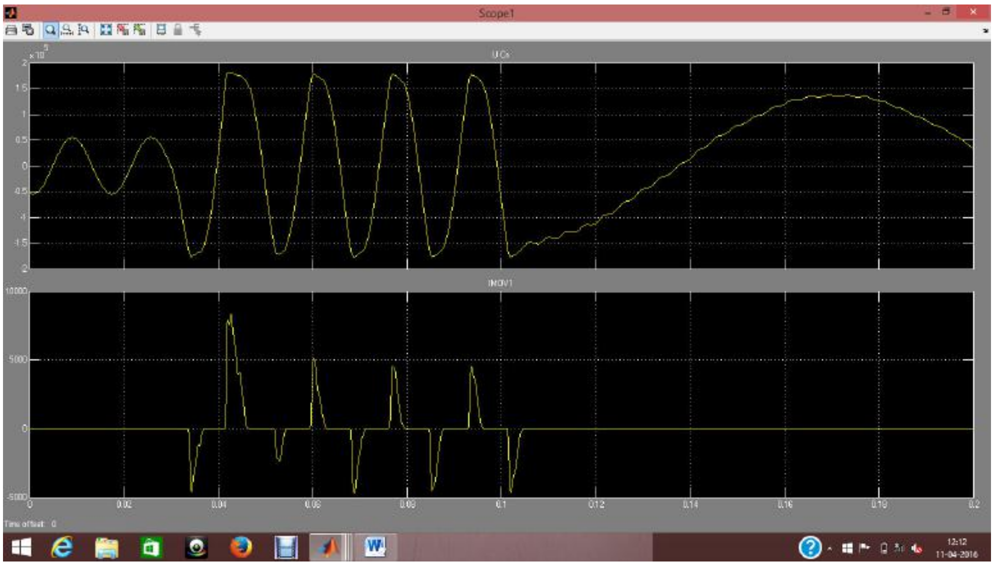
A 735 kV equivalent transmission systems feeds a load through a 200 km transmission line. The line is series compensated at the middle point and shunt compensated at its receiving end. A fault is applied at the load terminals. The fault is cleared by load breaker opening. For simplification purpose, only one phase of the transmission system is modeled. All parameters correspond to positive-sequence.The three-phase short circuit level of the transmission system is 15000 MVA. The line is 40% series compensated by a capacitor (26.2 ohms at 60 Hz) and shunt compensated by a 330 Mvar (110 M var/phase) inductor at the load end.

The series capacitor and shunt inductor are both protected by metal oxide varistors (MOV). The series capacitor varistor MOV1 consists of 30 columns protecting the capacitor at 2.5 times its rated voltage (rated voltage is obtained for a 2000 kA line rated current). The corresponding protection voltage (defined at 15 kA = 500 A per column) is 2.5*26.2 *2kA*sqrt(2) = 185 kV.The shunt inductor is protected by a 2-column arrester (MOV2) at 1.8 p.u. of nominal phase-to-ground voltage (424.4 kVrms). The corresponding protection voltage (defined at 1 kA or 500A /column) is 1.8*424.4*sqrt(2) = 1080 kV

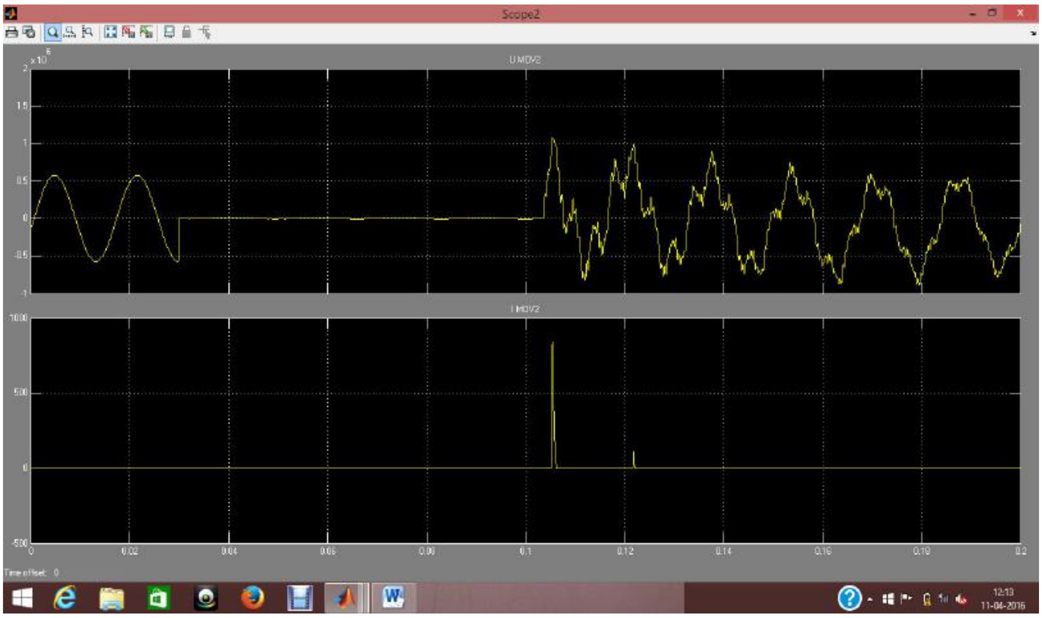
Simulink Model



Results-Scope1



Scope 2



Conclusion

EXPERIMENT NO-3

AIM-Time and frequency domain analysis of single phase transmission system.

Circuit Description

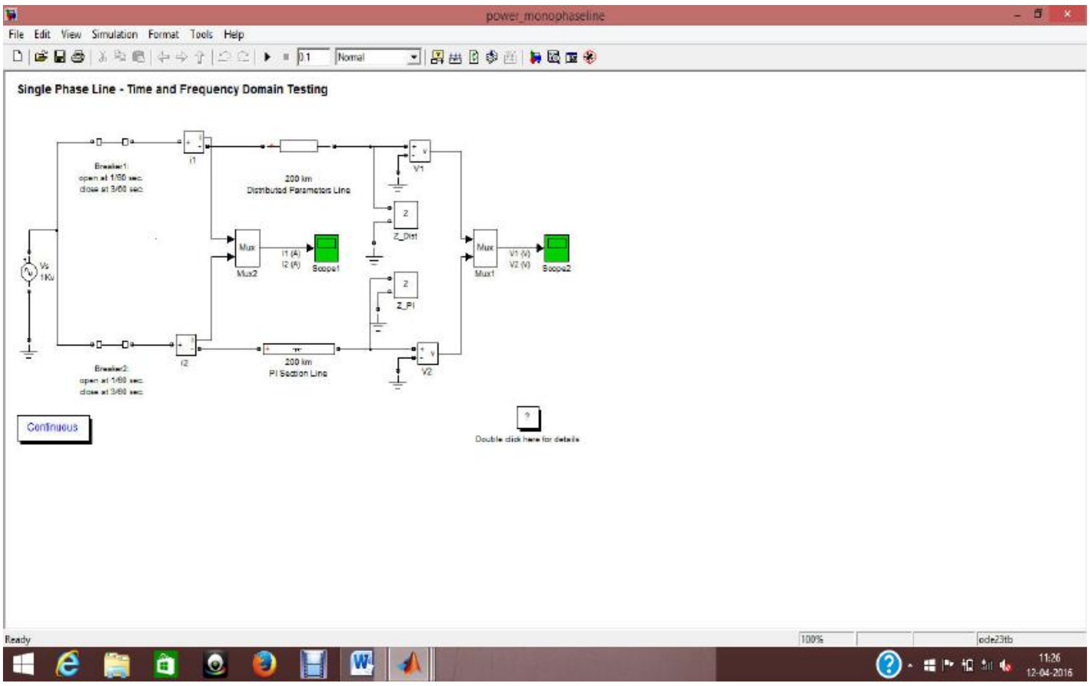
A 200 km line is connected on a 1 kV, 60 Hz infinite source. The line is deenergized and then reenergized after 2 cycles. The simulation is performed simultaneously with two different line models:

- Distributed parameter line

- PI section line consisting of two 100 km sections.

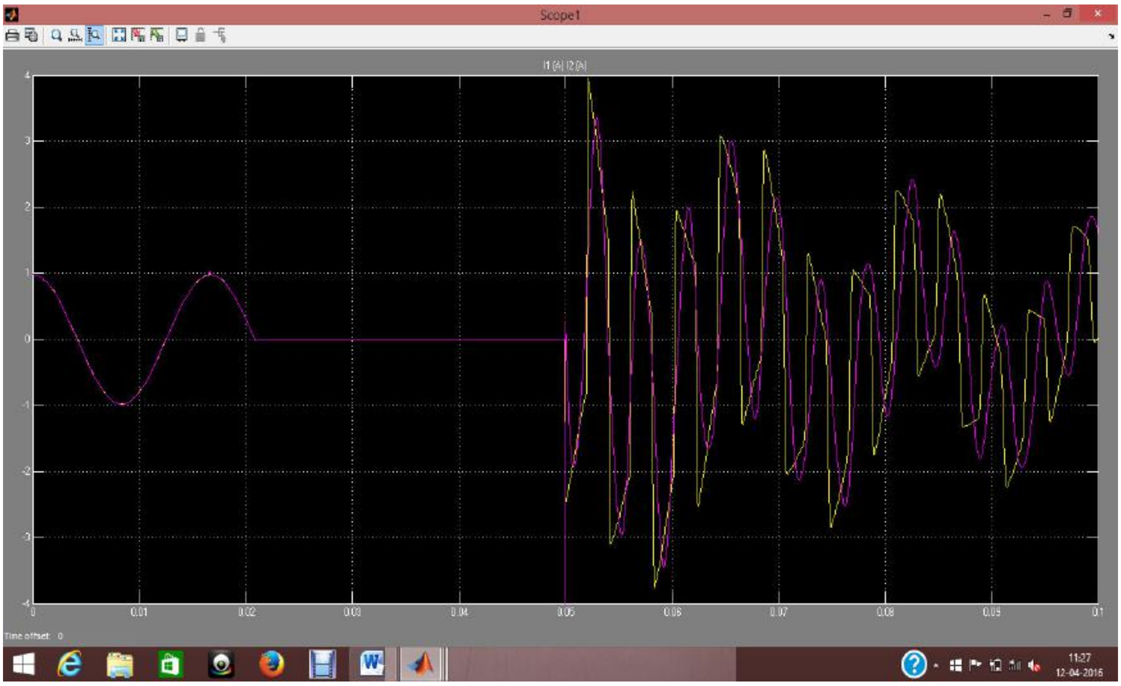
Currents at the sending end and voltages at the receiving end are compared for the two line models. Impedance Measurement blocks are connected at the open end of both lines in order to compare their frequency responses

Simulink Model

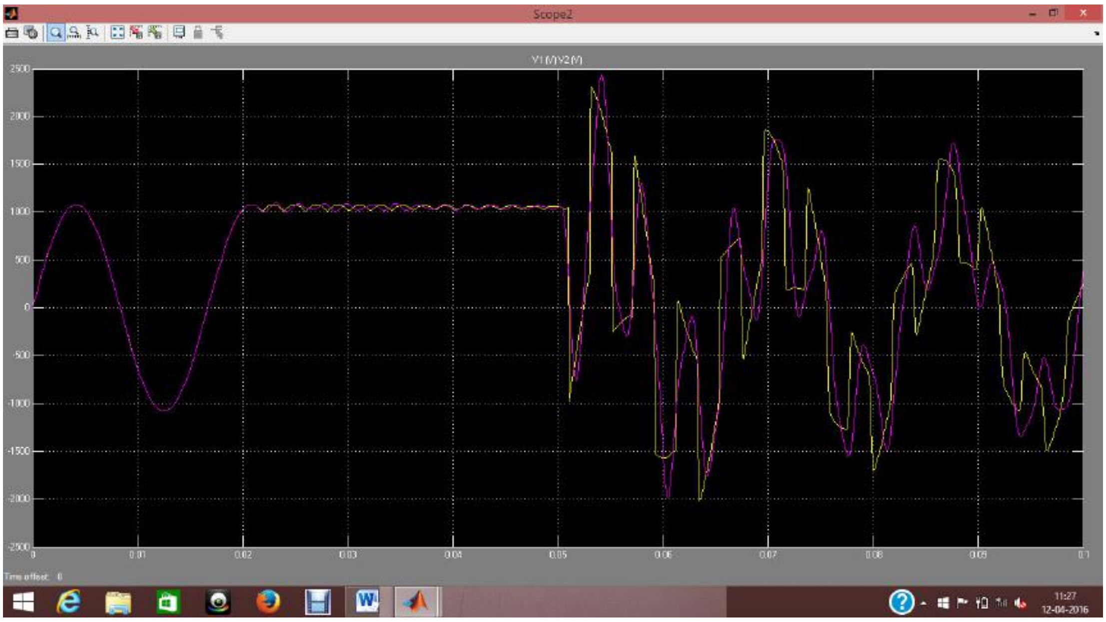


Results

Scope 1



Scope2



Conclusion

EXPERIMENT NO-4

AIM- Develop a Simulink model for single phase rectifier circuit.

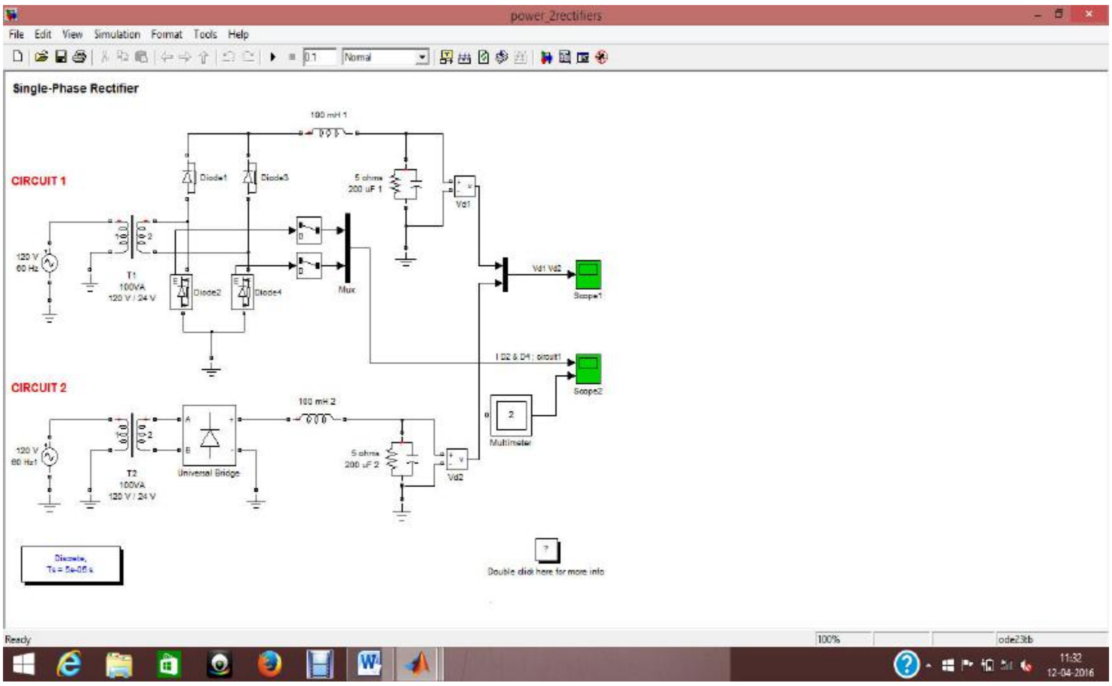
Circuit Description

This system contains two identical circuits showing the operation of a single phase rectifier.The rectifiers are fed by a 120 V/ 24 V linear transformer. The rectified voltage is filtered by a 100 mH / 200 uF filter and applied to a 5 ohm resistive load. The load voltages are measured by two Voltage Measurement blocks Vd1 and Vd2

The top circuit (circuit 1) uses four individual diodes connected in a bridge configuration. The currents of diodes 2 and 4 are obtained at the measurement 'm' output of the diode blocks and sent to input 1 of Scope 2 through Selector and Multiplex blocks.

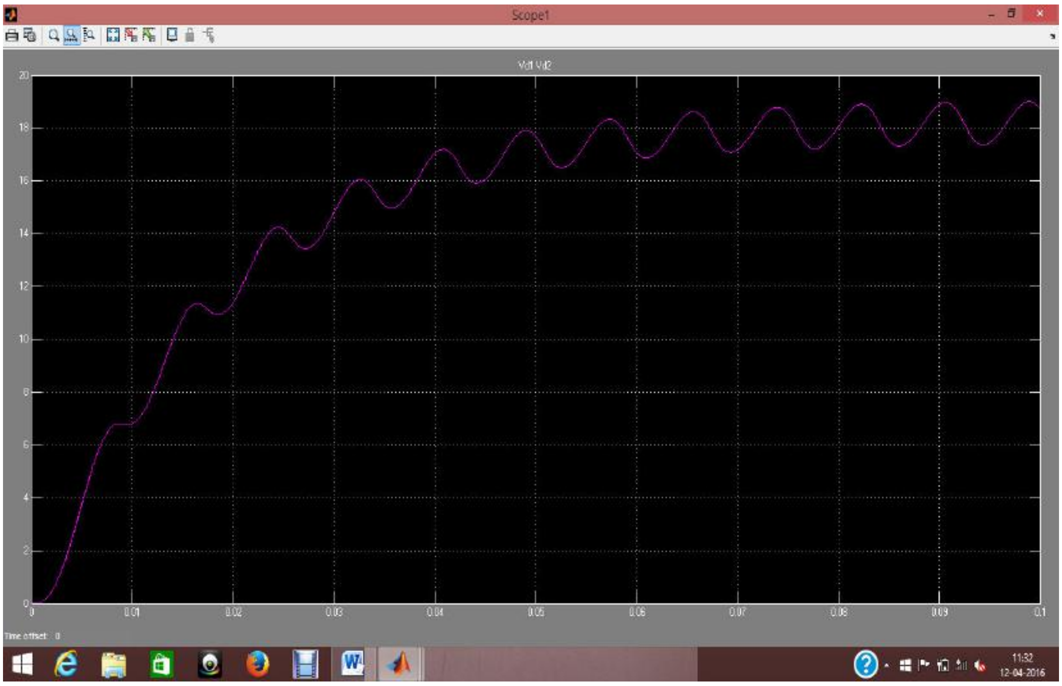
The bottom circuit (circuit 2) is functionally identical to circuit 1, but the circuit assembly is considerably simplified by the use of the Universal Bridge.

Simulink Model

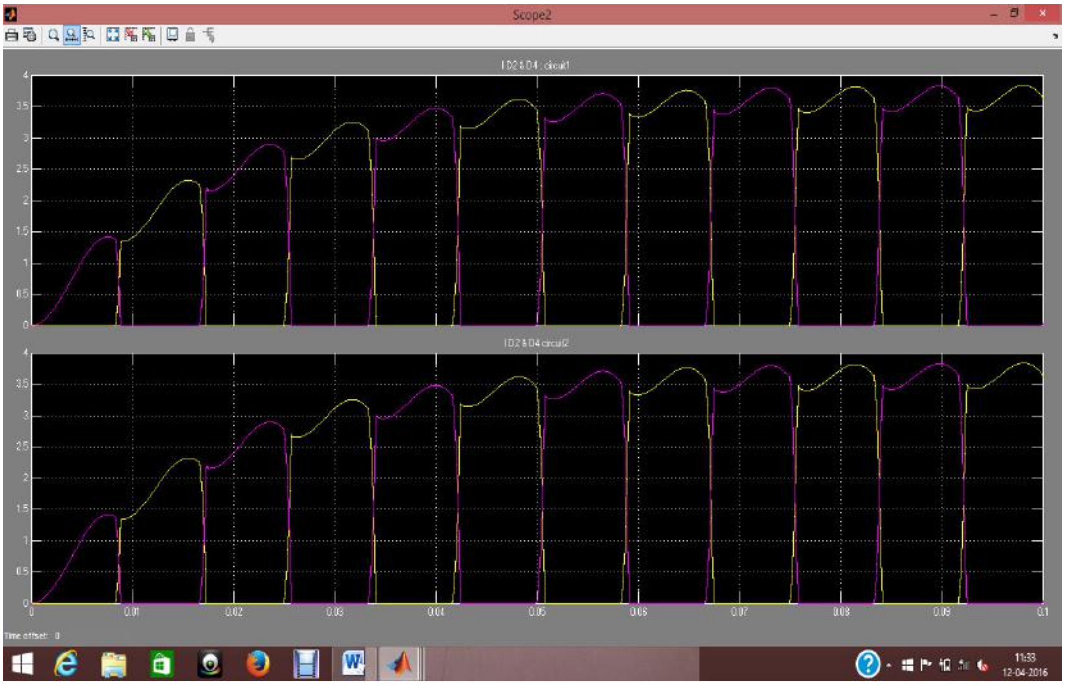


Results-

Scope1



Scope2



Conclusion

<u>EXPERIMENT NO-5</u>

<u>AIM</u>- Develop a Simulink model for Ideal Switching Device solution method to simulate a full wave rectifier using ideal diodes.

<u>Circuit Description</u>

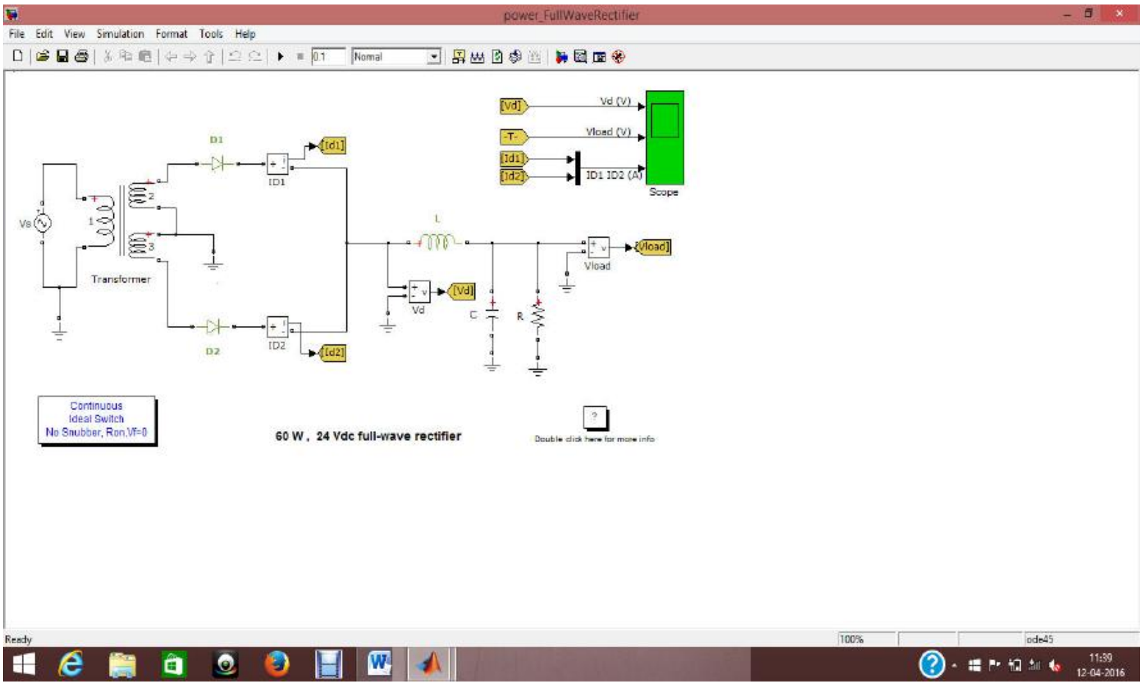
Open the power gui block and click on the Configure parameters button to access the block parameters. The Enable use of ideal switching devices option is selected and it tells Sim Power System to use the Ideal Switching Device solution for this model.

A convenient way of disabling snubbers of all switches in the model is to select the Disable snubbers in switching devices. Alternatively you can individually disable the snubbers of selected switches by specifying Rs=inf in their block menu.

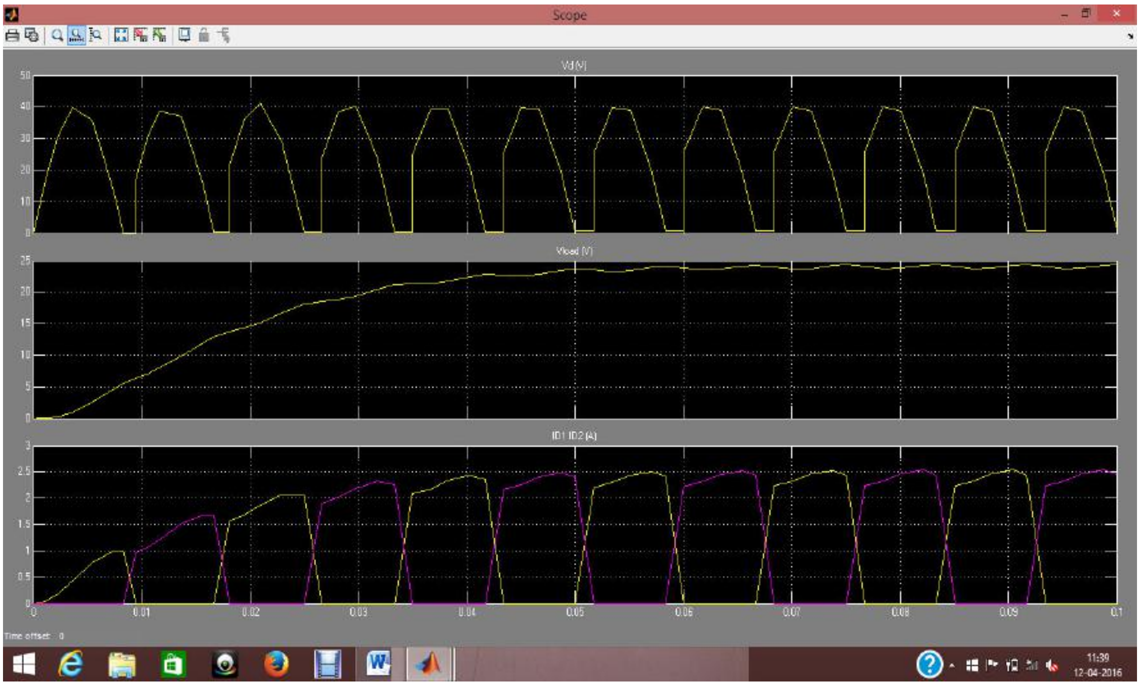
In order to simulate diodes as ideal switches, the Disable On resistance and Disable Forward voltage options are also selected.The elimination of the snubbers reduces the circuit stiffness so that you can use a non-stiff solver such as ode45. This solver produces correct results and good simulation speed.

To simulate this circuit with the original SimPowerSystems simulation method, you need to use snubbers across diodes D1 and D2 because these elements are connected in series with inductances (transformer leakage inductances of the two secondary windings and filter inductance L). Otherwise, when you start the simulation SimPowerSystems will prompt an error message. In this case if you attempt to simulate this circuit with specified resistive snubbers (Rs=1e6 ohms) by using the ode45 solver, you will observe a slow simulation speed and numerical oscillations. If you use instead the ode23tb recommended solver, you will observe wrong simulation results unless you use a relative tolerance of 1e-6.

Simulink model



Results



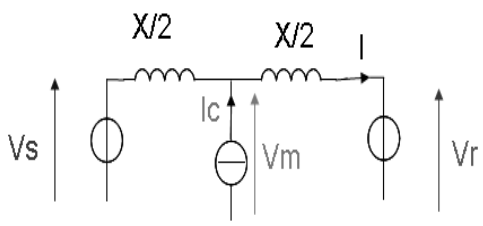
Conclusion

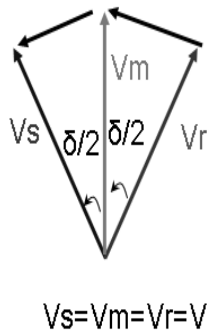
EXPERIMENT NO-6

AIM- Effect of compensation on voltage profile in transmission line Using MATLAB-SIMULINK

Shunt compensation

In shunt compensation, power system is connected in shunt (parallel) with the FACTS. It works as a controllable current source. Shunt compensation is of two types:





Shunt Compensation

Shunt capacitive compensation

This method is used to improve the power factor. Whenever an inductive load is connected to the transmission line, power factor lags because of lagging load current. To compensate, a shunt capacitor is connected which draws current leading the source voltage. The net result is improvement in power factor.

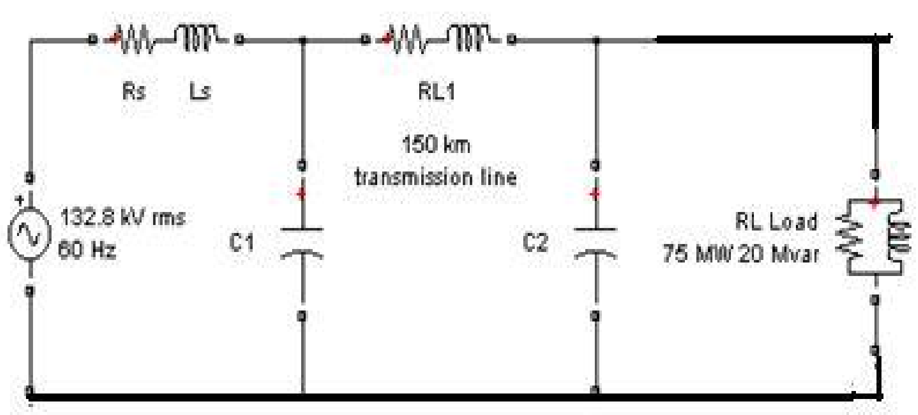
Shunt inductive compensation

This method is used either when charging the transmission line, or, when there is very low load at the receiving end. Due to very low, or no load – very low current flows through the transmission line. Shunt capacitance in the transmission line causes voltage amplification

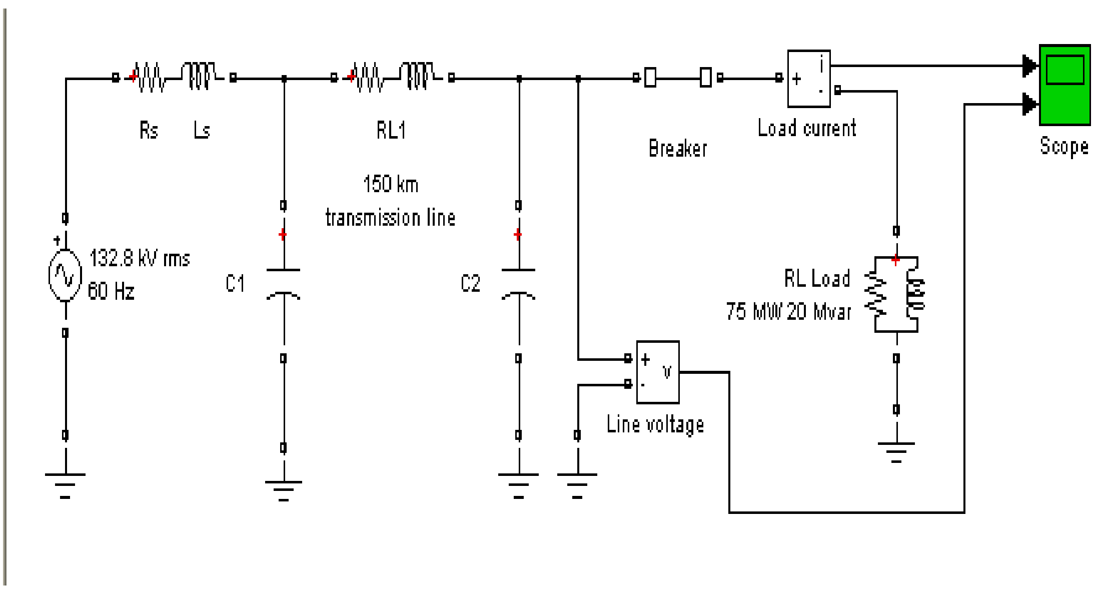
(Ferranti Effect). The receiving end voltage may become double the sending end voltage (generally in case of very long transmission lines). To compensate, shunt inductors are connected across the transmission line.

Exercise

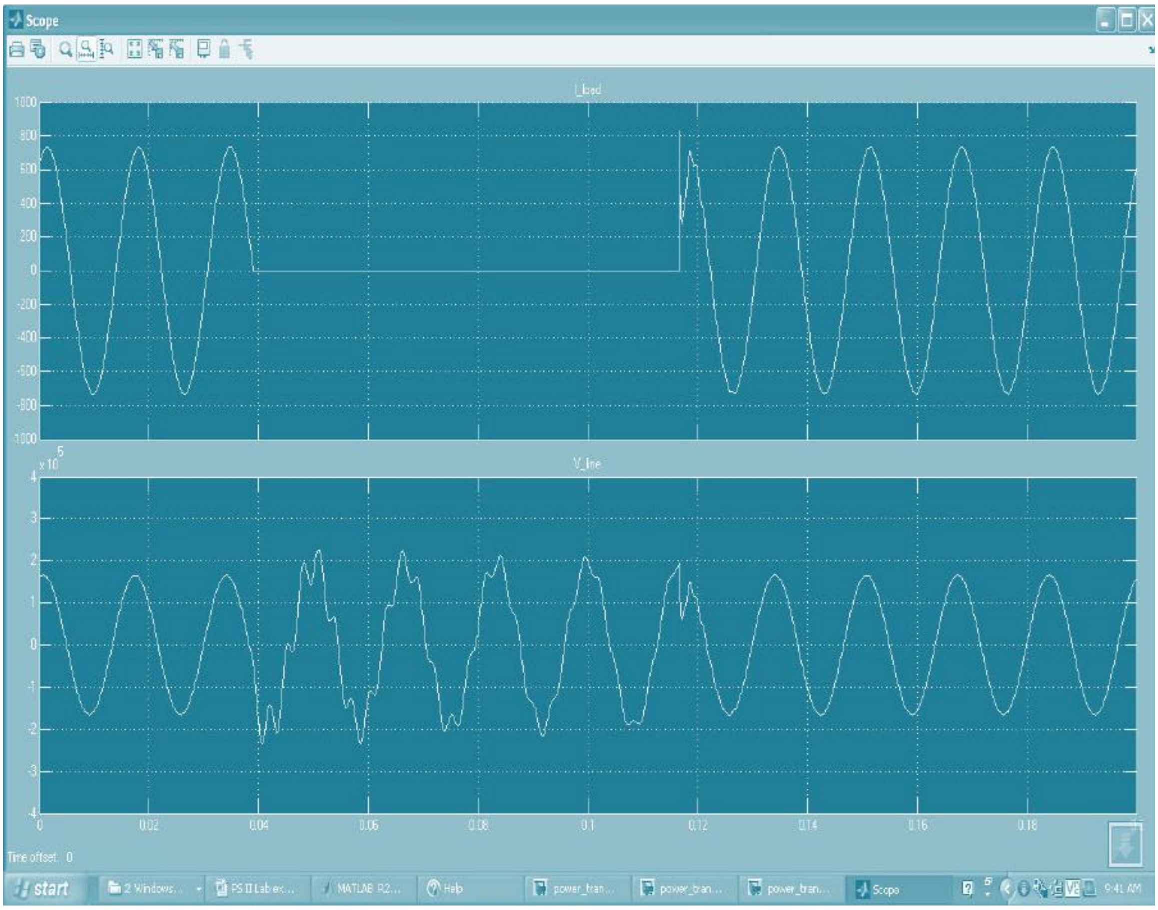
A single line diagram is shown in figure in where a generator of 132.8 kvrms,60hz is connected to 150 km long transmission line with a load of 75 MW , 20 Mvar. Show the effect os shunt compensation in the line in the form of capacitor and inductor with the help of simulink model.



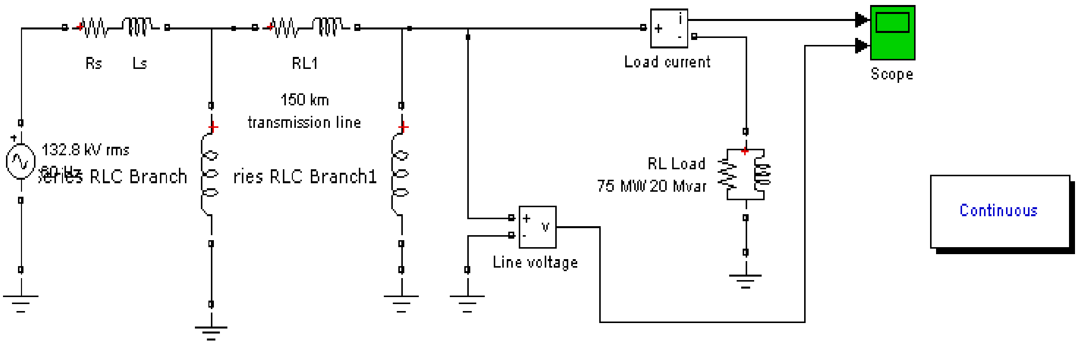
MATLAB-SIMULINK for compensation using shunt capacitor



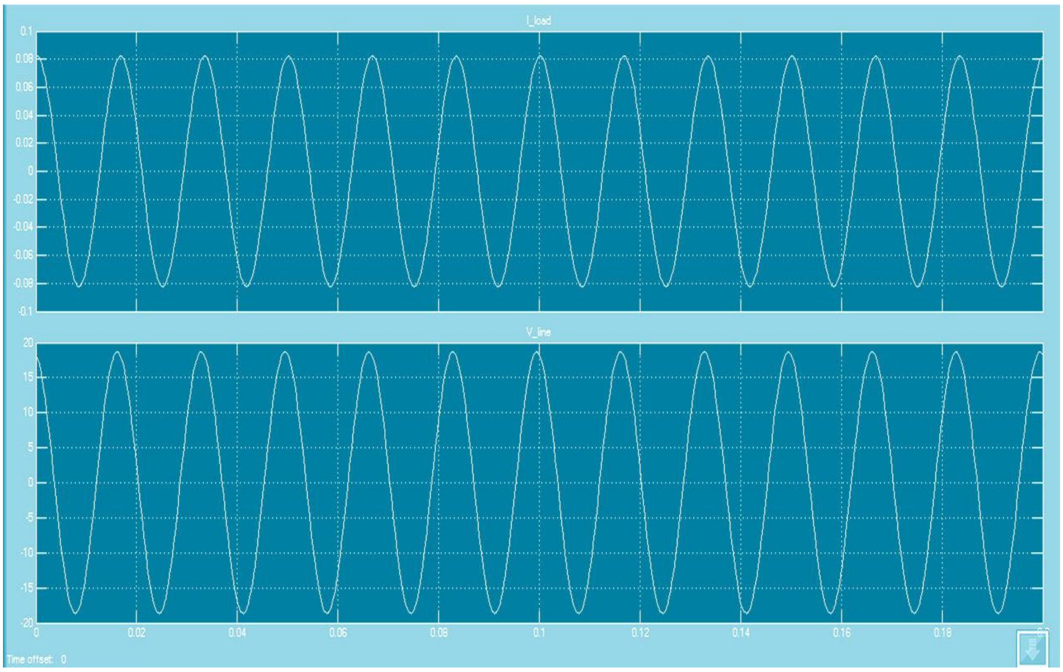
Results



MATLAB-SIMULINK for compensation using shunt inductor



Results



Conclusion

<u>**EXPERIMENT NO-7**</u>

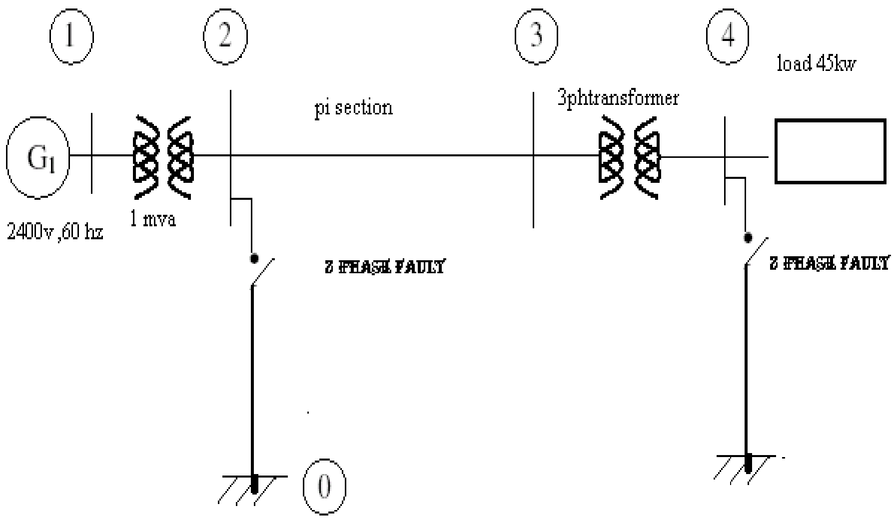
<u>AIM</u>- Transmission system behavior before and after fault condition Using MATLAB-SIMULINK

<u>THEORY</u>

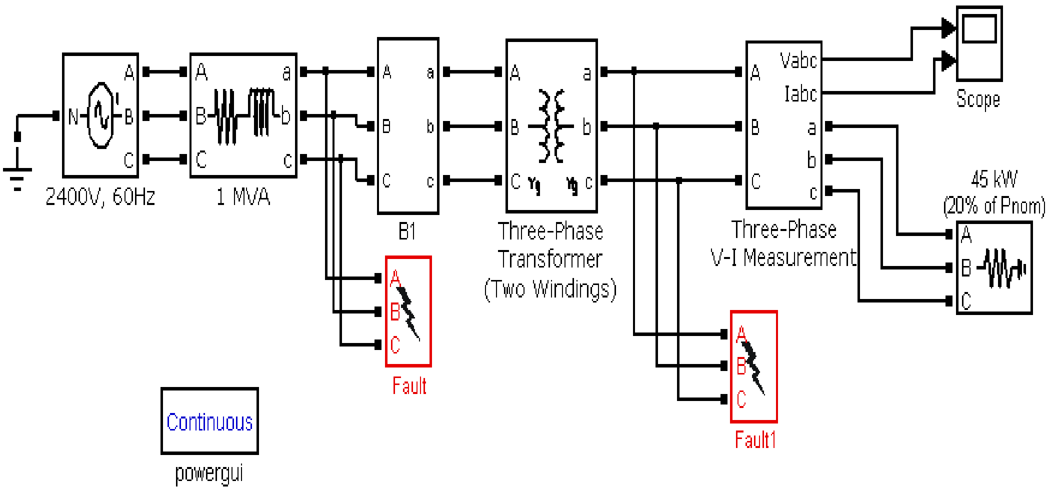
Short circuit studies are performed to determine bus voltages and currents flowing in different parts the system when it is subjected to a fault. The current flowing immediately after the fault consists of an a.c component which eventually reaches steady state and a fast decaying d.c component which decays to zero. Only the a.c component is considered in the analysis. The analysis is done using simulation technique assuming the system to be under quasi-steady state and is done for various types of faults such as three-phase-to ground, line-to-ground, line-to-line and double-line-to-ground. The results of fault studies are verified for pre and after fault conditions and to assess the voltage dips and flow af currents during fault. It is one of the primary studies to be performed whenever system expansion is planned.

<u>Exercise</u>

For the two-bus system shown in fig determine the fault voltage and current for a three phase fault at at bus 1 and 2 . Load current can be assumed to be negligible.

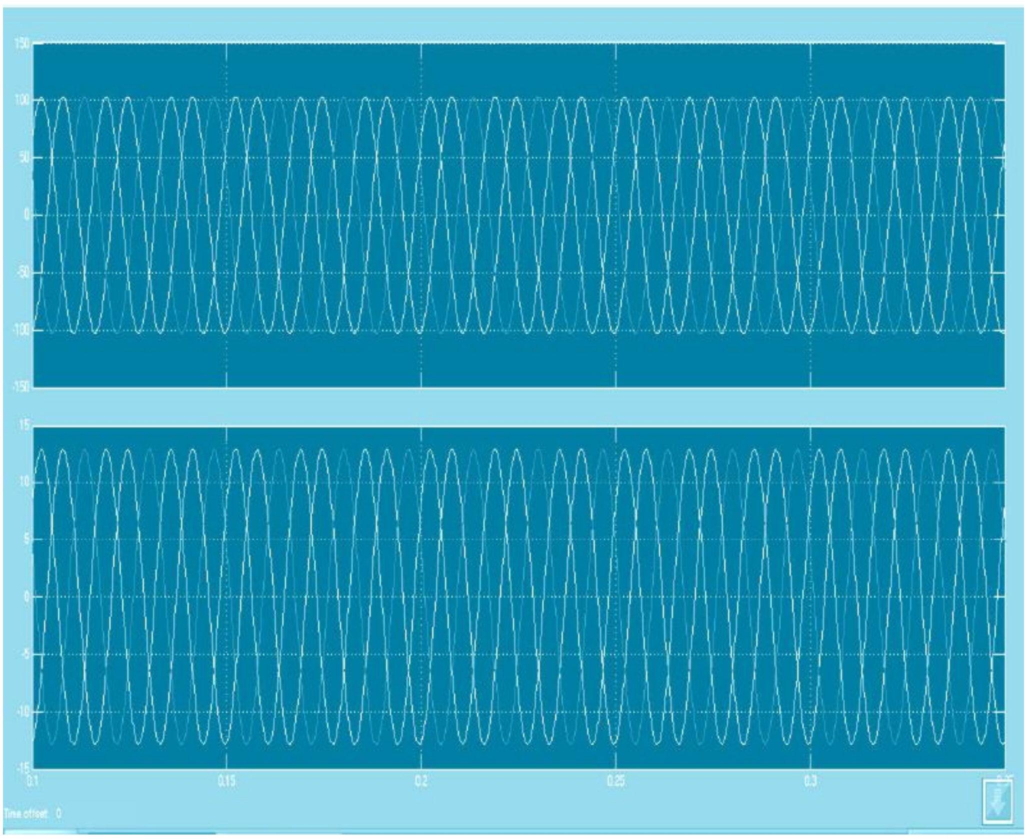


MATLAB-SIMULINK model for transmission line with fault condition

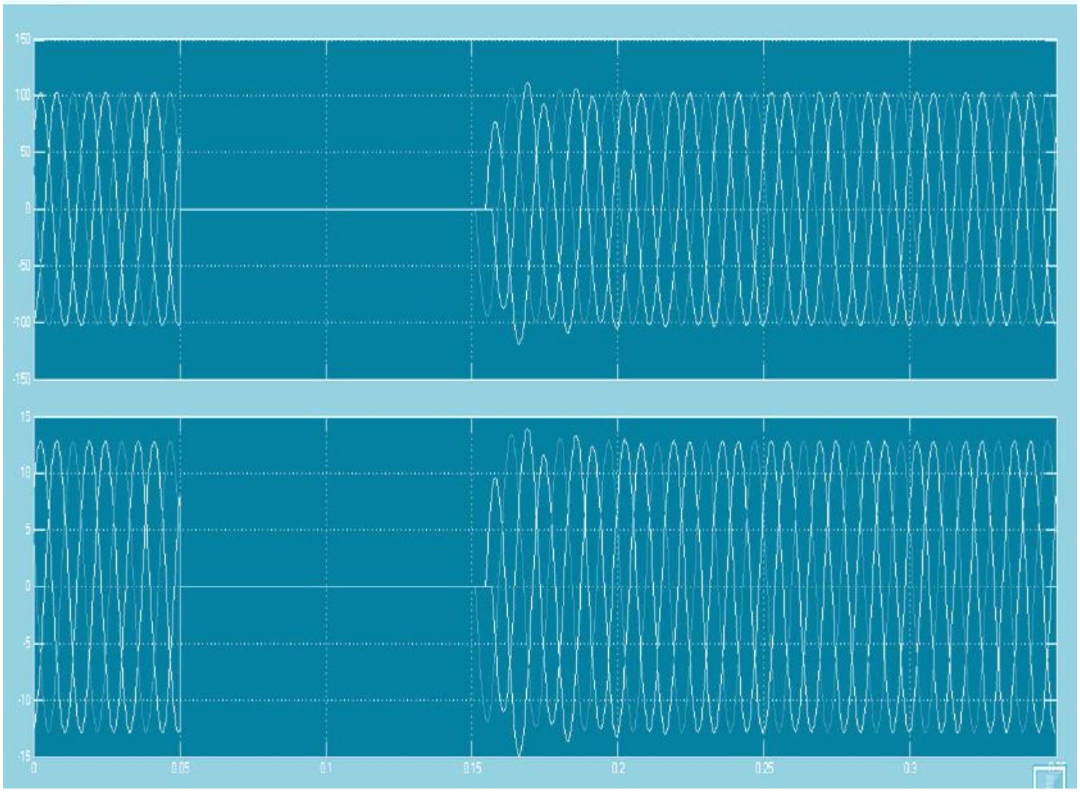


RESULTS

Before fault conditions



After fault conditions

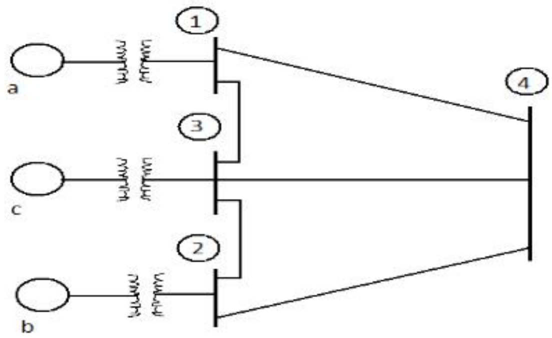


Conclusion

EXPERIMENT NO-8

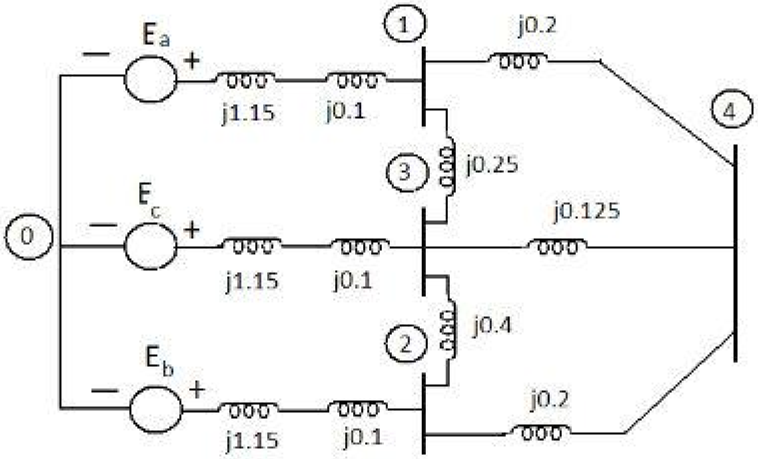
AIM- Evaluate the value of voltages for a 4-BUS system using node equations in MATLAB

GIVEN ONE UNEDIAGRAM



REACTANCE DIAGRAM

In the first step, we draw the reactance diagram of the given one-line diagram as shown below:



- Applying KCL at node-2:

$h= (V_2\text{-}O)\ Y20 + (V_2\text{-}V_3)\ Y23 + (V_Z\text{-}V_4)\ Y24$

12= OV,+ (Y20+Y23+Y24) V2+ (-Y23) V3+ (-Y24) V4

- Applying KCL at node-3:

13= (VrO) Y30+ (VrV,) Y3'+ (VrV4) Y34+ (V3-V2) Y32

h= (-Y3') V,+ *(-Y3Z)* V2+ (Y30+Y31+Y34) V3+ (-Y34) V4

- Applying KCLatnode-4:

0= (V4-VdY'4+ (V4-V3) Y43+ (V4-V2) Y42

0= (-YI4) V,+*(-Y41)* V**1** + (-Y34) V?,+ *(Y'4+Y43+Y41)* V4

Matrix formofthe node equations is:

$$\begin{bmatrix} I_1 \\ I_2 \\ I_3 \\ I_4 \end{bmatrix} = \begin{bmatrix} Y_{11} & Y_{12} & Y_{13} & Y_{14} \\ Y_{21} & Y_{22} & Y_{23} & Y_{24} \\ Y_{31} & Y_{32} & Y_{33} & Y_{34} \\ Y_{41} & Y_{42} & Y_{43} & Y_{44} \end{bmatrix} \begin{bmatrix} V_1 \\ V_2 \\ V_3 \\ V_4 \end{bmatrix}$$

Where:

$$Y_{11} = y_{10}+y_{14}+y_{13}$$

$$Y_{22} = y_{20}+y_{23}+y_{24}$$

$$Y_{33} = y_{30}+y_{31}+y_{34}$$

$$Y_{44} = y_{14}+y_{43}+y_{42}$$

EXPERIMENT NO-9

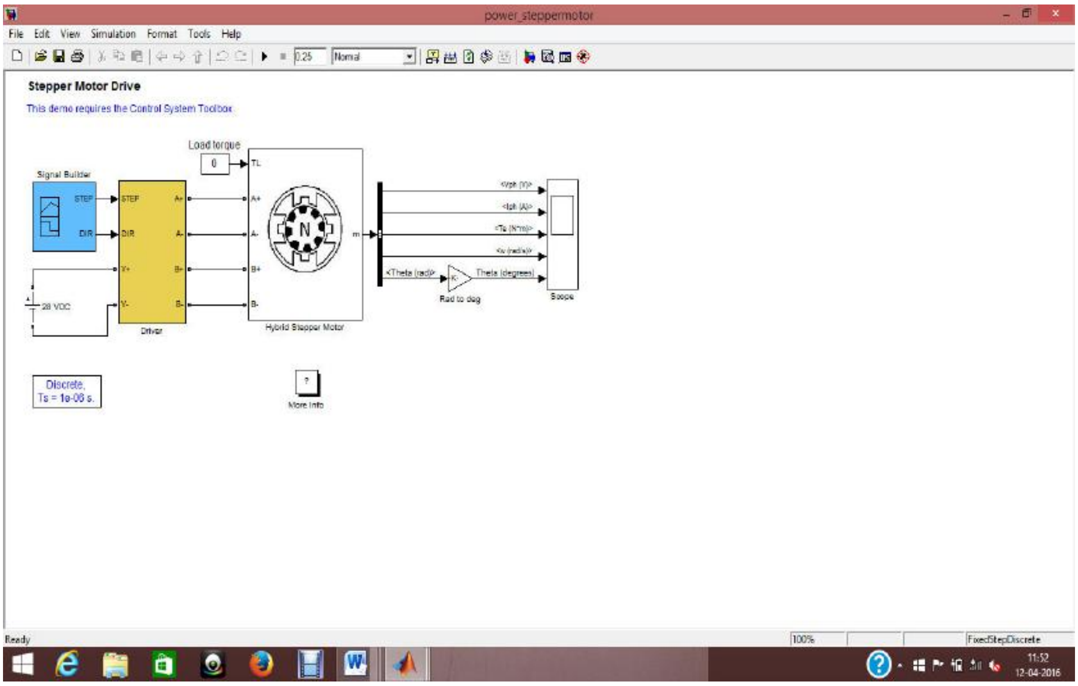
AIM- Develop a Simulink model for hybrid Stepper Motor Drive

Model Description

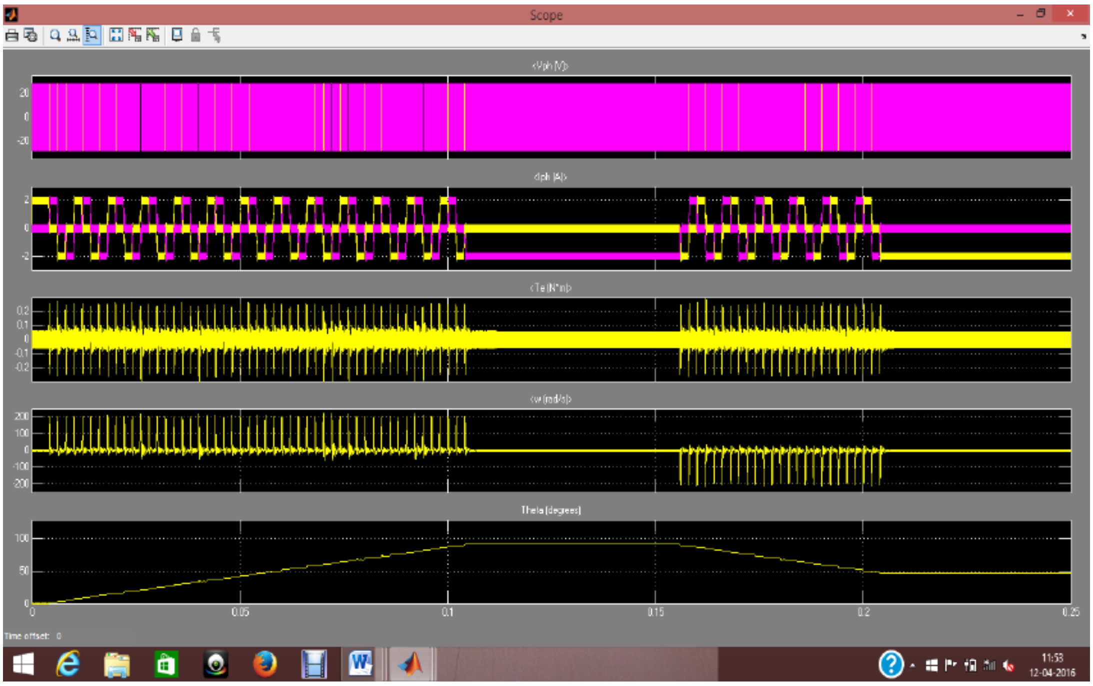
This example presents a stepper motor drive using the Hybrid Two-Phase model selected among the options on the dialog window. The motors parameters are those of a small stepper motor (size 23). The motor phases are fed by two H-bridge MOSFETPWM converters. The DC bus is represent by a 28-V DC voltage source. The motor currents are independently regulated by two hysteresis-based controllers that generate the MOSFETs drive signals by comparing the measured currents with their references. The ripple in the current waveforms is controlled by the hysteresis band of the comparators. The switching frequency is variable and dependent on the motor parameters.

In this example, single-phase-on excitation scheme is used because of its simplicity. Square-wave current references are generated using the current amplitude and the step frequency parameters specified in the dialog window. The movement of the stepper drive is controlled by the STEP and DIR signals received from Signal Builder block.

Simulink Model



Results



Conclusion