NRI INSTITUTE OF INFORMATION SCIENCE AND TECHNOLOGY ,BHOPAL

LAB MANNUAL

SIMULATION LAB

DEPARTMENT OF ELECTRICAL AND ELECTRONICS ENGINEERING

EX 804 - MATLAB

List of Experiments

1. Introduction of various simulink blocks.
2. Study of various Electrical Toolbox i.e. Power System, Power Electronics, Control system, Electrical Measurement, Flexible AC Transmission.
3. Modeling and simulation of half wave rectifier.
4. Modeling and simulation of full wave rectifier.
5. Modeling and simulation of half wave controlled rectifier.
6. Modeling and simulation of full wave controlled rectifier.
7. Modeling and simulation of thyristorised control Reactor.
8. Modeling and simulation of thyristorised switched Reactor.
9. Developing Simulation Models for single phase Inverter, for different load models.
10. Developing Simulation Models using FACTs Devices.

EXPERIMENT NO. 1

AIM: INTRODUCTION OF VARIOUS SIMULINK BLOCKS.

INTRODUCTION: MATLAB (matrix laboratory) is a numerical computing environment and fourth-generation programming language. Developed by MathWorks, MATLAB allows matrix manipulations, plotting of functions and data, implementation of algorithms, creation of user interfaces, and interfacing with programs written in other languages, including C, C++, Java, and Fortran.

Although MATLAB is intended primarily for numerical computing, an optional toolbox uses the MuPAD symbolic engine, allowing access to symbolic computing capabilities. An additional package, Simulink, adds graphical multi-domain simulation and Model-Based Design for dynamic and embedded systems.

In 2004, MATLAB had around one million users across industry and academia. MATLAB users come from various backgrounds of engineering, science, and economics. MATLAB is widely used in academic and research institutions as well as industrial enterprises.

Cleve Moler, the chairman of the computer science department at the University of New Mexico, started developing MATLAB in the late 1970s. He designed it to give his student's access to LINPACK and EISPACK without them having to learn Fortran. It soon spread to other universities and found a strong audience within the applied mathematics community. Jack Little, an engineer, was exposed to it during a visit Moler made to Stanford University in 1983. Recognizing its commercial potential, he joined with Moler and Steve Bangert. They rewrote MATLAB in C and founded MathWorks in 1984 to continue its development. These rewritten libraries were known as JACKPAC. In 2000, MATLAB was rewritten to use a newer set of libraries for matrix manipulation, LAPACK.

MATLAB was first adopted by researchers and practitioners in control engineering, Little's specialty, but quickly spread to many other domains. It is now also used in education, in particular the teaching of linear algebra and numerical analysis and is popular amongst scientists involved in image processing.

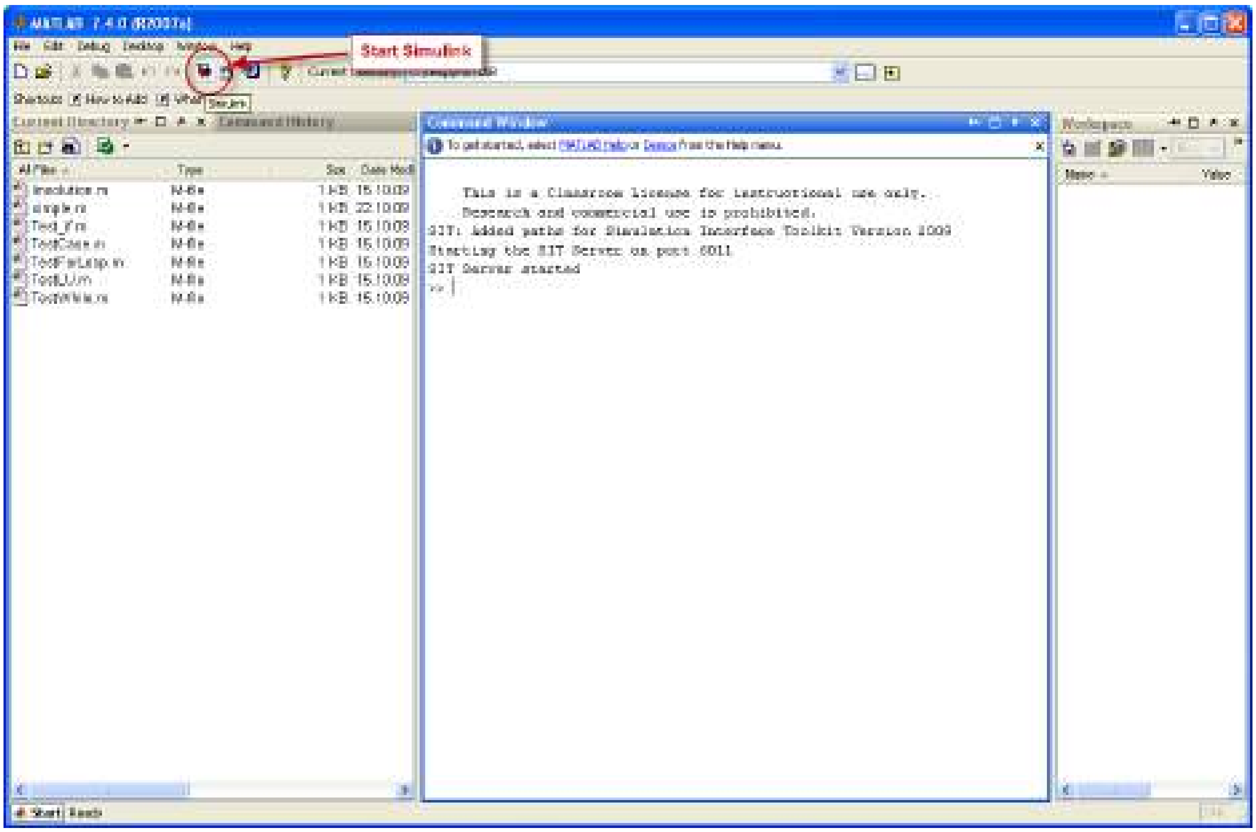
THEORY: Simulink is an environment for simulation and model-based design for dynamic and embedded systems. It provides an interactive graphical environment and a customizable set of block libraries that let you design, simulate, implement, and test a variety of time-varying systems, including communications, controls, signal processing, video processing, and image processing.
Simulink offers:

- A quick way of develop your model in contrast to text based-programming language such as e.g., C.
- Simulink has integrated **solvers**. In text based-programming language such as e.g., C you need to write your own solver.

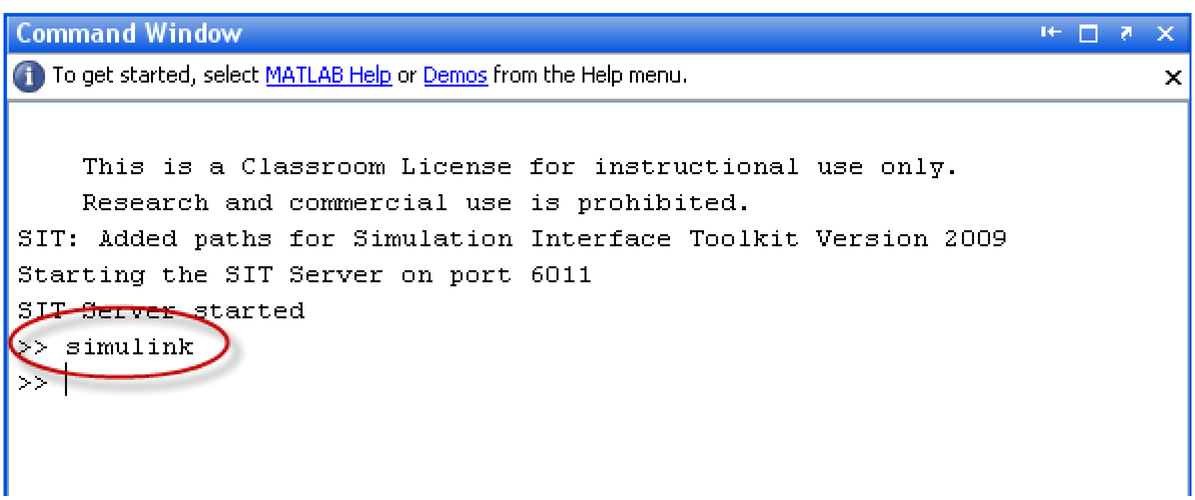
2 Start using Simulink

You start Simulink from the MATLAB IDE:
Open MATLAB and select the Simulink icon in the Toolbar:

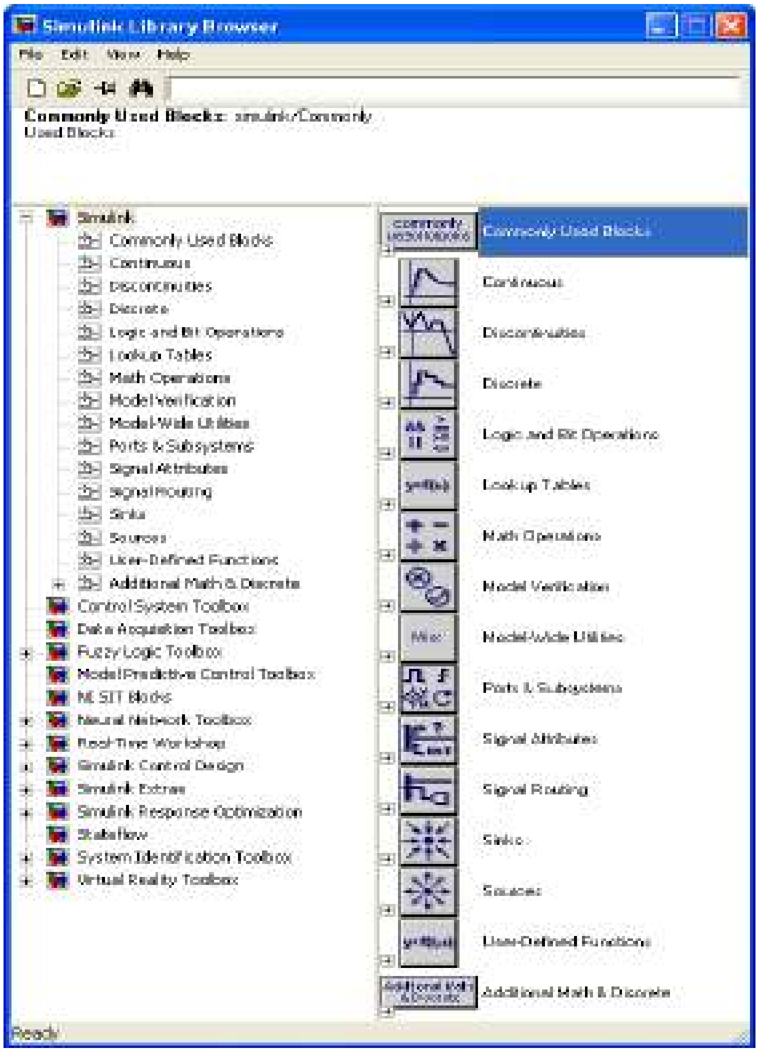
Or type

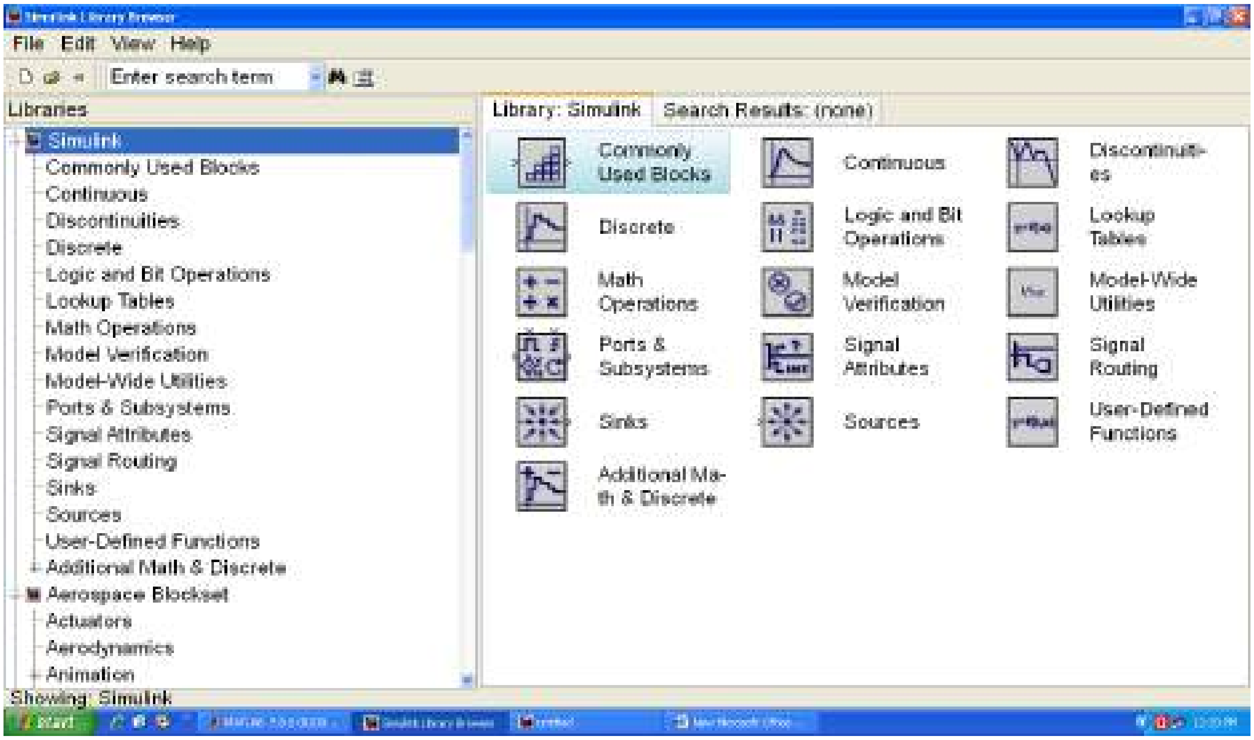


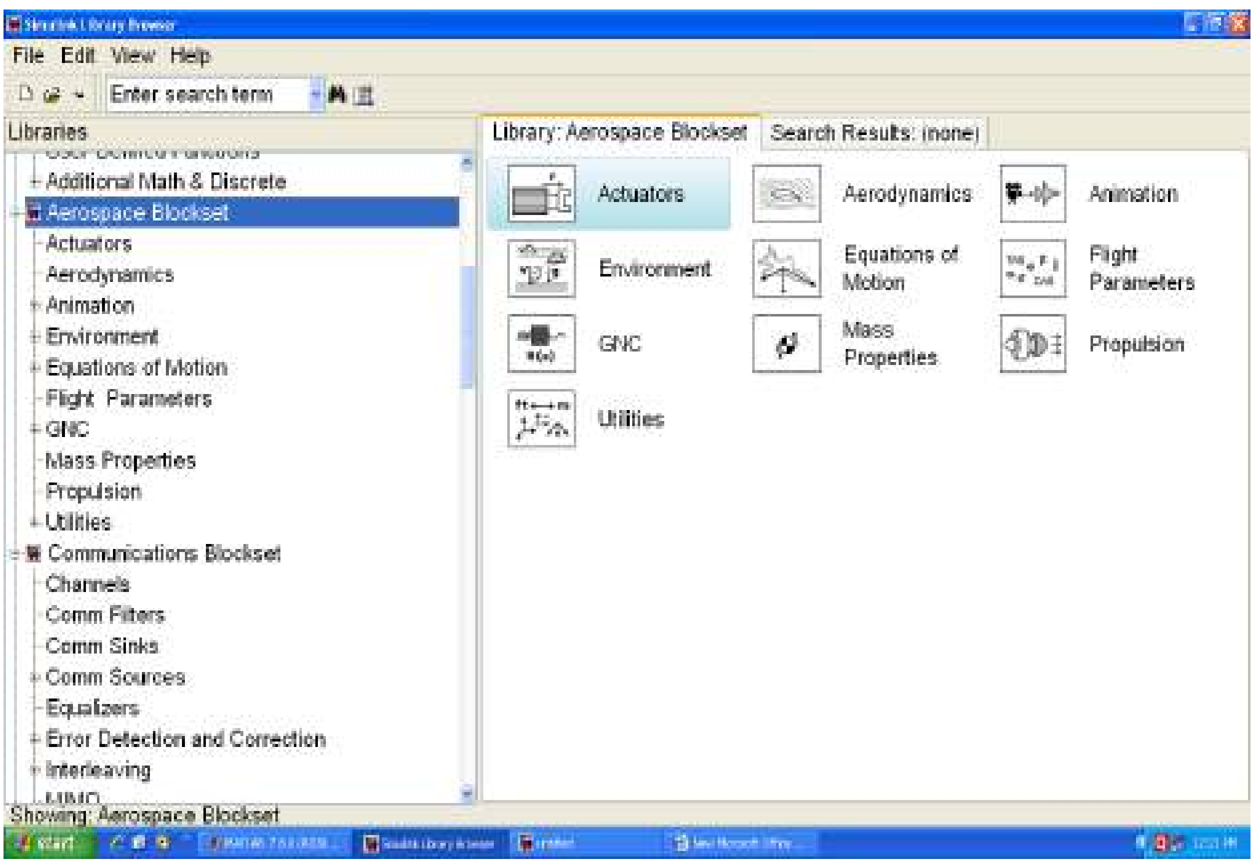
Or type "**simulink**" in the Command window, like this:

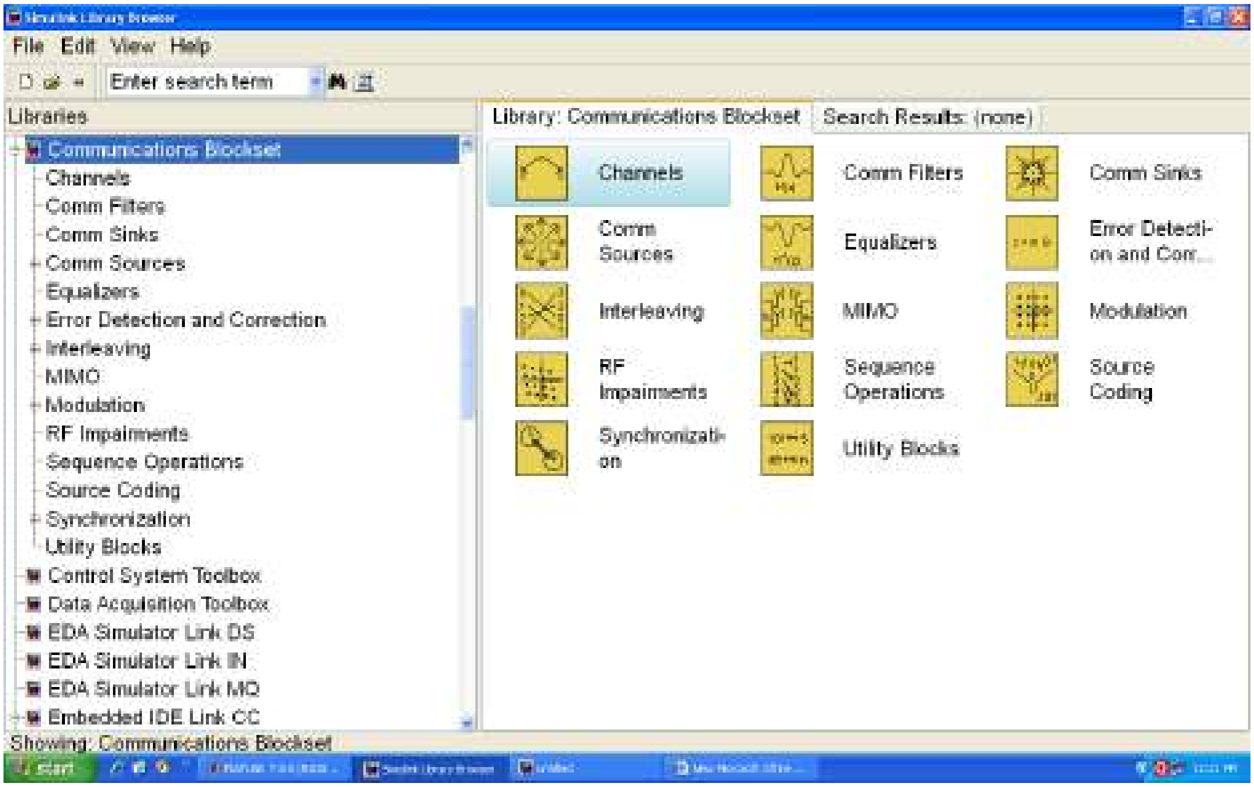


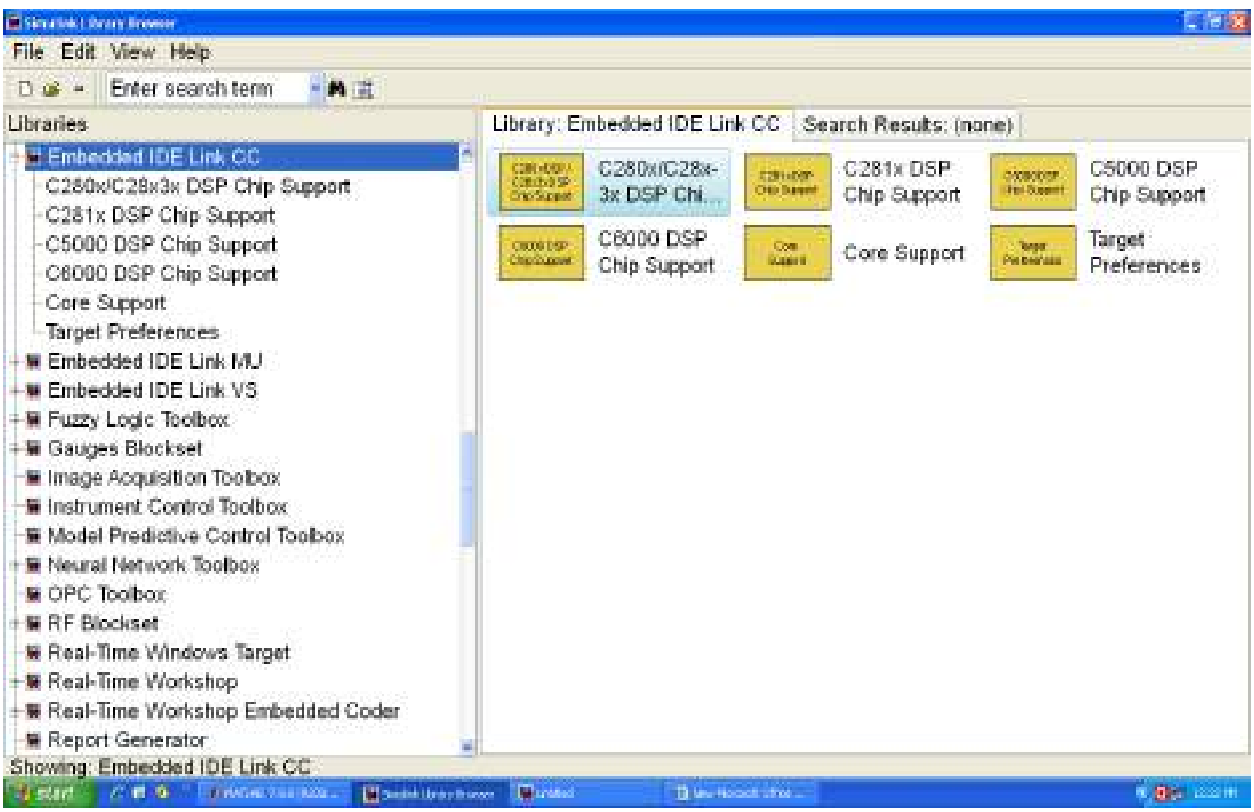
Then the following window appears (**Simulink Library Browser**):

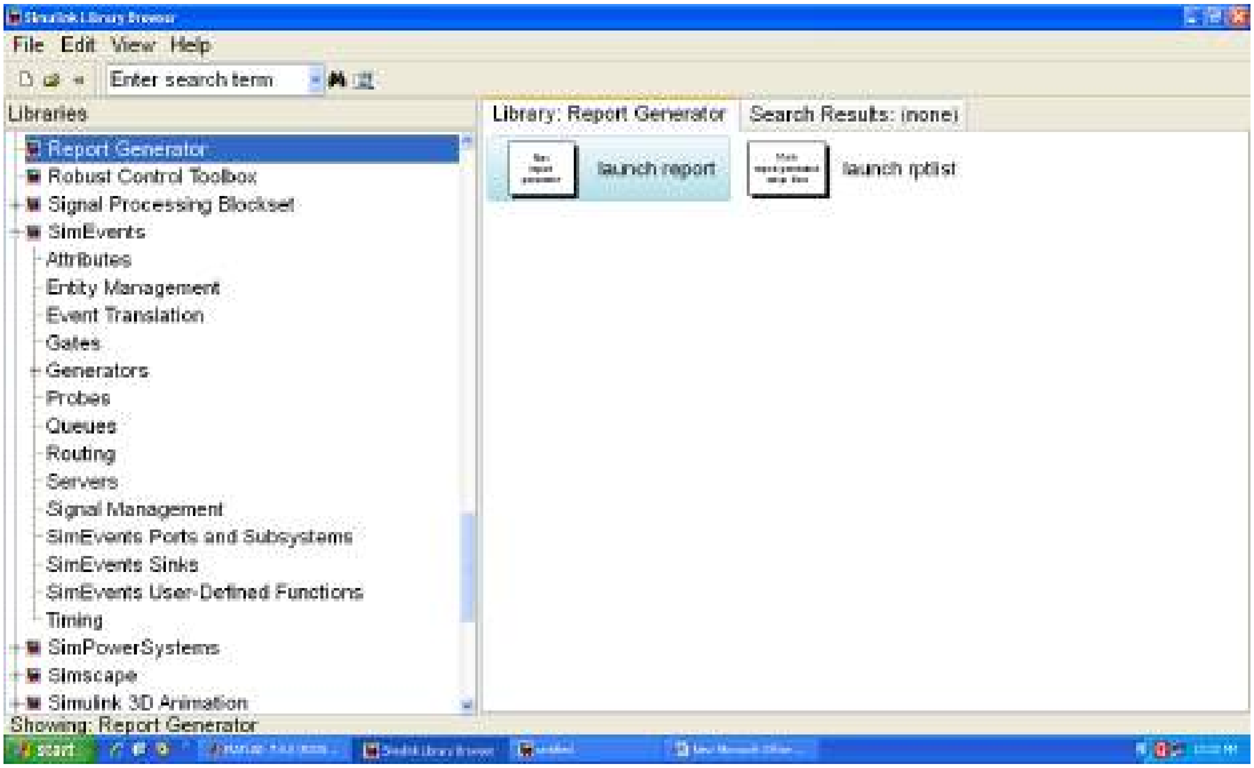


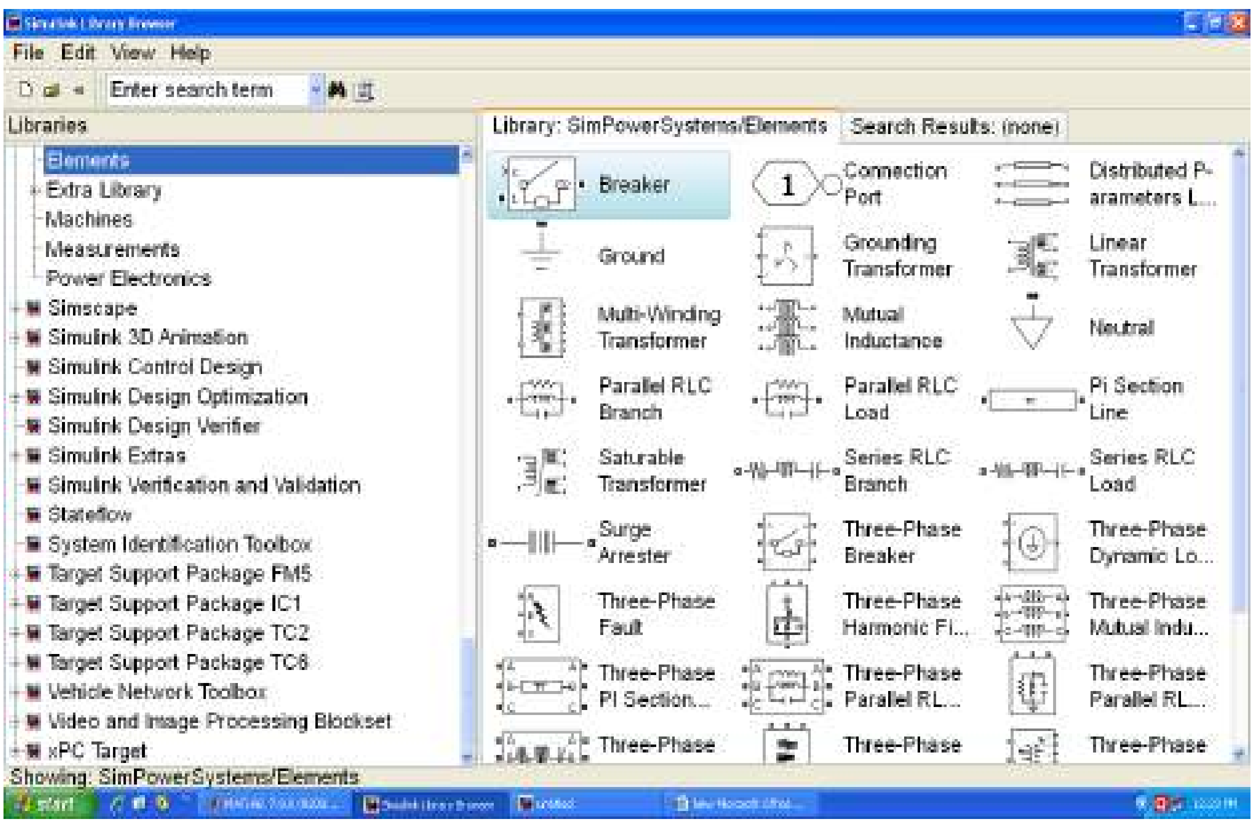

File Edit View Help
Enter search term
Libraries
Library: Simulink
Search Results: (none)
Simulink
Commonly Used Blocks
Continuous
Discontinuities
Discrete
Logic and Bit Operations
Lookup Tables
Math Operations
Model Verification
Model-Wide Utilities
Ports & Subsystems
Signal Attributes
Signal Routing
Sinks
Sources
User-Defined Functions
Additional Math & Discrete
Aerospace Blockset
Actuators
Aerodynamics
Animation
Commonly Used Blocks
Continuous
Discontinuities
Discrete
Logic and Bit Operations
Lookup Tables
Math Operations
Model Verification
Model-Wide Utilities
Ports & Subsystems
Signal Attributes
Signal Routing
Sinks
Sources
User-Defined Functions
Additional Math & Discrete
Showing: Simulink
start


File Edit View Help
Enter search term
Libraries
Library: Aerospace Blockset
Search Results: (none)
Additional Math & Discrete
Aerospace Blockset
Actuators
Aerodynamics
Animation
Environment
Equations of Motion
Flight Parameters
GNC
Mass Properties
Propulsion
Utilities
Communications Blockset
Channels
Comm Filters
Comm Sinks
Comm Sources
Equalizers
Error Detection and Correction
Interleaving
Actuators
Aerodynamics
Animation
Environment
Equations of Motion
Flight Parameters
GNC
Mass Properties
Propulsion
Utilities
Showing: Aerospace Blockset
start


File Edit View Help
Enter search term
Libraries
Library: Communications Blockset
Search Results: (none)
Communications Blockset
Channels
Comm Filters
Comm Sinks
Comm Sources
Equalizers
Error Detection and Correction
Interleaving
MIMO
Modulation
RF Impairments
Sequence Operations
Source Coding
Synchronization
Utility Blocks
Control System Toolbox
Data Acquisition Toolbox
EDA Simulator Link DS
EDA Simulator Link IN
EDA Simulator Link MQ
Embedded IDE Link CC
Error Detecti-on and Corr...
Synchronizati-on
Showing: Communications Blockset


File Edit View Help
Enter search term
Libraries
Library: Embedded IDE Link CC
Search Results: (none)
Embedded IDE Link CC
C280x/C28x3x DSP Chip Support
C281x DSP Chip Support
C5000 DSP Chip Support
C6000 DSP Chip Support
Core Support
Target Preferences
Embedded IDE Link MU
Embedded IDE Link VS
Fuzzy Logic Toolbox
Gauges Blockset
Image Acquisition Toolbox
Instrument Control Toolbox
Model Predictive Control Toolbox
Neural Network Toolbox
OPC Toolbox
RF Blockset
Real-Time Windows Target
Real-Time Workshop
Real-Time Workshop Embedded Coder
Report Generator
C280x/C28x-3x DSP Chi...
Showing: Embedded IDE Link CC


Simulink Library Browser
File Edit View Help
Enter search term
Libraries
Library: Report Generator
Search Results: (none)
Report Generator
Robust Control Toolbox
Signal Processing Blockset
SimEvents
Attributes
Entity Management
Event Translation
Gates
Generators
Probes
Queues
Routing
Servers
Signal Management
SimEvents Ports and Subsystems
SimEvents Sinks
SimEvents User-Defined Functions
Timing
SimPowerSystems
Simscape
Simulink 3D Animation
launch report
launch rptlist
Showing: Report Generator
start


Simulink Library Browser
File Edit View Help
Enter search term
Libraries
Library: SimPowerSystems/Elements
Search Results: (none)
Elements
Extra Library
Machines
Measurements
Power Electronics
Simscape
Simulink 3D Animation
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Extras
Simulink Verification and Validation
Stateflow
System Identification Toolbox
Target Support Package FM5
Target Support Package IC1
Target Support Package TC2
Target Support Package TC6
Vehicle Network Toolbox
Video and Image Processing Blockset
xPC Target
Breaker
Connection Port
Distributed Parameters L...
Ground
Grounding Transformer
Linear Transformer
Multi-Winding Transformer
Mutual Inductance
Neutral
Parallel RLC Branch
Parallel RLC Load
Pi Section Line
Saturable Transformer
Series RLC Branch
Series RLC Load
Surge Arrester
Three-Phase Breaker
Three-Phase Dynamic Lo...
Three-Phase Fault
Three-Phase Harmonic Fi...
Three-Phase Mutual Indu...
Three-Phase PI Section...
Three-Phase Parallel RL...
Three-Phase Parallel RL...
Three-Phase
Three-Phase
Three-Phase
Showing: SimPowerSystems/Elements
start

The **Simulink Library Browser** is the library where you find all the blocks you may use in Simulink. Simulink software includes an extensive library of functions commonly used in modeling a system. These include:

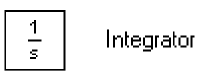
- Continuous and discrete dynamics blocks, such as Integration, Transfer functions, Transport Delay, etc.

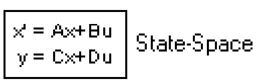
- Math blocks, such as Sum, Product, Add, etc

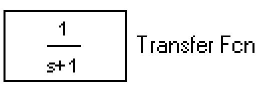
- Sources, such as Ramp, Random Generator, Step, etc

2.1 Block Libraries

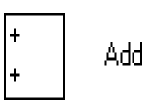
Here are some common used **Continuous** Blocks:

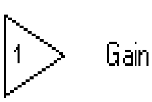


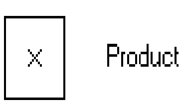


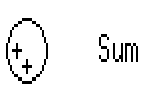


Here are some common used **Math Operations** Blocks:







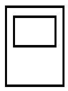


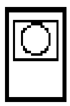
Here are some common used **Signal Routing** Blocks:

 Mux

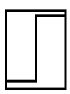
 Demux

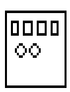
Here are some common used **Sinks** Blocks:

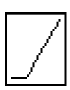
 Scope

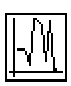
 XY Graph

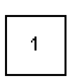
Here are some common used **Sources** Blocks:

 Step

 Signal Generator

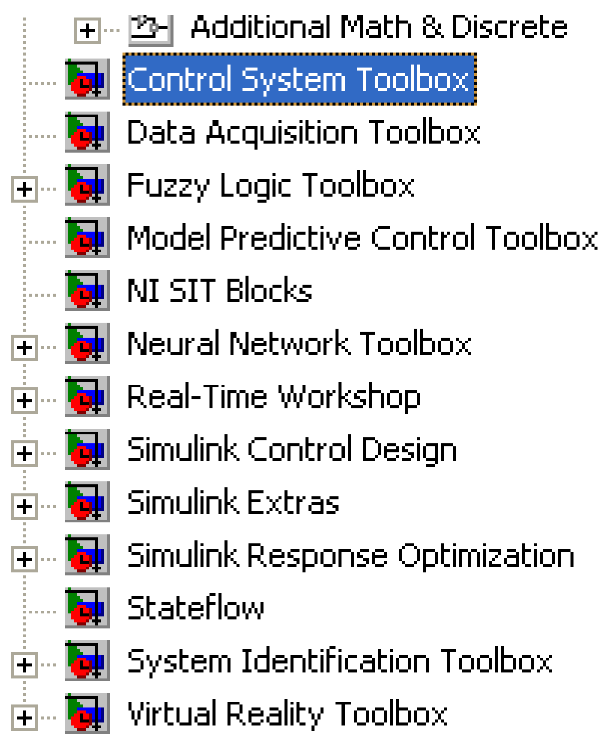
 Ramp

 Random Number



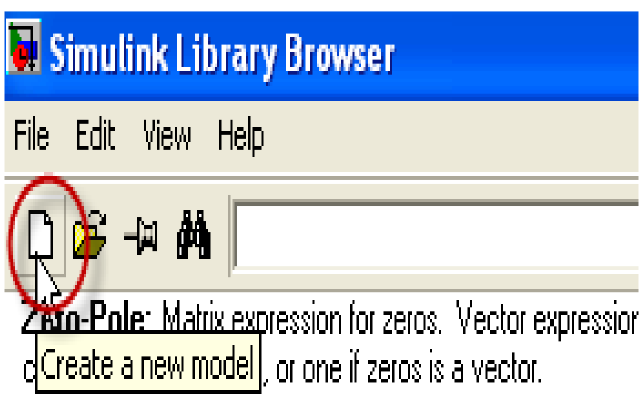
Constant

In addition there are lots of block in different Toolboxes:

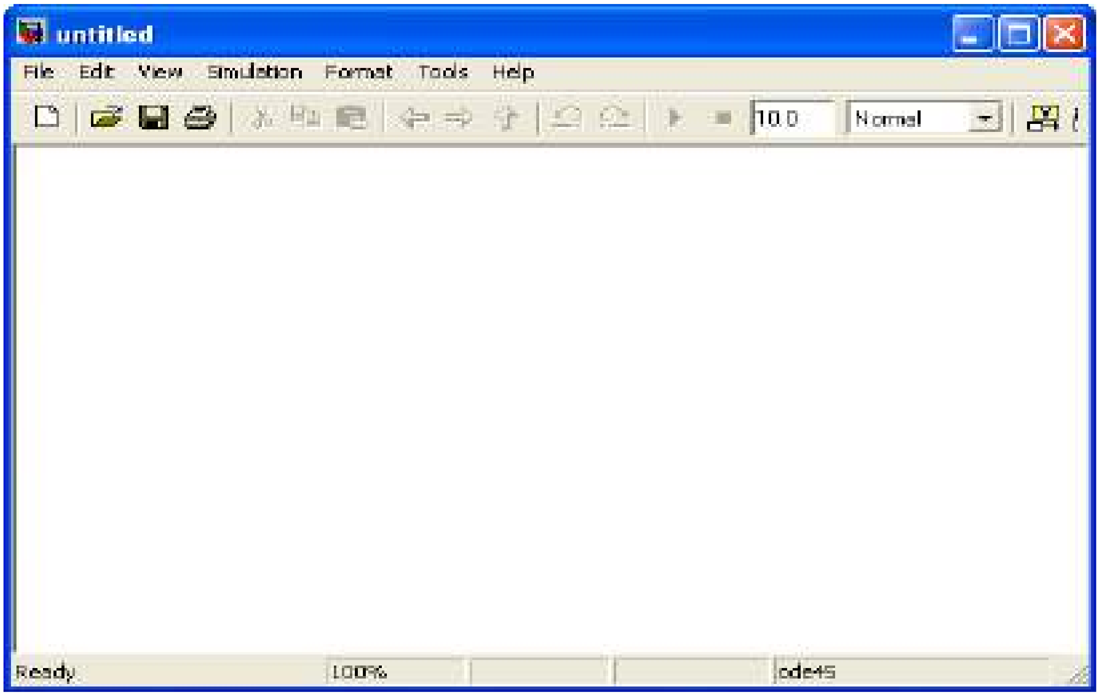


2.2 Create a new Model

Click the New icon on the Toolbar in order to create a new Simulink model:



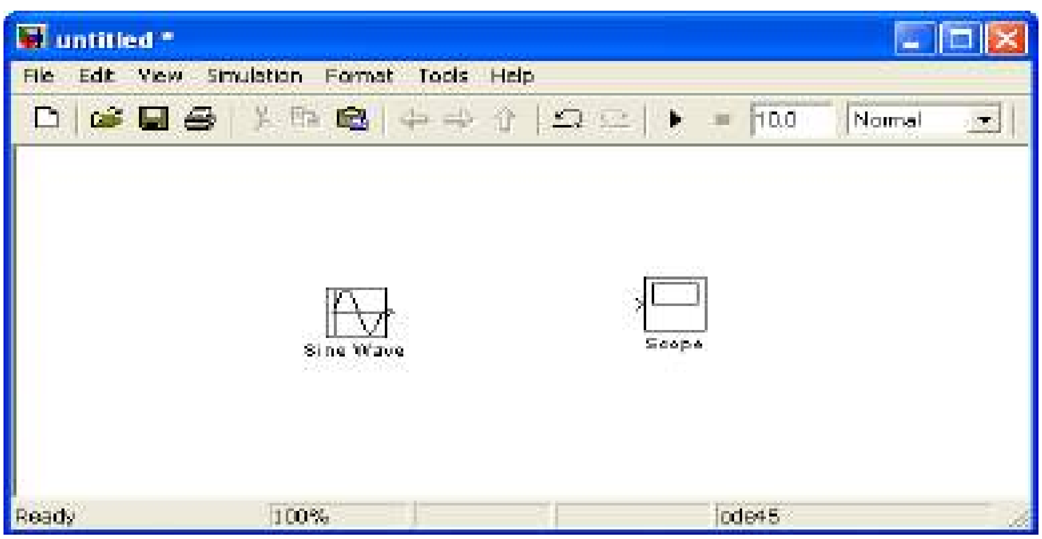
The following window appears:



You may now drag the blocks you want to use from the Simulink Library Browser to the model surface (or right-click on a block and select "Add to…").

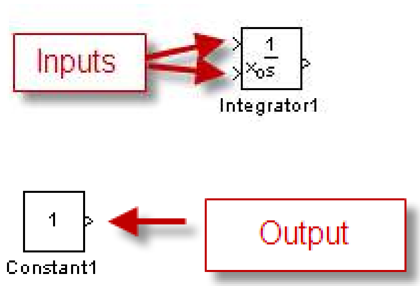
Example:

In this example we place (drag and drop) to blocks, a Sine Wave and a Scope, on the model surface:



2.3 Wiring techniques

Use the mouse to wire the **inputs** and **outputs** of the different blocks. Inputs are located on the left side of the blocks, while outputs are located on the right side of the blocks.



When holding the mouse over an input or an output the mouse changes to the following symbol.

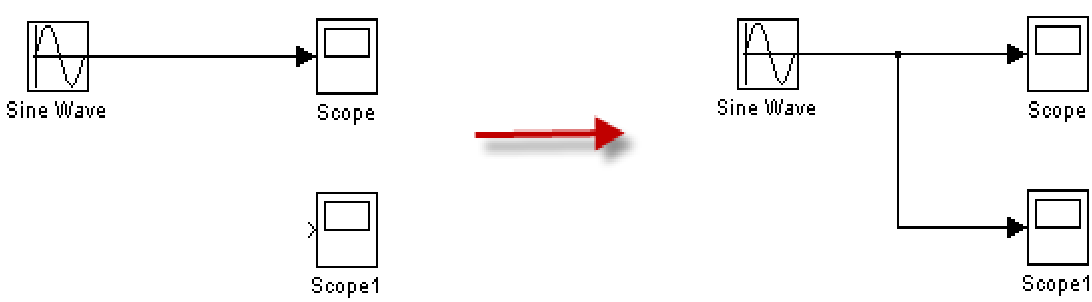
Use the mouse, while holding the left button down, to drag wires from the input to the output.

Automatic Block Connection:

Another wiring technique is to select the source block, then hold down the **Ctrl** key while left-clicking on the destination block.
Try the different techniques on the example above.

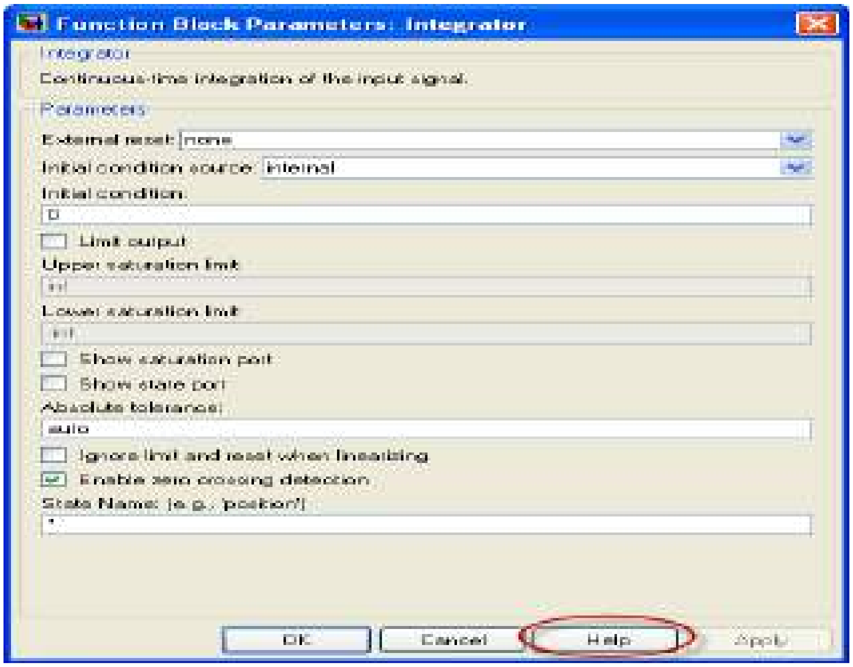
Connection from a wire to another block

If wire a connection from a wire to another block, like the example below, you need to hold down the **Ctrl** key while left-clicking on the wire and then to the input of the desired block.



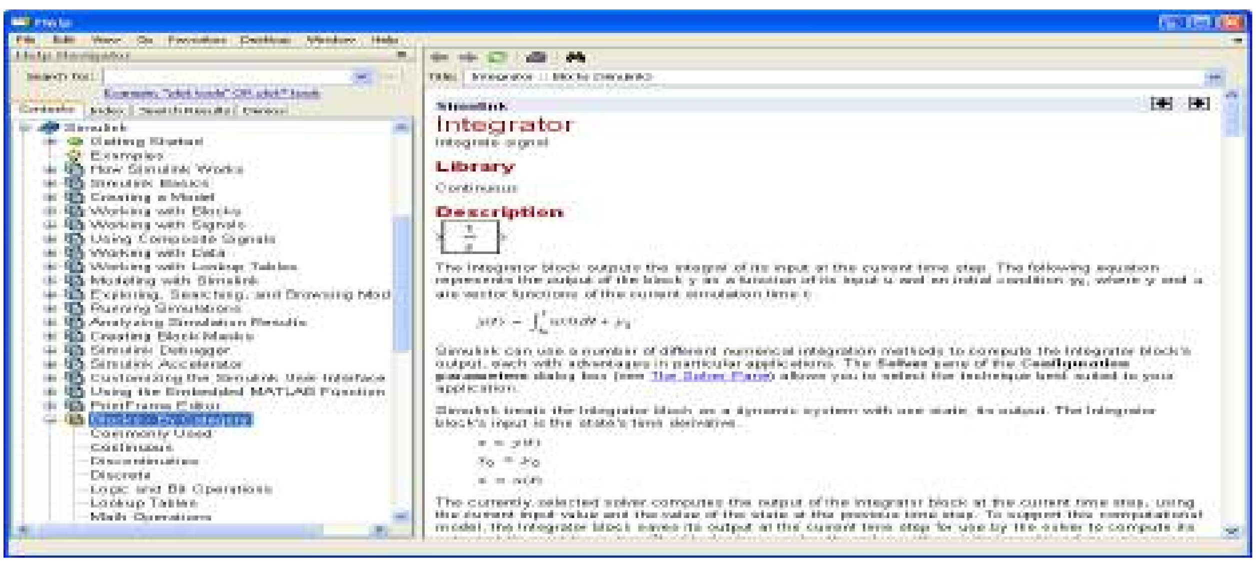
2.4 Help window

In order to see detailed information about the different blocks, use the built-in Help system.



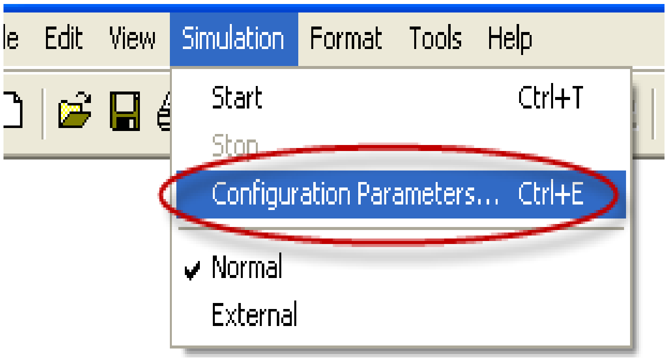
All standard blocks in Simulink have detailed Help. Click the Help button in the Block Parameter window for the specific block in order to get detailed help for that block.

The Help Window then appears with detailed information about the selected block

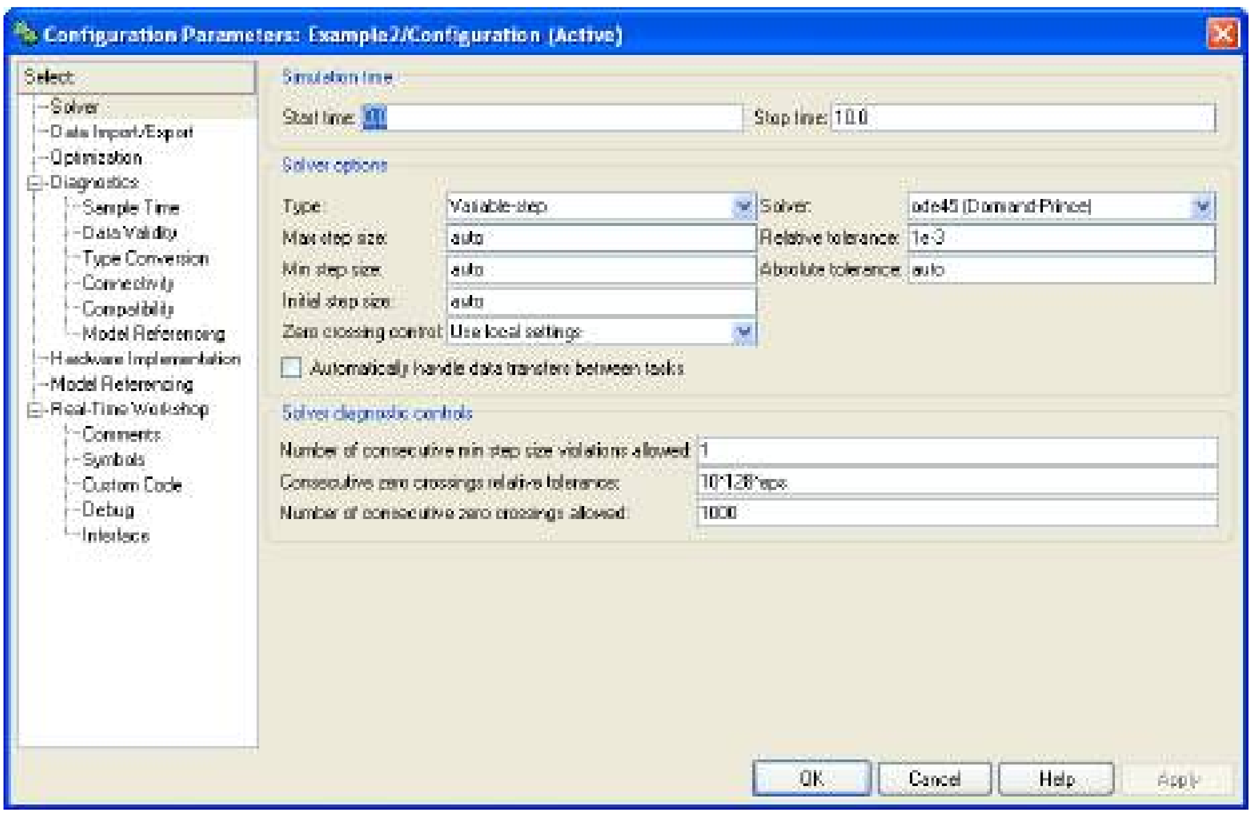


2.5 Configuration

There are lots of parameters you may want to configure regarding your simulation. Select "Configuration Parameters…" in the Simulation menu.



The following window appears:



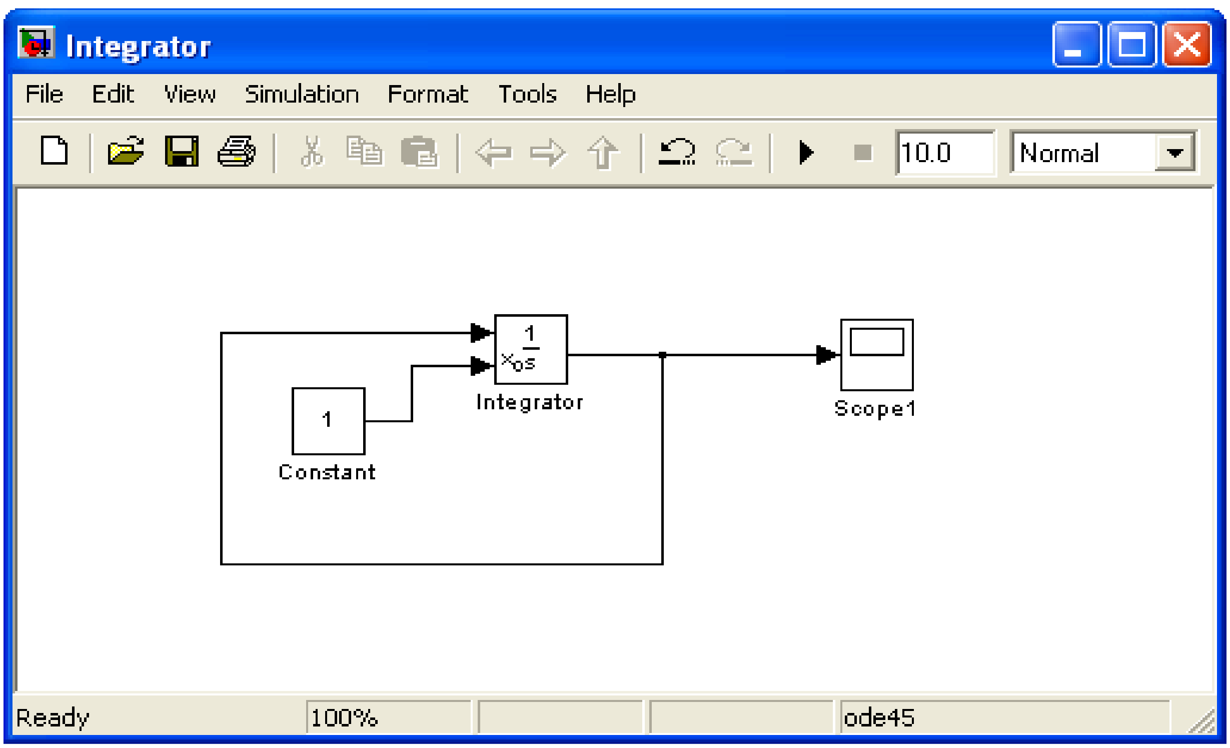
Here you set important parameters such as:

- Start and Stop time for the simulation
- What kind of Solver to be used (ode45, ode23 etc.)
- Fixed-step/Variable-step

Note! Each of the controls on the Configuration Parameters dialog box corresponds to a configuration parameter that you can set via the "**sim**" and "**simset**" commands. You will learn more about these commands later.
Solvers are numerical integration algorithms that compute the system dynamics over time using information contained in the model. Simulink provides solvers to support the simulation of a broad range of systems, including continuous-time (analog), discrete-time (digital), hybrid (mixed-signal), and multirate systems of any size.
2.6 Examples
Below we will go through some examples in order to illustrate how to create block diagrams and related functionality.

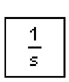
Example: Integrator with initial value

Create the following model (an integrator) and run the simulation:

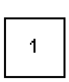


Step1: Place the blocks on the model surface

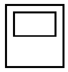
This example use the following blocks:



Integrator

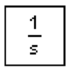


Constant



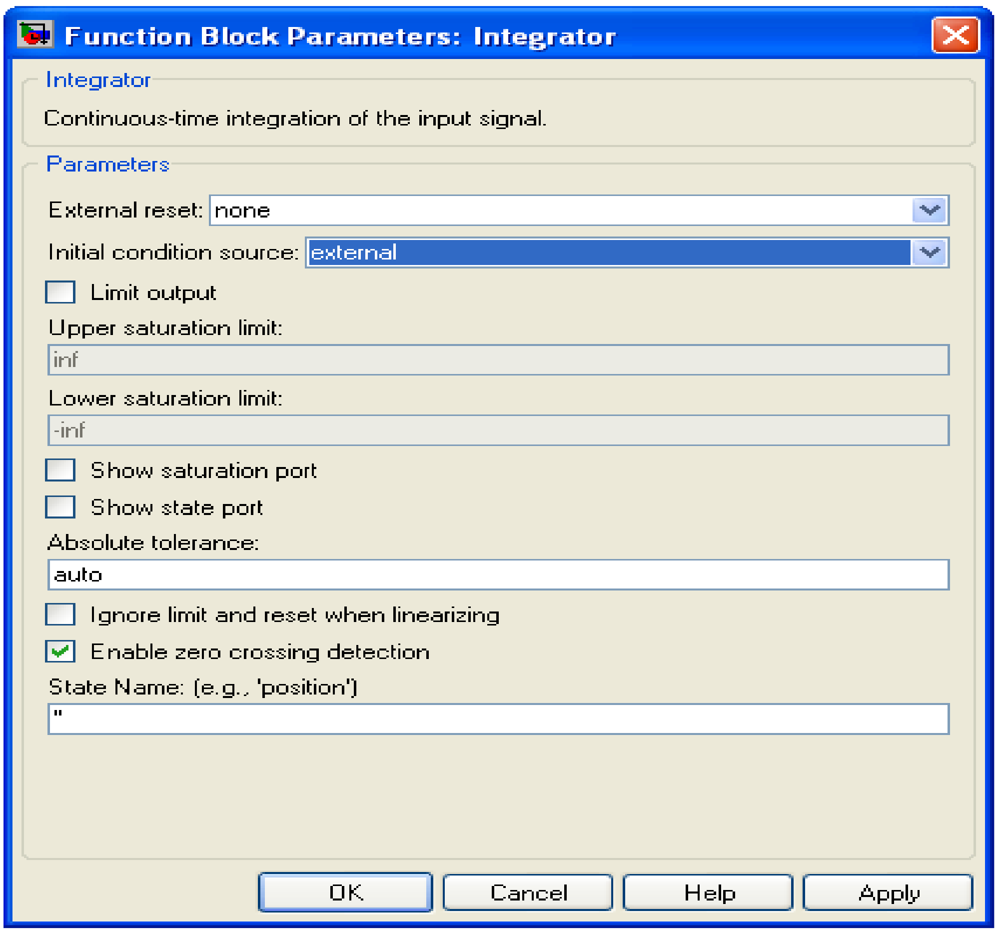
Scope

Step 2: Configuration

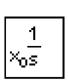


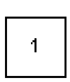
Integrator

Double-click on the Integrator block. The Parameter window for the Integrator block appears:



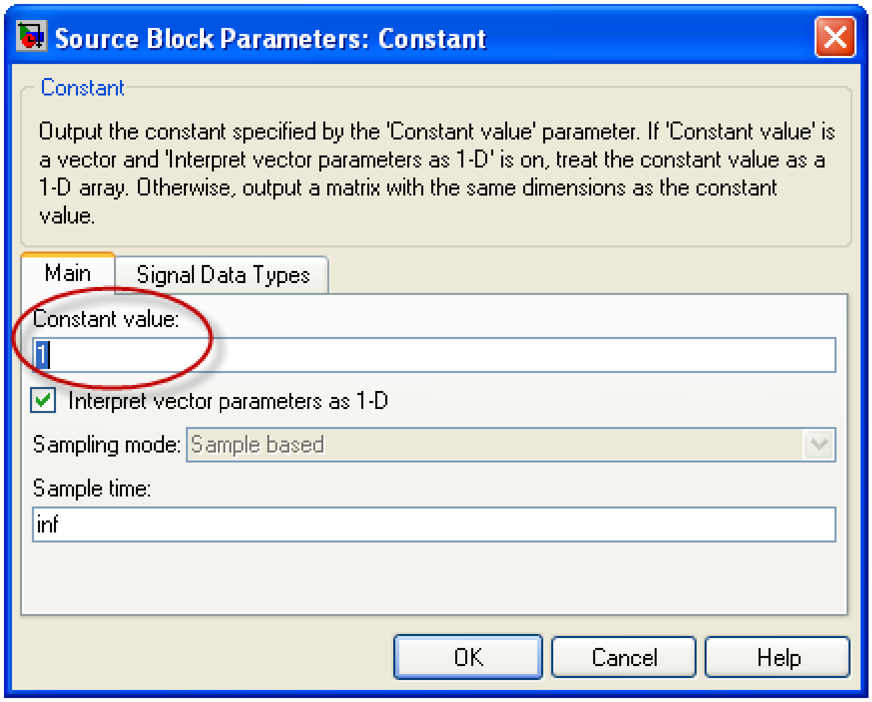
Select "**Initial condition source=external**". The Integrator block now looks like this:





Constant

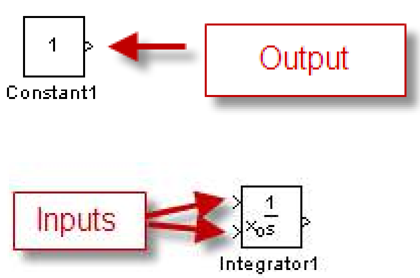
Double-click on the Constant block. The Parameter window for the Constant block appears:



In the Constant value field we type in the initial value for the integrator, e.g., type the value 1.

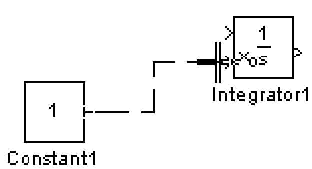
Step 3: Wiring

Use the mouse to wire the inputs and outputs of the different blocks.

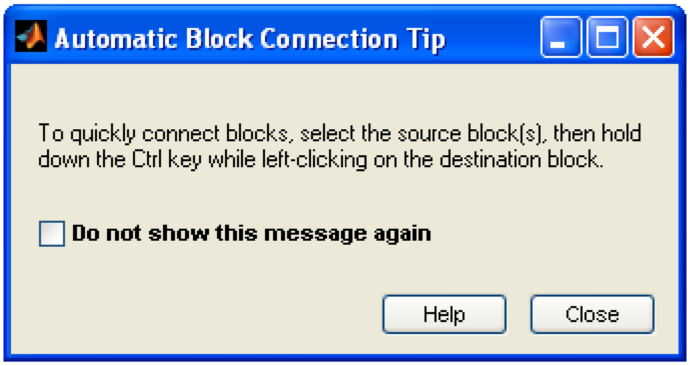


When holding the mouse over an input or an output the mouse change to the following symbol.

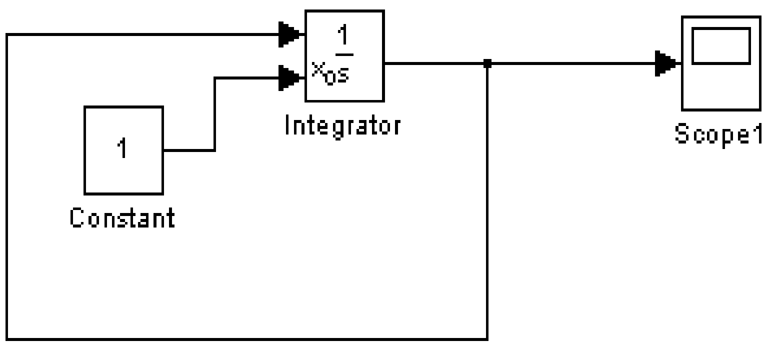
Draw a wire between the output on the Constant block to the lower input in the Integrator block, like this:



You could also do like this:

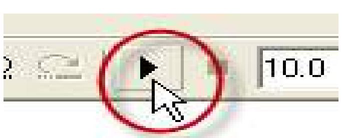


Wire the rest of the blocks together and you will get the following diagram:



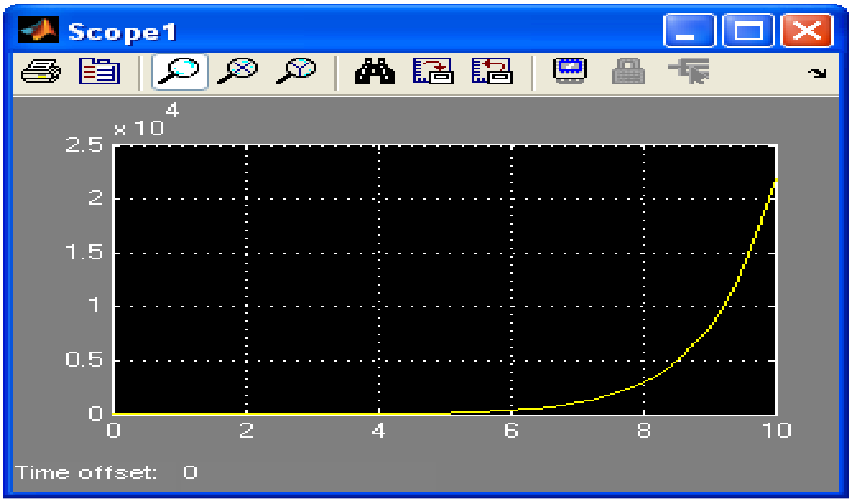
Step 4: Simulation

Start the simulation by clicking the “Start Simulation” icon in the Toolbar:



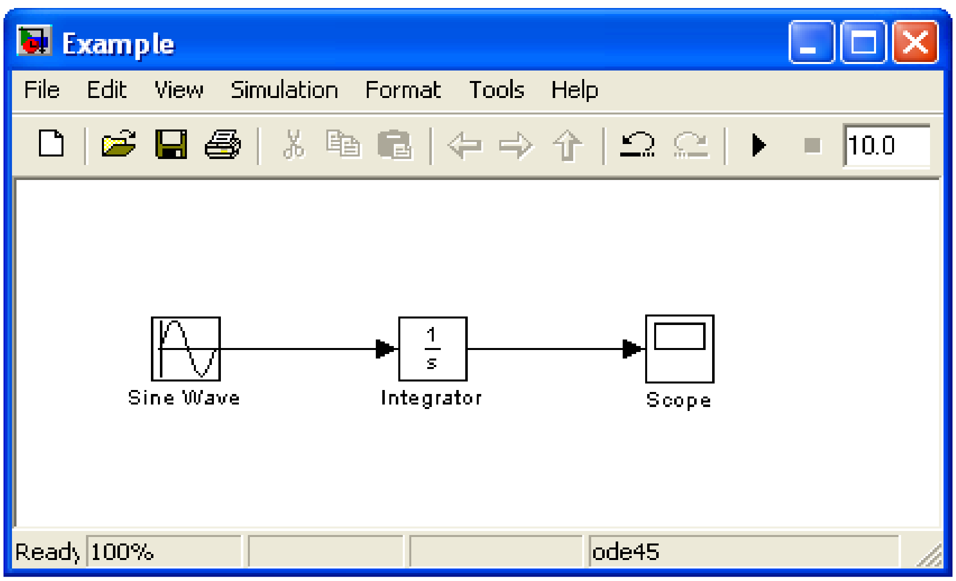
Step 5: The Results

Double-click in the Scope block in order to see the simulated result:

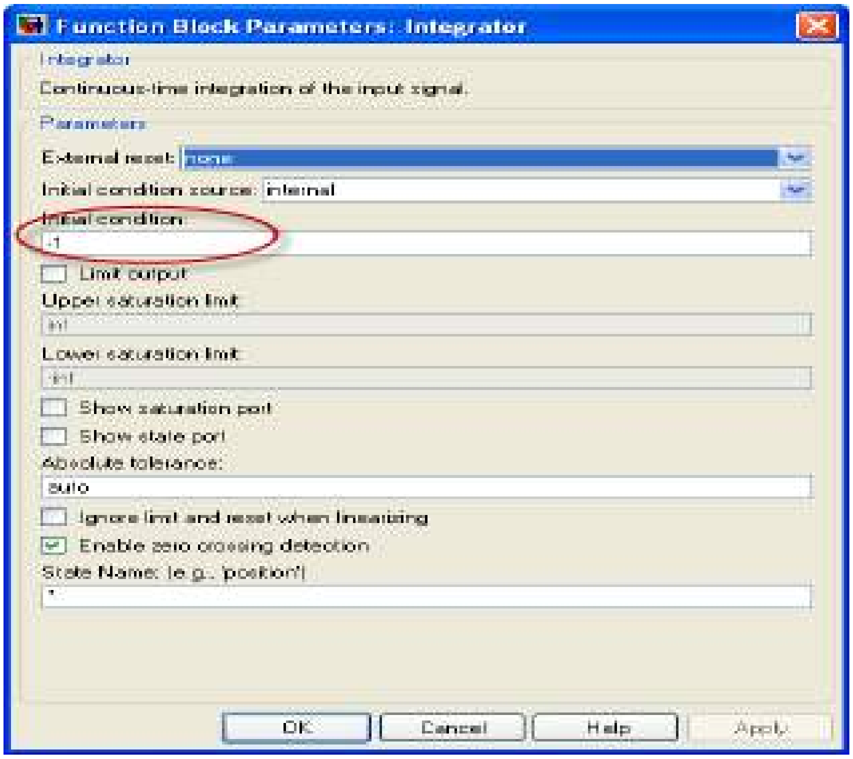


Example: Sine Wave

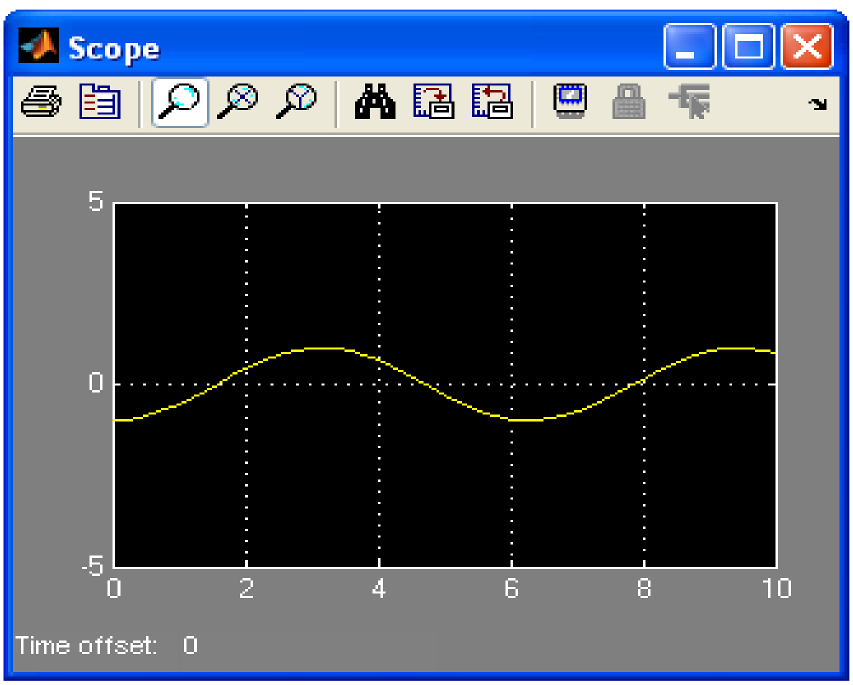
Create the block diagram as shown below:



Set the following parameter for the Integrator block:



The result should be like this:



EXPERIMENT NO. 2

AIM: Study of various Electrical Toolbox i.e. Power System, Power Electronics, Control system, Electrical Measurement, Flexible AC Transmission.

Theory: Sim Power Systems provides component libraries and analysis tools for modeling and simulating electrical power systems. The libraries offer models of electrical power components, including three-phase machines, electric drives, and components for applications such as flexible AC transmission systems (FACTS) and renewable energy systems. Harmonic analysis, calculation of total harmonic distortion (THD), load flow, and other key electrical power system analyses are automated.

Sim Power Systems models can be used to develop control systems and test system-level performance. You can parameterize your models using MATLAB® variables and expressions, and design control systems for your electrical power system in Simulink. You can add mechanical, hydraulic, pneumatic, and other components to the model using Simscape™ and test them together in a single simulation environment. To deploy models to other simulation environments, including hardware-in-the-loop (HIL) systems, Sim Power Systems supports C-code generation.

Key Features

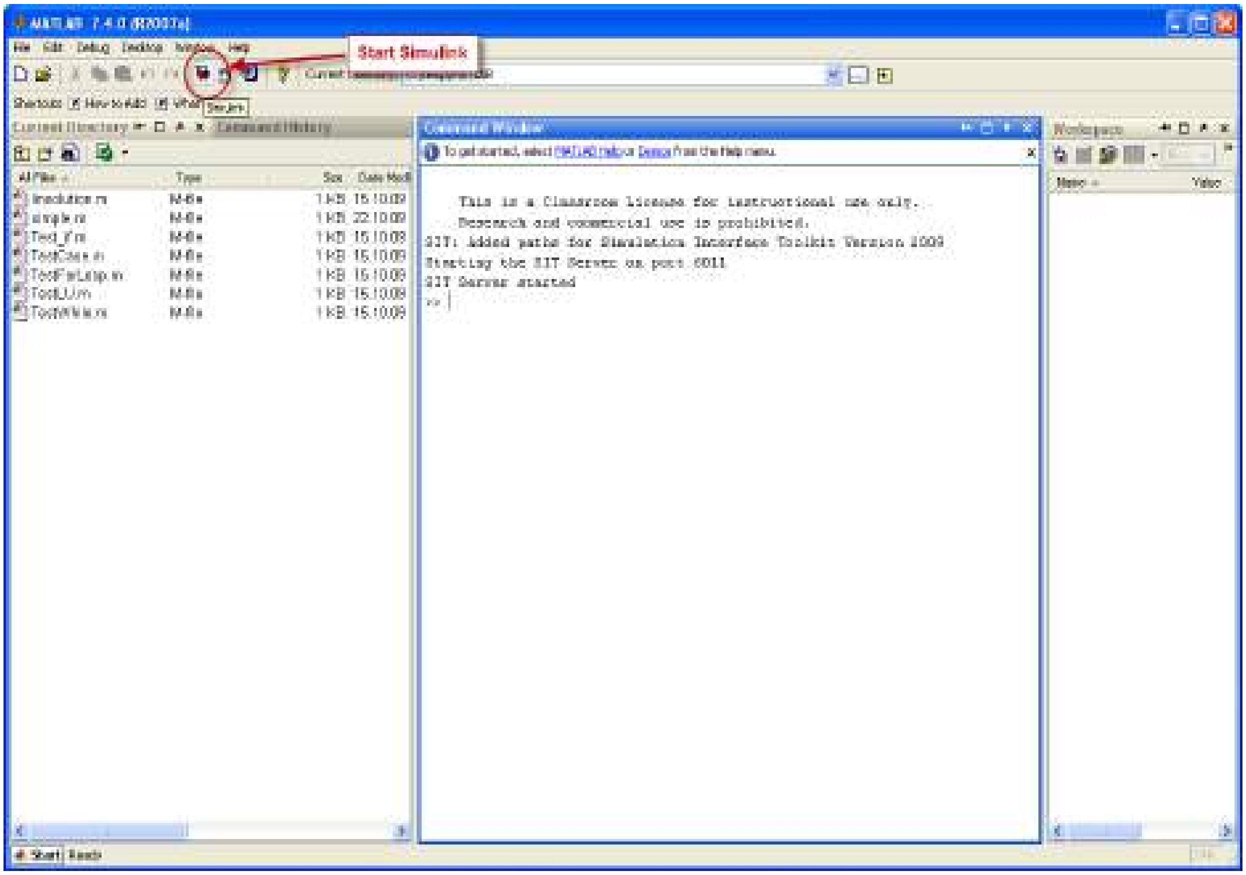
- Libraries of application-specific models, including models of common AC and DC electric drives, flexible AC transmission systems (FACTS), and renewable energy systems
- Discretization and phasor simulation models for faster model execution
- Ideal switching algorithm for accelerated simulation of power electronic devices
- Analysis methods for obtaining state-space representations of circuits and computing load flow for machines
- Basic models for developing key electrical technologies
- Ability to extend component libraries using the Simscape language
- Support for C-code generation

Start using Simulink

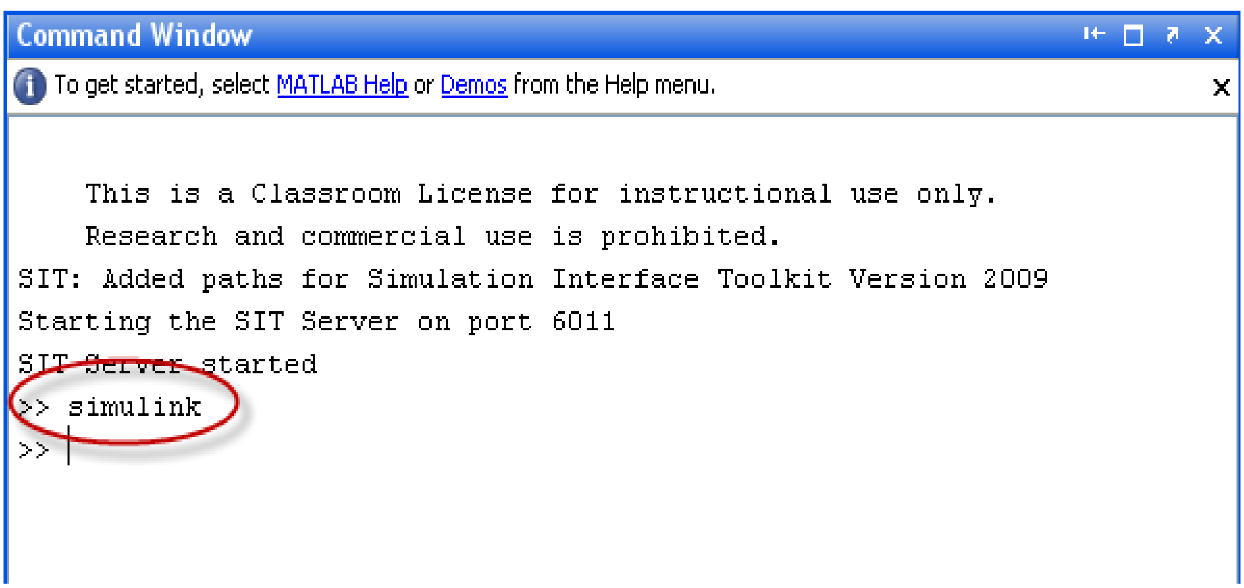
You start Simulink from the MATLAB IDE:

Open MATLAB and select the Simulink icon in the Toolbar:

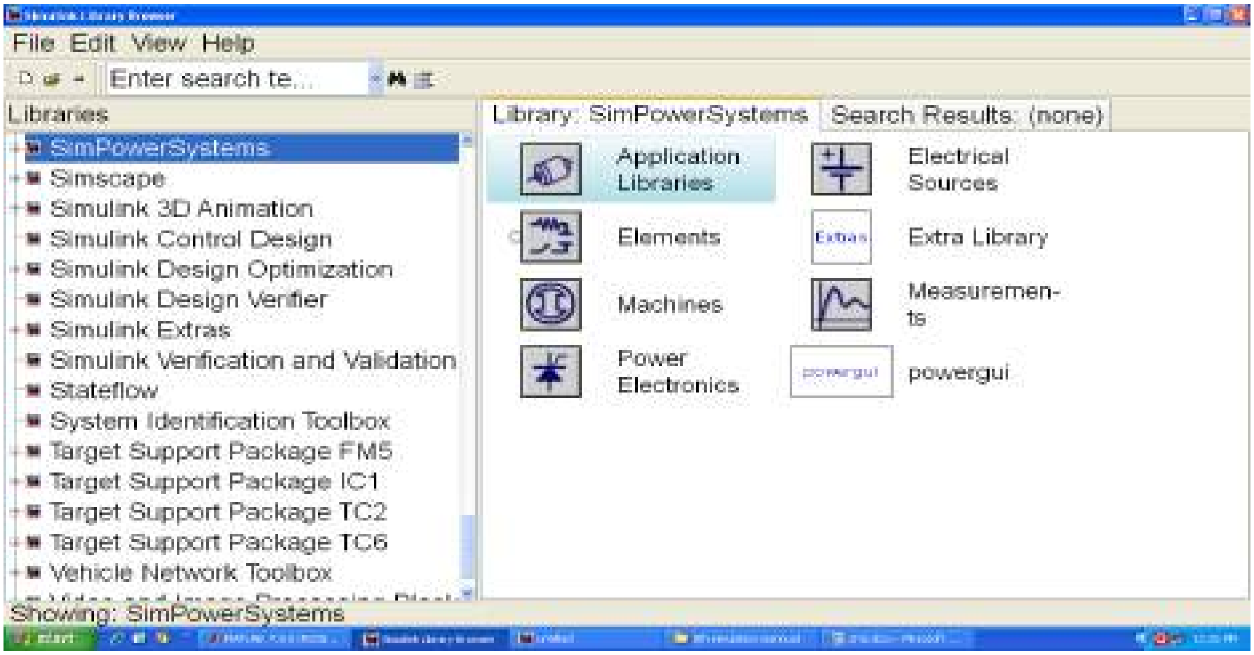
Or type

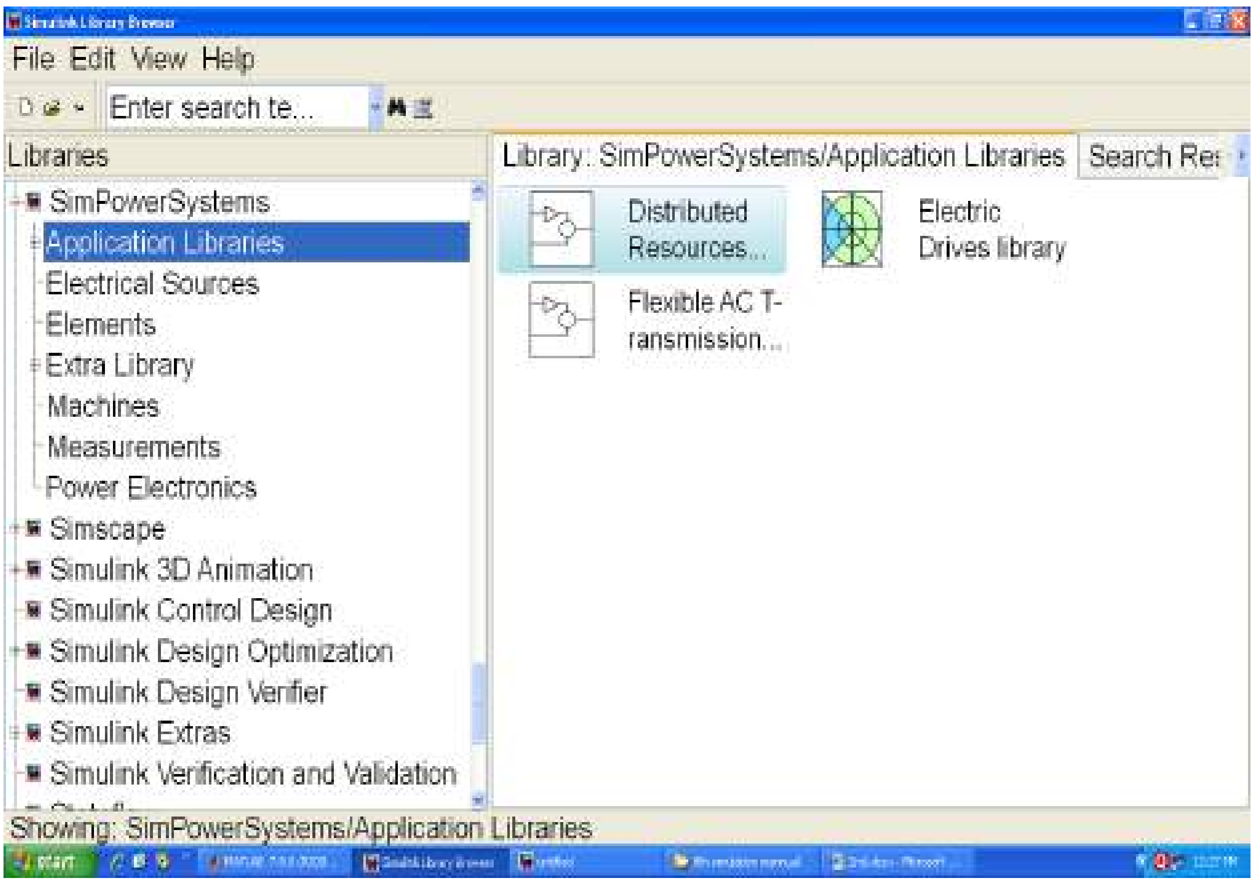


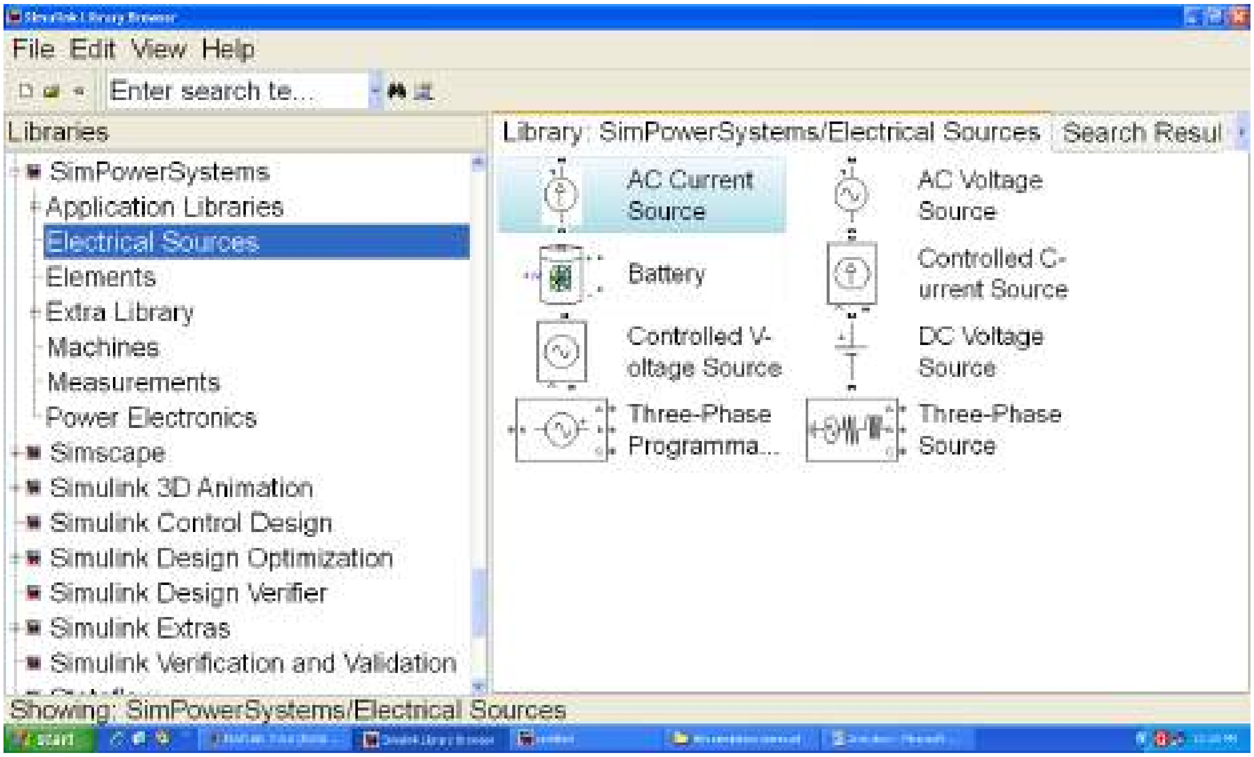
- Or type "**simulink**" in the Command window, like this:

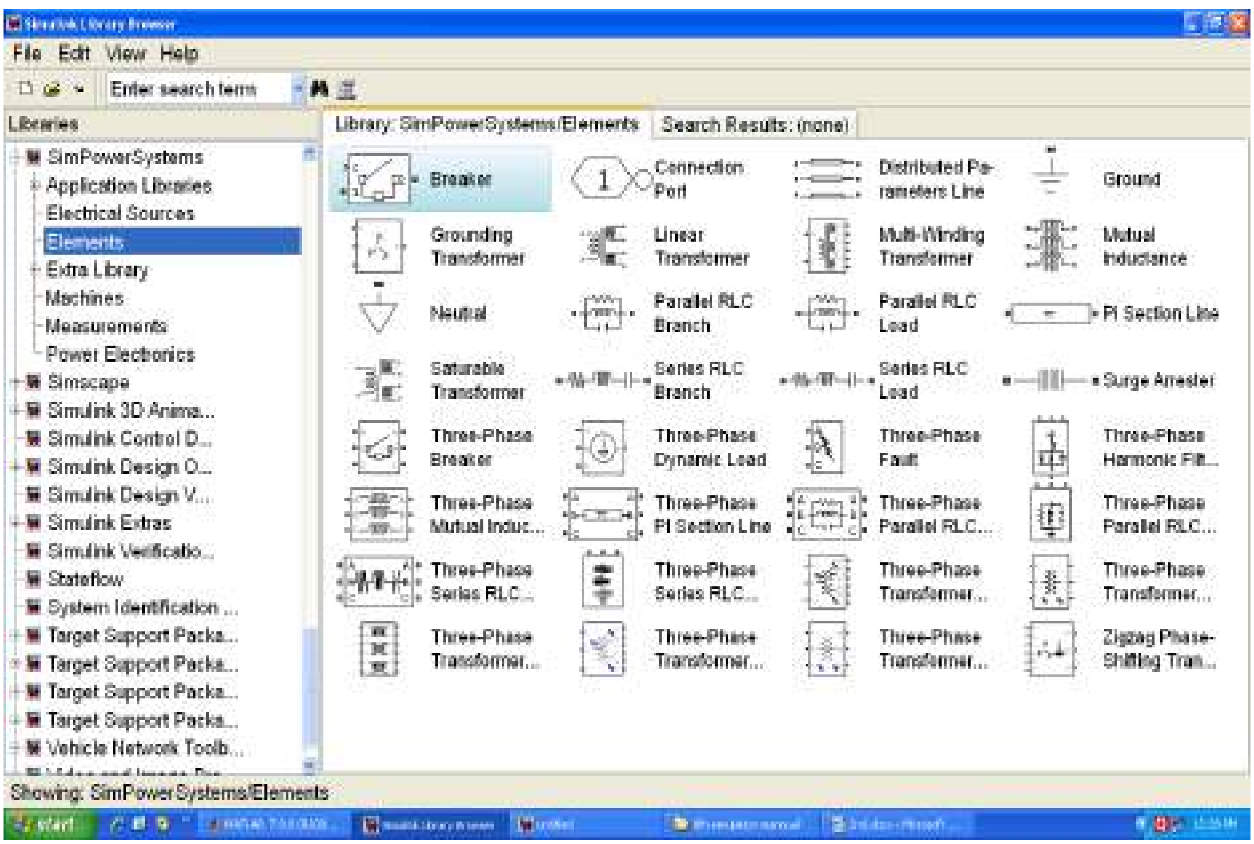


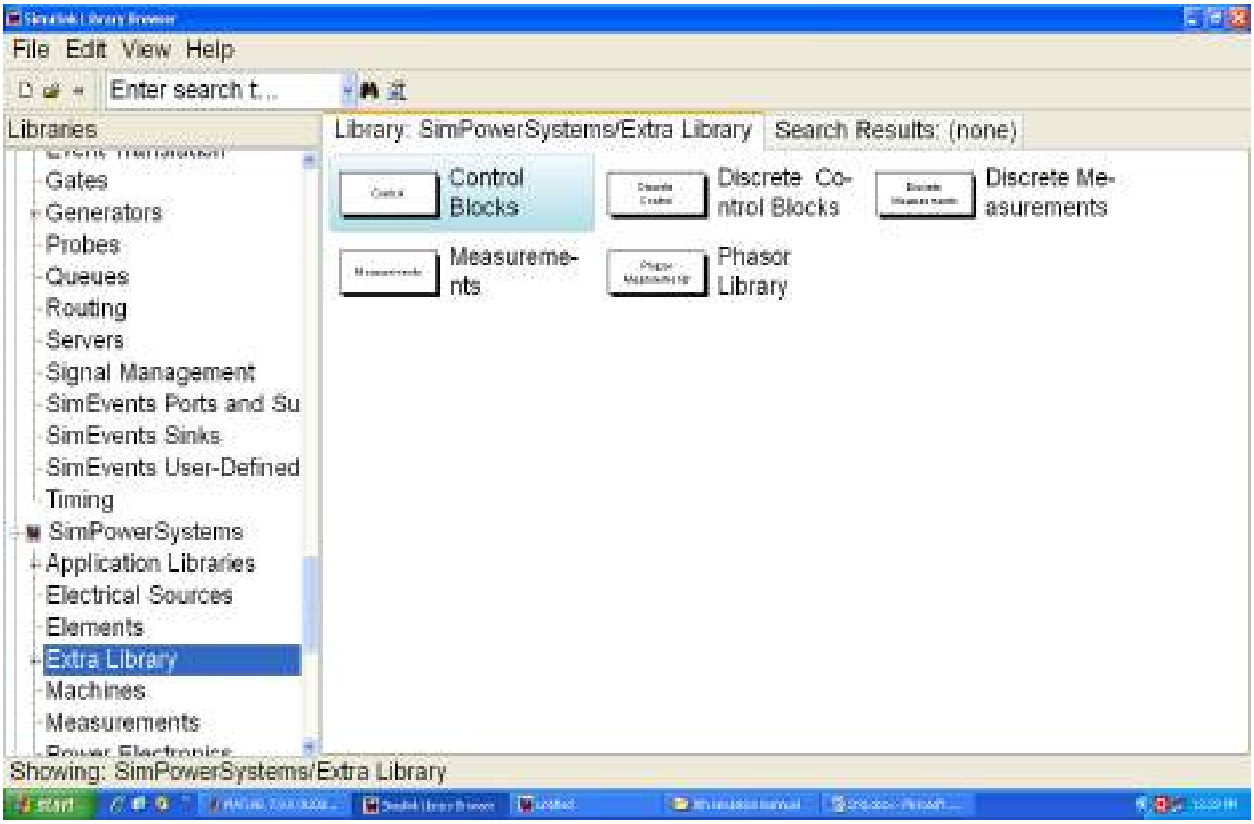
- Then the following window appears (**Simulink Library Browser**):

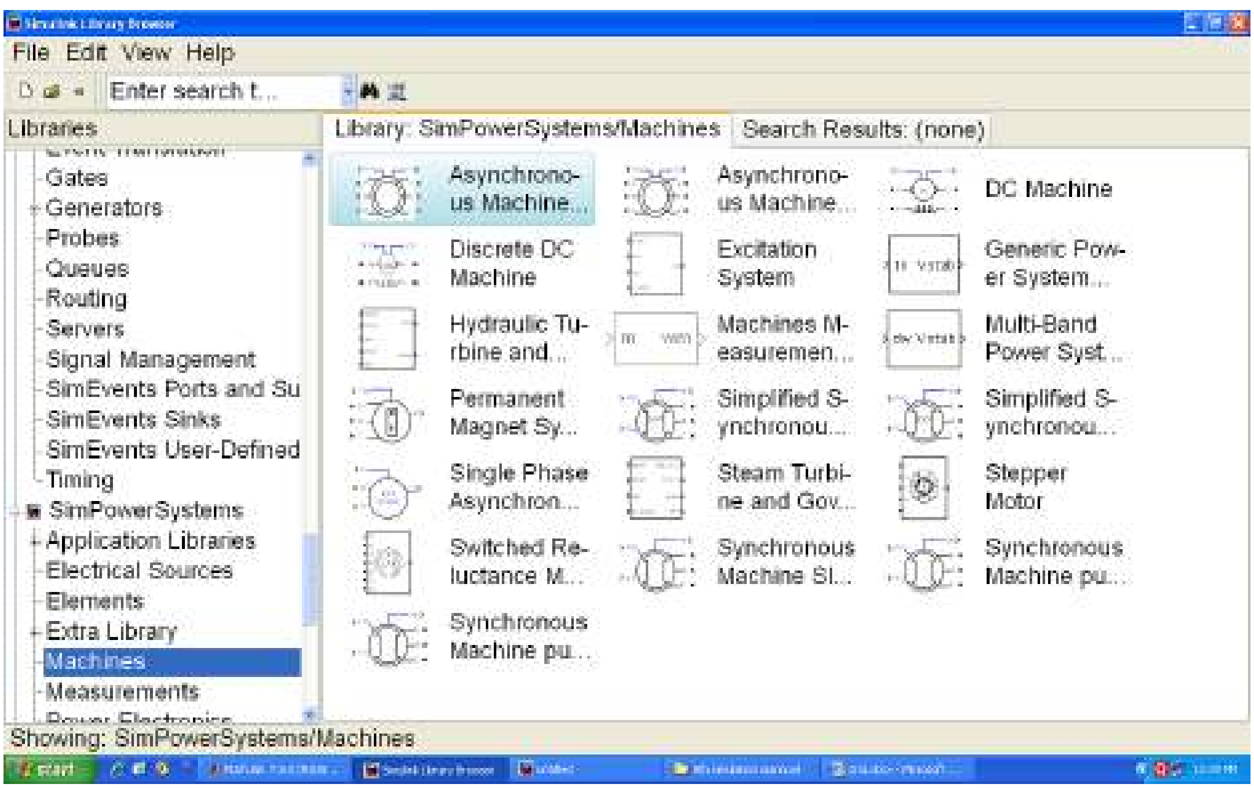

Simulink Library Browser
File Edit View Help
Enter search te...
Libraries
Library: SimPowerSystems
Search Results: (none)
SimPowerSystems
Simscape
Simulink 3D Animation
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Extras
Simulink Verification and Validation
Stateflow
System Identification Toolbox
Target Support Package FM5
Target Support Package IC1
Target Support Package TC2
Target Support Package TC6
Vehicle Network Toolbox
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measuremen-ts
Power Electronics
powergui
Showing: SimPowerSystems

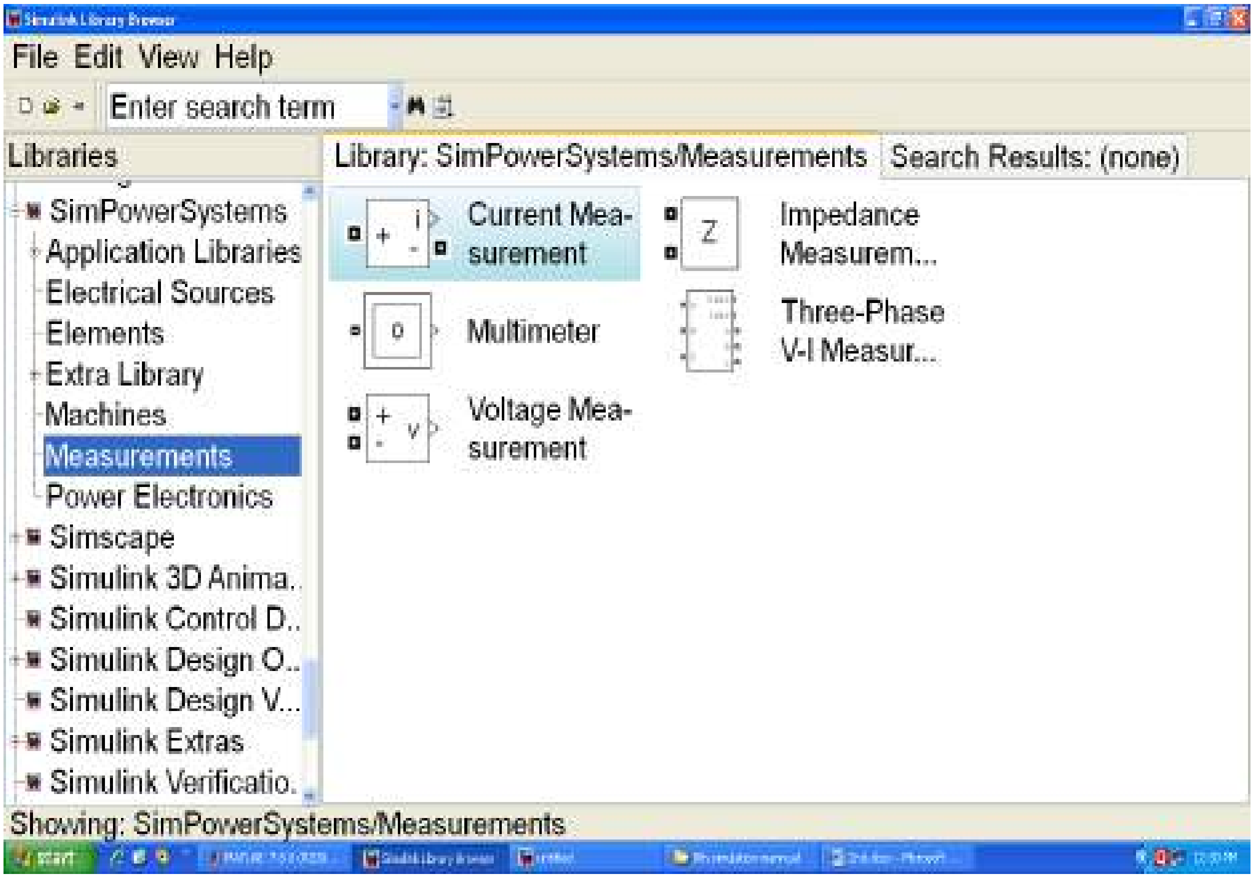

Simulink Library Browser
File Edit View Help
Enter search te...
Libraries
Library: SimPowerSystems/Application Libraries
SimPowerSystems
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measurements
Power Electronics
Simscape
Simulink 3D Animation
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Extras
Simulink Verification and Validation
Distributed Resources...
Electric Drives library
Flexible AC T-ransmission...
Showing: SimPowerSystems/Application Libraries

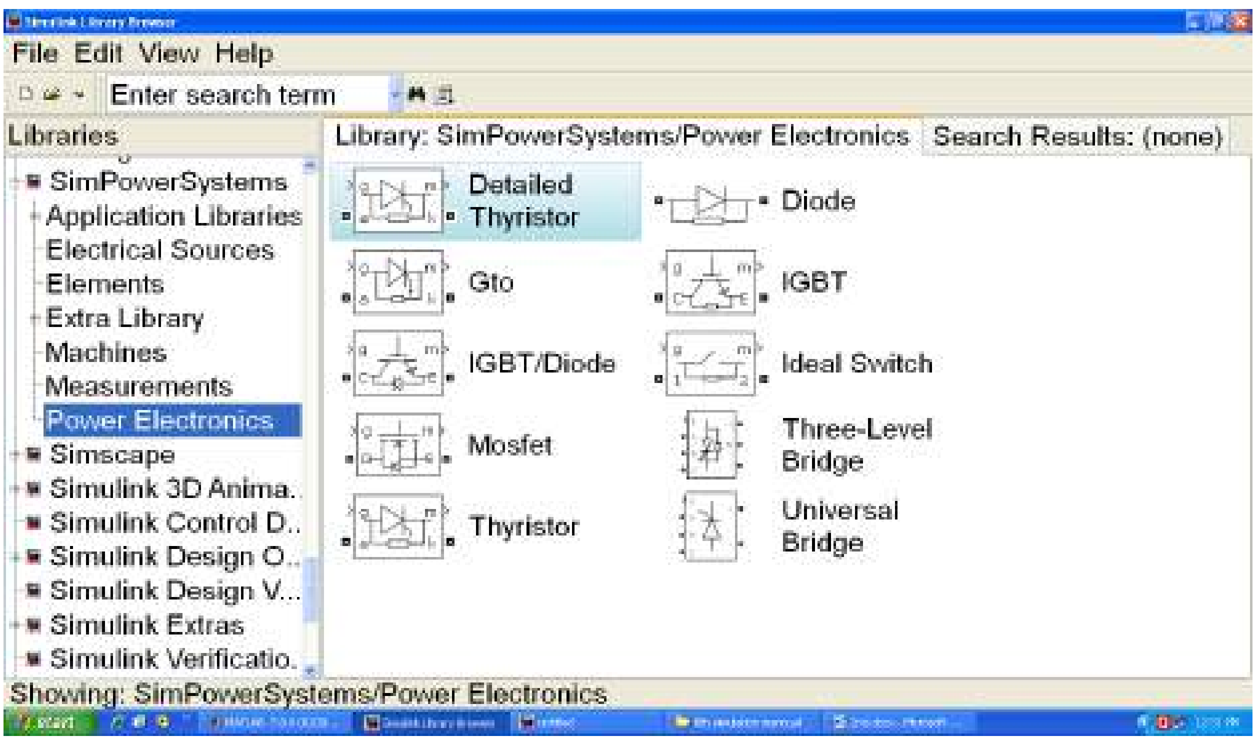

File Edit View Help
Enter search te...
Libraries
Library: SimPowerSystems/Electrical Sources
Search Resul
SimPowerSystems
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measurements
Power Electronics
Simscape
Simulink 3D Animation
Simulink Control Design
Simulink Design Optimization
Simulink Design Verifier
Simulink Extras
Simulink Verification and Validation
AC Current Source
AC Voltage Source
Battery
Controlled C-urrent Source
Controlled V-oltage Source
DC Voltage Source
Three-Phase Programma...
Three-Phase Source
Showing: SimPowerSystems/Electrical Sources


File Edit View Help
Enter search term
Libraries
Library: SimPowerSystems/Elements
Search Results: (none)
SimPowerSystems
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measurements
Power Electronics
Simscape
Simulink 3D Anima...
Simulink Control D...
Simulink Design O...
Simulink Design V...
Simulink Extras
Simulink Verificatio...
Stateflow
System Identification ...
Target Support Packa...
Target Support Packa...
Target Support Packa...
Target Support Packa...
Vehicle Network Toolb...
Breaker
Connection Port
Distributed Parameters Line
Ground
Grounding Transformer
Linear Transformer
Multi-Winding Transformer
Mutual Inductance
Neutral
Parallel RLC Branch
Parallel RLC Load
PI Section Line
Saturable Transformer
Series RLC Branch
Series RLC Load
Surge Arrester
Three-Phase Breaker
Three-Phase Dynamic Load
Three-Phase Fault
Three-Phase Harmonic Filt...
Three-Phase Mutual Induc...
Three-Phase PI Section Line
Three-Phase Parallel RLC...
Three-Phase Parallel RLC...
Three-Phase Series RLC...
Three-Phase Series RLC...
Three-Phase Transformer...
Three-Phase Transformer...
Three-Phase Transformer...
Three-Phase Transformer...
Three-Phase Transformer...
Zigzag Phase-Shifting Tran...
Showing: SimPowerSystems/Elements


File Edit View Help
Enter search t...
Libraries
Library: SimPowerSystems/Extra Library
Search Results: (none)
Gates
Generators
Probes
Queues
Routing
Servers
Signal Management
SimEvents Ports and Su
SimEvents Sinks
SimEvents User-Defined
Timing
SimPowerSystems
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measurements
Control Blocks
Discrete Control Blocks
Discrete Measurements
Measurements
Phasor Library
Showing: SimPowerSystems/Extra Library
start


File Edit View Help
Enter search t...
Libraries
Library: SimPowerSystems/Machines
Search Results: (none)
Gates
Generators
Probes
Queues
Routing
Servers
Signal Management
SimEvents Ports and Su
SimEvents Sinks
SimEvents User-Defined
Timing
SimPowerSystems
Application Libraries
Electrical Sources
Elements
Extra Library
Machines
Measurements
Asynchronous Machine...
Asynchronous Machine...
DC Machine
Discrete DC Machine
Excitation System
Generic Power System...
Hydraulic Turbine and...
Machines Measuremen...
Multi-Band Power Syst...
Permanent Magnet Sy...
Simplified Synchronou...
Simplified Synchronou...
Single Phase Asynchron...
Steam Turbine and Gov...
Stepper Motor
Switched Reluctance M...
Synchronous Machine SI...
Synchronous Machine pu...
Synchronous Machine pu...
Showing: SimPowerSystems/Machines
start





EXPERIMENT NO. 3

AIM: MODELING AND SIMULATION OF HALF WAVE RECTIFIER.

Theory:

Single-Phase Rectifier:

This demonstration illustrates use of the Universal Bridge, Multimeter, and Discrete System blocks

Contents

- **Circuit Description**
- **Demonstration**

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 transformers. 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.

Below the Half wave rectifier circuit shown, so drag the boxes which are required for circuit from simulink library and join them according to as given figure.

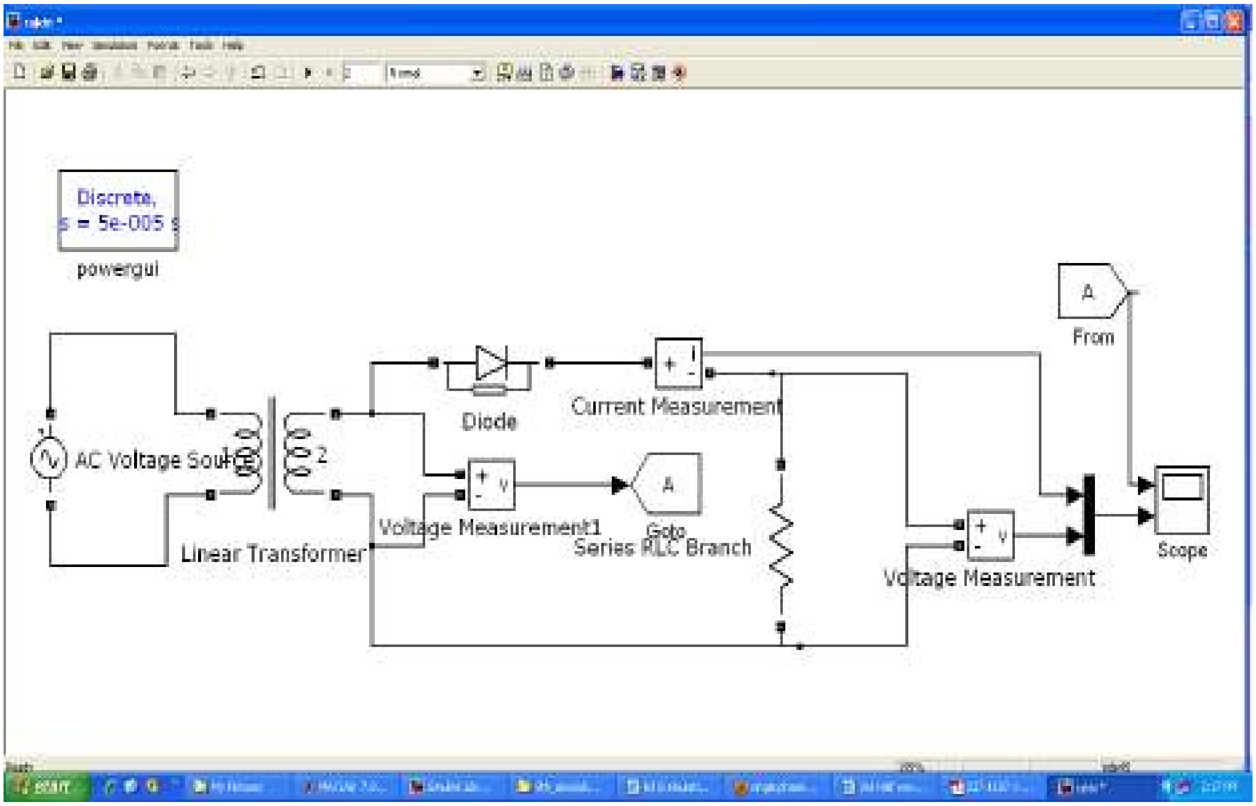
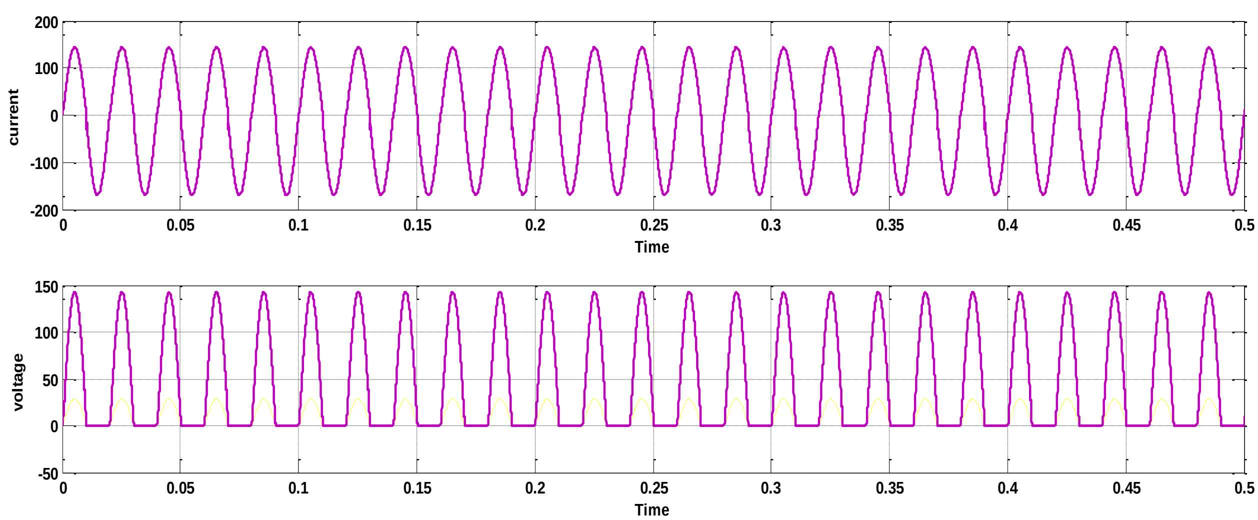


Fig. 1 single phase half wave rectifier

RESULTS:



Conclusion: In above result we can see the negative cycle is zero that means half wave is rectified.

EXPERIMENT NO. 4

AIM: MODELING AND SIMULATION OF FULLWAVE RECTIFIER.

Theory:

Full Wave Rectifier:

This demonstration illustrates use of the Ideal Switching Device solution method to simulate a full wave rectifier using ideal diodes.

G. Sybille (Hydro-Quebec)

Contents

- **Demonstration**

Demonstration: Open the powergui 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 tell Sim Power Systems 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 perfectly ideal diodes, the **Disable On resistance** and **Disable Forward votage** 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 Sim Power Systems 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 Sim power Systems will prompt an error message. In this case if you attempt to use the ode45 solver instead of recommended Ode23tb, you will observe numerical oscillations unless you use a Relative tolerance of 1e-6, which results in slow simulation.

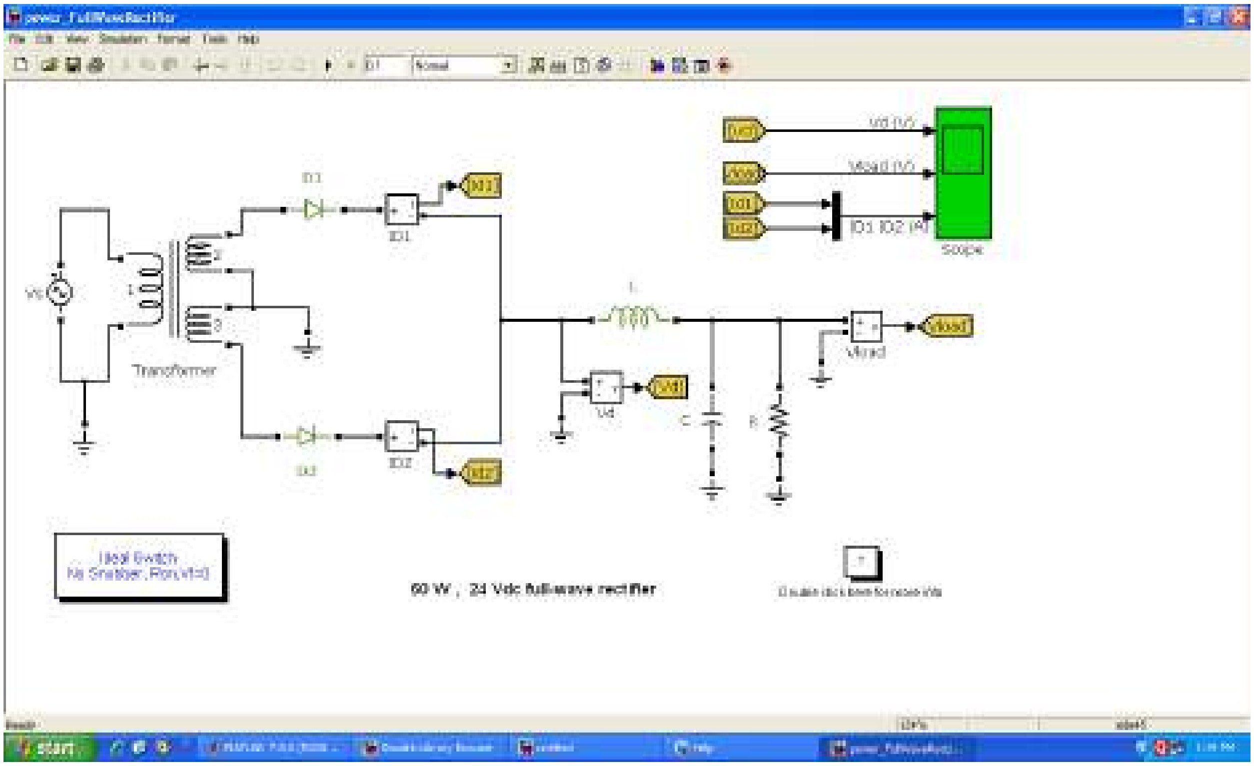
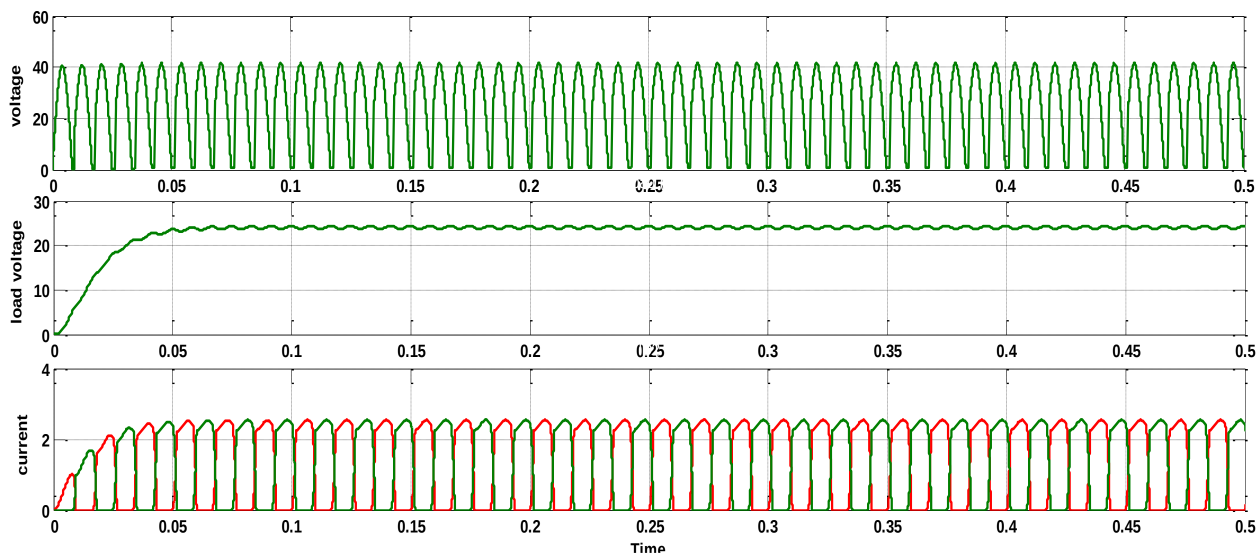


Fig. 1 full wave rectifier

RESULTS:



Conclusion: In above result we can see the negative cycle and positive cycle wave both are rectified that means full wave is rectified.

EXPERIMENT NO. 5

AIM: MODELING AND SIMULATION OF HALF WAVE CONTROLLED RECTIFIER**.**

Theory:

INTRODUCTION:

Power electronics technology encompasses the use of electronic components, the application of circuit theory and design techniques, and the development of analytical tools toward efficient electronic conversion, control, and conditioning of electric power. The typical undergraduate syllabus will have topics like: uncontrolled and controlled rectifiers with R, RL loads; choppers; single-phase and 3-phase inverters; AC voltage controllers, etc. [1]. The basic information of MATLAB i.e. what is MATLAB, MATLAB Toolbox & SIMULINK, MATLAB Advantages & its application, is presented in [2]. In this paper the characteristics of SCR, simulation of single phase half wave & full wave controlled & uncontrolled rectifiers are presented on MATLAB software.

MATLAB/SIMULINK FOR POWER ELECTRONICS:

The following section will looks at how the modeling and simulation of a power electronic converter can be carried out using MATLAB/SIMULINK software. Firstly open MATLAB then SIMULINK. A SIMULINK library browser will open From this select the blocks which are used to achieve the modeling as follow: first from SIMULINK from commonly used blocks we choose scope (used for show output waveform), bus selector (used for measure signal for example voltage and current waveform in case of diode or SCR) from sources choose pulse generator (used to providing pulse for GATE of THYRISTOR). In SIMULINK library browser Sim power systems is first chosen & select powerugi (used to select the supply type i.e. continues or discrete). In Sim power systems various sub library are present for electrical engineering application. From power electronics choose THYRISTOR, MOSFET, GTO etc. Power electronics devices from electrical sources choose supply i.e. AC /DC voltage supply. Then from elements choose RLC series branch for R, L, C, RL, RC or RLC load. Then from measurement choose current & voltage measurement. All the blocks listed above are used in this paper for simulation purpose. By right click we can change the property of blocks. Fig. 1 show the all blocks listed above.

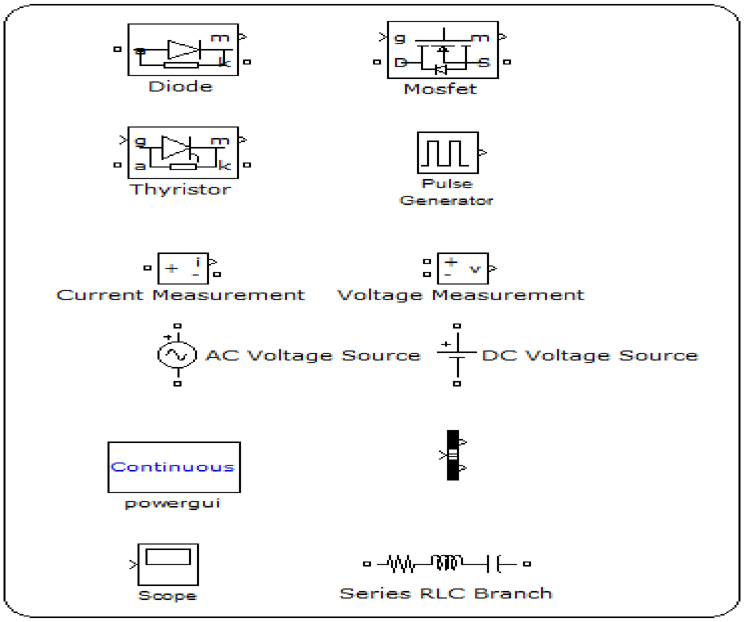


Fig. 1 Simulation Blocks

Controlled half wave Rectifier with RL Load:

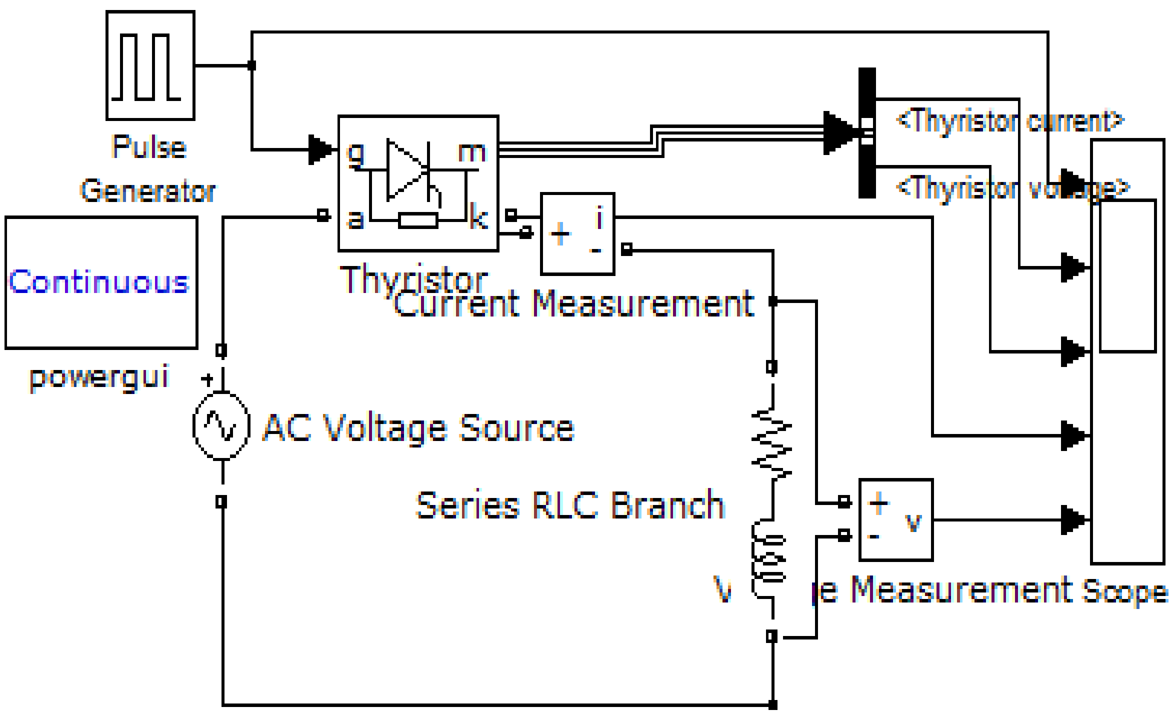


Fig. 2 Simulation Model for Controlled Half wave Rectifier with RL Load

RESULTS:

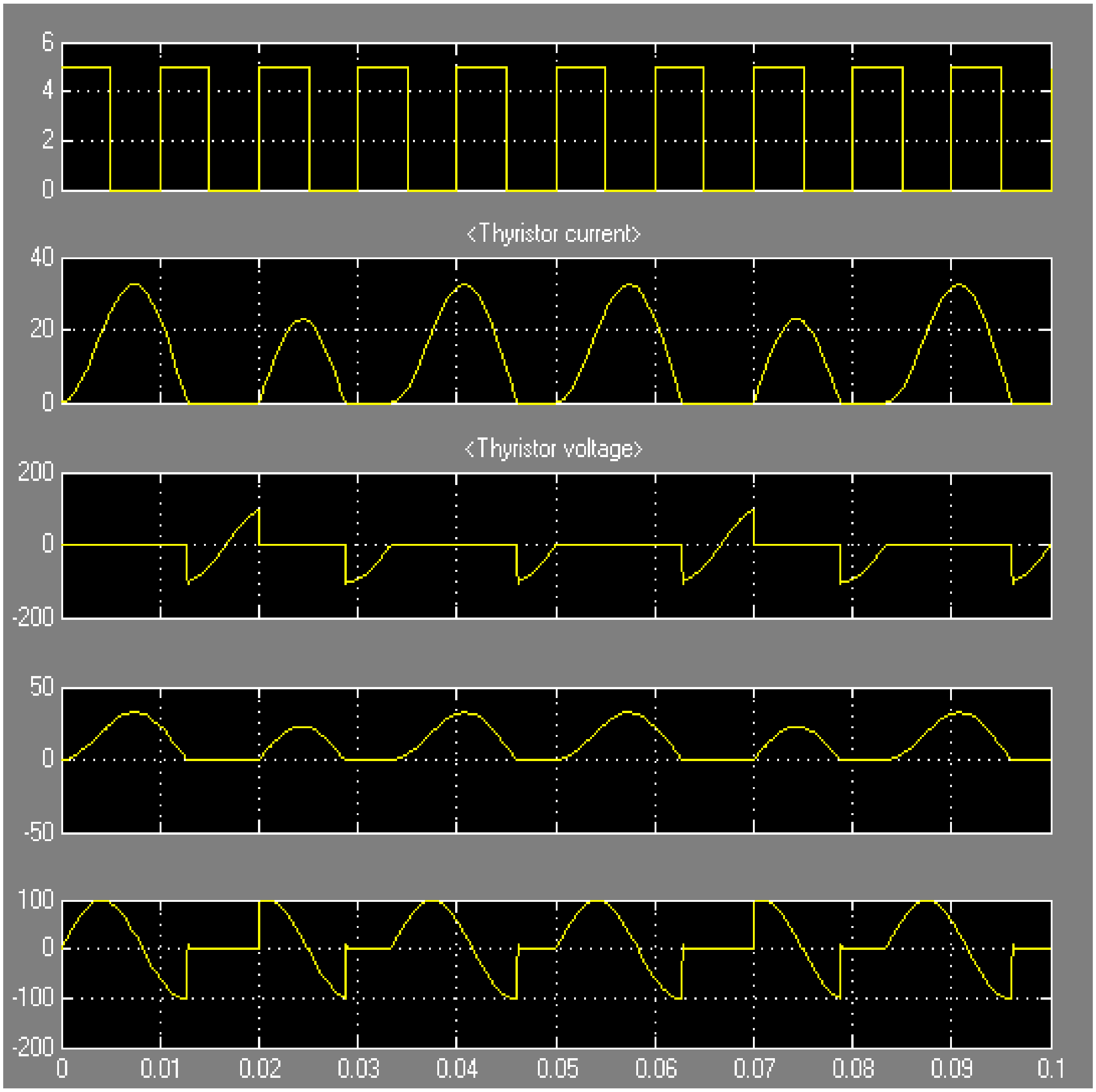


Fig. 3 Waveforms for Controlled Half wave Rectifier with RL Load

CONCLUSION:

The following conclusion may be derived when using SIMULINK in teaching power electronics courses: As power electronic systems are getting more complex today, the simulation used for education is requiring more features. Some directions in the development of simulation are discussed in this paper, with the help of present model students can simulate the power electronics circuit with various load & conditions.

EXPERIMENT NO. 6

AIM: MODELING AND SIMULATION OF FULL WAVE CONTROLLED RECTIFIER**.**

Theory:

INTRODUCTION:

Power electronics technology encompasses the use of electronic components, the application of circuit theory and design techniques, and the development of analytical tools toward efficient electronic conversion, control, and conditioning of electric power. The typical undergraduate syllabus will have topics like: uncontrolled and controlled rectifiers with R, RL loads; choppers; single-phase and 3-phase inverters; AC voltage controllers, etc. [1]. The basic information of MATLAB i.e. what is MATLAB, MATLAB Toolbox & SIMULINK, MATLAB Advantages & its application, is presented in [2]. In this paper the characteristics of SCR, simulation of single phase half wave & full wave controlled & uncontrolled rectifiers are presented on MATLAB software.

MATLAB/SIMULINK FOR POWER ELECTRONICS:

The following section will looks at how the modeling and simulation of a power electronic converter can be carried out using MATLAB/SIMULINK software. Firstly open MATLAB then SIMULINK. A SIMULINK library browser will open From this select the blocks which are used to achieve the modeling as follow: first from SIMULINK from commonly used blocks we choose scope (used for show output waveform), bus selector (used for measure signal for example voltage and current waveform in case of diode or SCR) from sources choose pulse generator (used to providing pulse for GATE of THYRISTOR). In SIMULINK library browser Sim power systems is first chosen & select powerugi (used to select the supply type i.e. continues or discrete). In Sim power systems various sub library are present for electrical engineering application. From power electronics choose THYRISTOR, MOSFET, GTO etc. Power electronics devices from electrical sources choose supply i.e. AC /DC voltage supply. Then from elements choose RLC series branch for R, L, C, RL, RC or RLC load. Then from measurement choose current & voltage measurement. All the blocks listed above are used in this paper for simulation purpose. By right click we can change the property of blocks. Fig. 1 show the all blocks listed above.

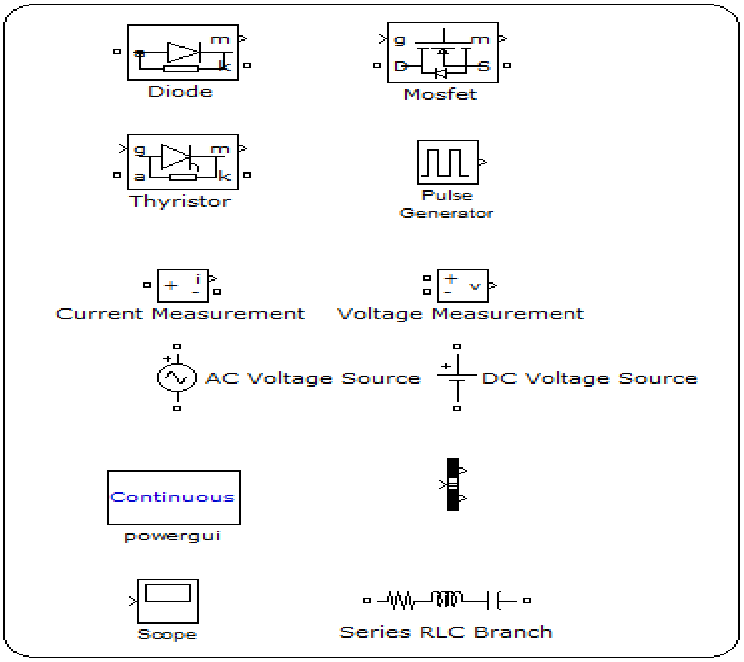


Fig. 1 Simulation Blocks

Controlled half wave Rectifier with RL Load:

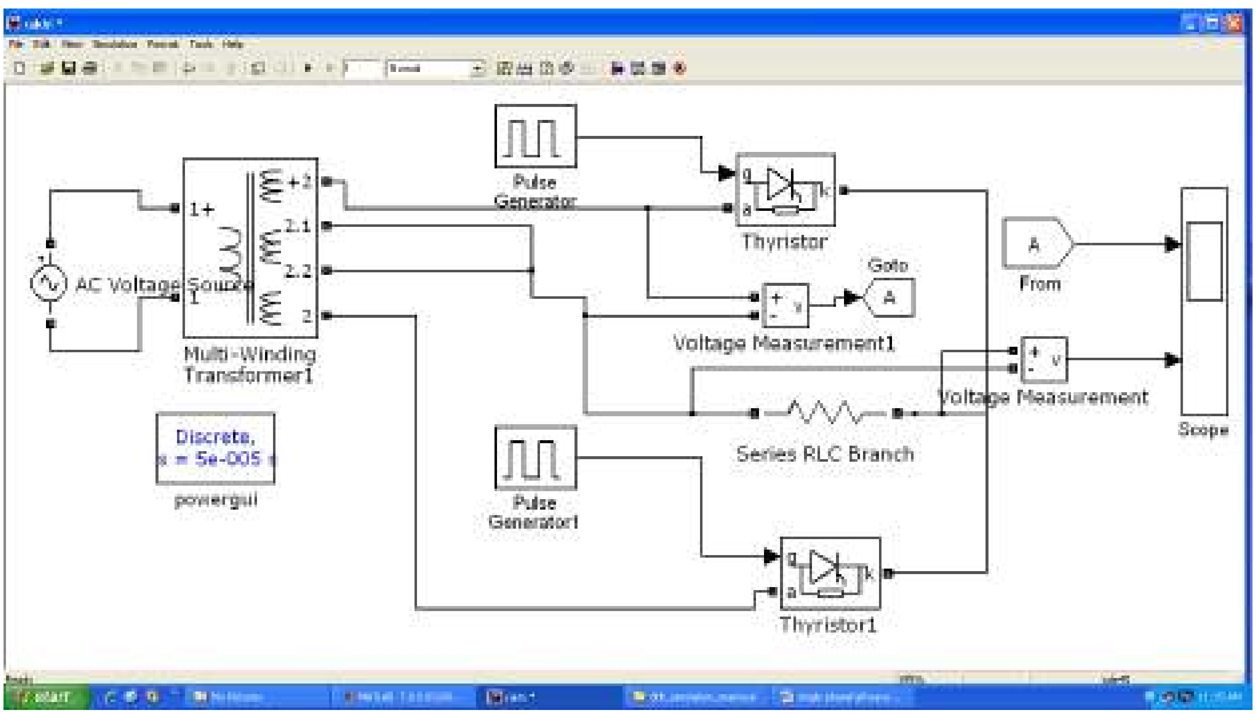


Fig. 2 Simulation Model for Controlled Full wave Rectifier with RLC Load

RESULTS:

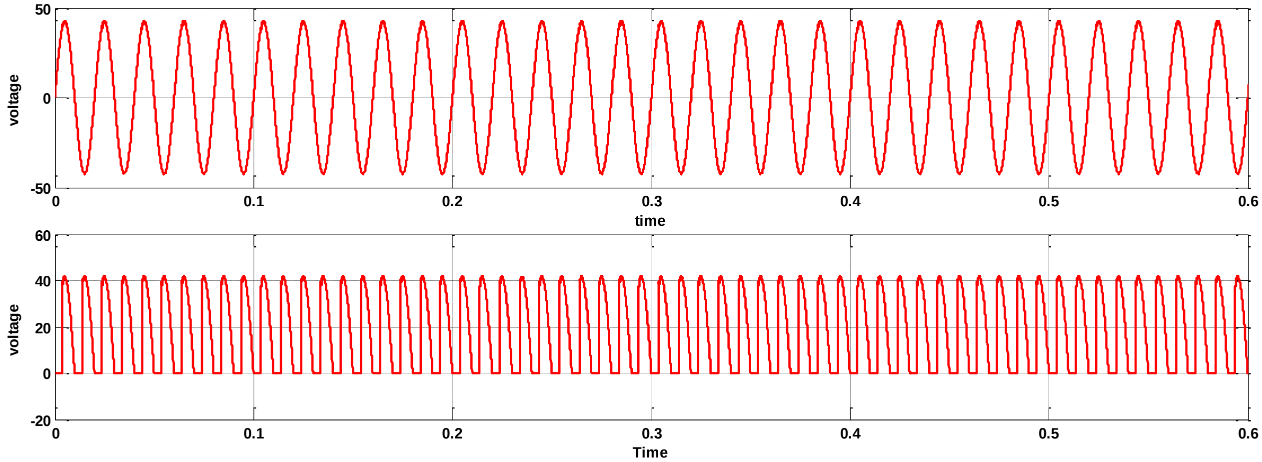


Fig. 3 Waveforms for Controlled Full wave Rectifier with RL Load

CONCLUSION:

The following conclusion may be derived when using SIMULINK in teaching power electronics courses: As power electronic systems are getting more complex today, the simulation used for education is requiring more features. Some directions in the development of simulation are discussed in this paper, with the help of present model students can simulate the power electronics circuit with various load & conditions.

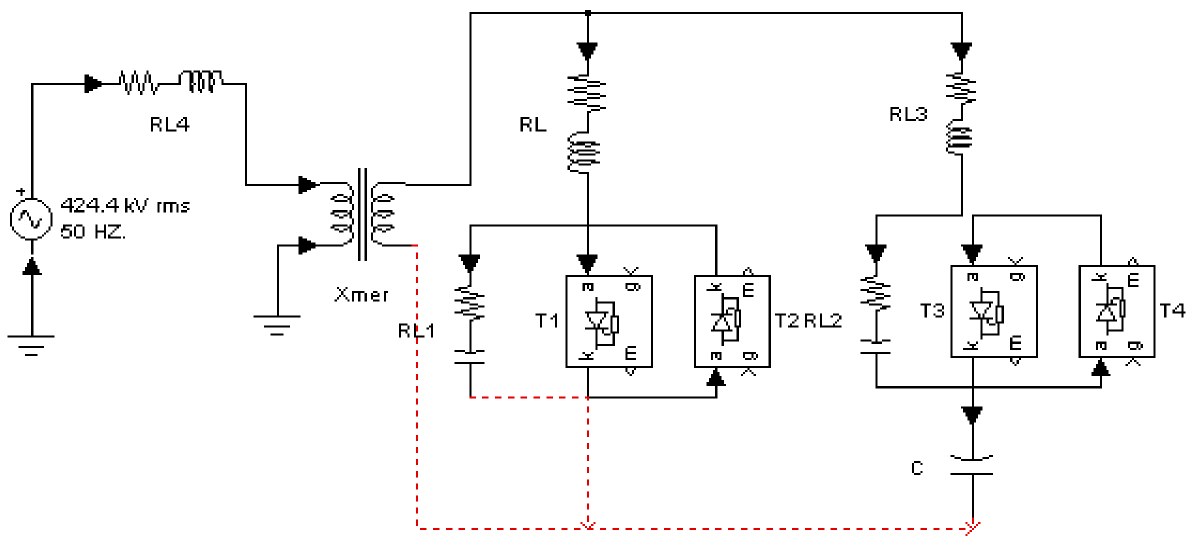
EXPERIMENT NO. 7

Aim: To simulate a Single Phase Thyristor Controlled Reactor.

Theory:

Electrical Power systems are combinations of electrical circuits, and electromechanical devices, like motors and generators. The blockset uses a Simulink environment, allowing a model to be built using click & drag procedure. The power system blockset has been designed to simulate power electronic devices.

Consider a circuit, representing one phase of static var compensator (SVC) used on a 735 KV transmission network. On the secondary of the 735 KV/ 16 KV transformer, two variable suceptance branches are connected in parallel: one thyristor controlled reactor (TCR) branch and one thyristor switched capacitor (TSC) branch.



TCR Branch TSC Branch

FIG.1 Single Line Model of Static Compensator

The TCR and TSC branches are both controlled by a valve consisting of two thyristor strings connected in antiparallel. An RC snubber circuit is connected across each valve. The TSC branch is switched on/off, thus providing discrete step variation of SVC capacitive current. The TCR branch is phase controlled in order to obtain a continuous variation of the net SVC reactive current.

MODEL OF THYRISTOR CONTROLLED REACTOR (TCR)

The ability of an FC-TCR (fixed-capacitor thyristor controlled reactor) compensator to change its reactive power within the theoretical time of half a period requires, as a prerequisite, the setting up of a proper control function. In other words, the firing angle α of the thyristors in antiparallel has to be related to properly detectable input variables, i.e. load reactive power. This comes in the category of shunt compensator.

The theoretical and computer-simulation approaches to this problem are examined and compared. The equations describing the relation between α and the load reactive power are introduced. The whole system is then modeled and simulated by computer, and the results are compared with the theoretical ones.

The control curves obtained theoretically agree well with those obtained by simulation. It is concluded that the proposed analytical approximating equation offers quite good results for practical purposes. The fixed control thyristor controlled reactor is a var generator arrangement using a fixed capacitance with a thyristor controlled reactor. The model of FC-TCR with the line voltage of 16 kV is shown.

In this TCR modeling, the firing angle is varied and for different angles of firing angles, the value of Active & Reactive power are observed along with the voltage and current waveforms of source and thyristor and are analyzed. This can be done by applying a formula for both the thyristors i.e. for changing the firing angle of thyristors T1 & T2.

Thyristor 1, T1= 1/50 + T

Thyristor 2, T2= 1/50 + 1/100 + T

Where T= Time period (for eg. For firing angle of 150°, T=1/150)

3.8 MODEL OF TCR

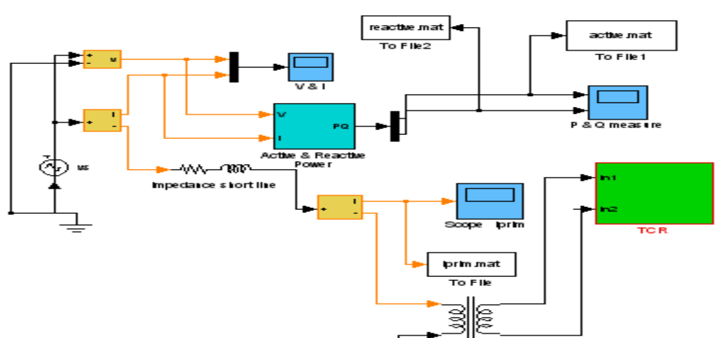


FIG.2 MODEL OF TCR (MAIN SYSTEM)

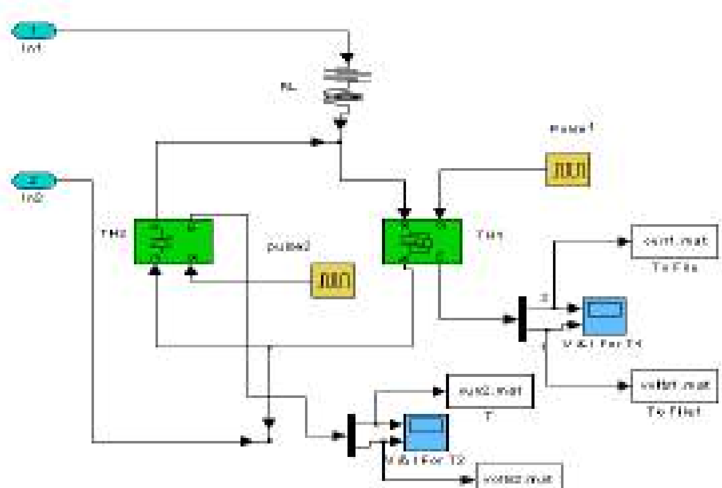
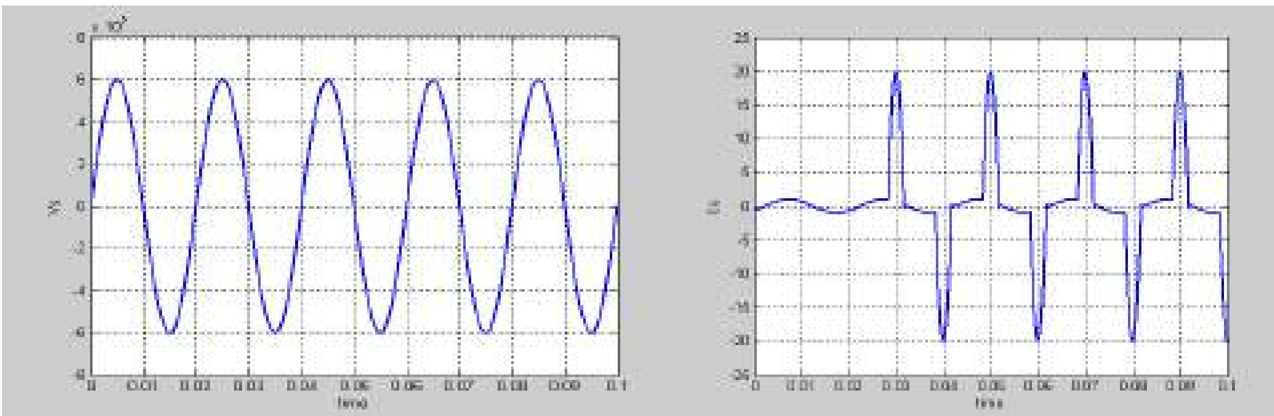


FIG.3 MODEL OF TCR (SUB SYSTEM)

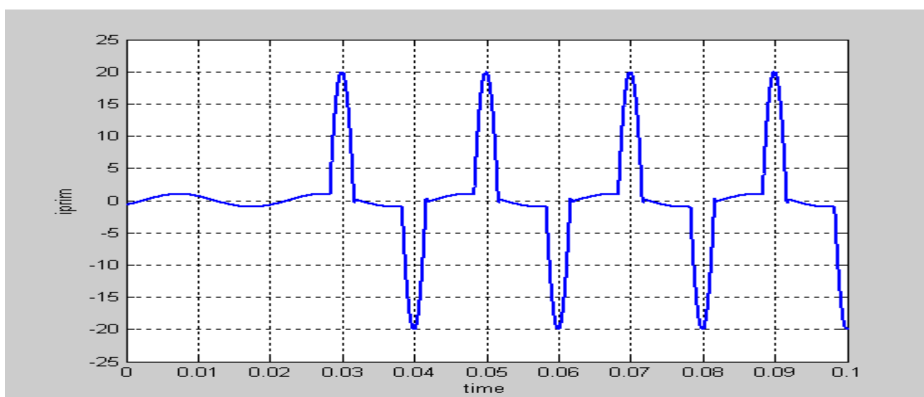
RESULTS:

Firing angle =120°

I) Source voltage and current



II) Primary current



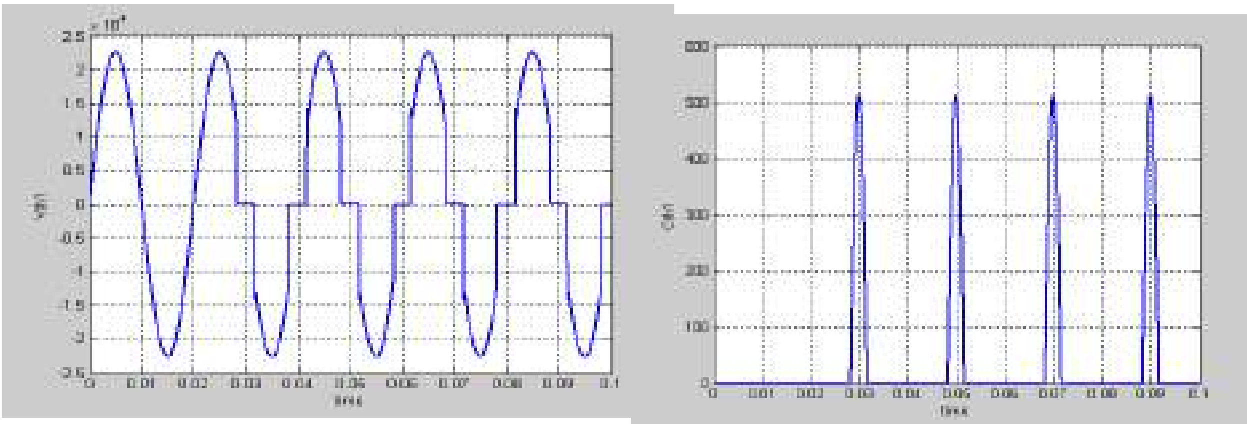
load iprim.mat

a=iprim

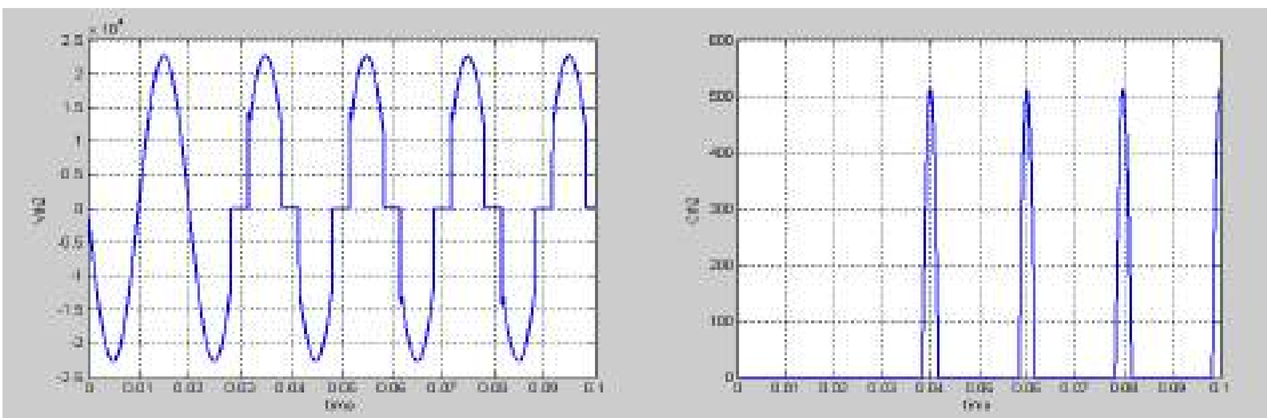
t=a(1,:)

plot (t,a)

III) Voltage & Current for Thyristor 1



IV) Voltage & Current for Thyristor 2



Result:

An overview of the technological development of VAR generators and compensators has been presented. Starting from the principles of VAR compensation, classical solutions using phase controlled semiconductors have been reviewed.

Heavy industrial loads having random, erratic and unbalanced changes in their real and reactive power requirement may cause considerable disturbances at the point of common coupling in the supply system. Static VAr compensators have been employed in many industrial applications throughout the world to compensate for the disturbance described and for the low lagging power factor associated with heavy inductive loads. The principal requirements of such compensators are the accurate computation of reactive power for each phase of the load and the rapid introduction of the computed swing in reactive power. Various alternative compensator designs have been used including schemes which, although effective, also introduce undesirable harmonic distortions.

EXPERIMENT NO. 8

Aim: To simulate a Single Phase Thyristor Switched Capacitor.

Theory: Electrical Power systems are combinations of electrical circuits, and electromechanical devices, like motors and generators. The block set uses a Simulink environment, allowing a model to be built using click & drag procedure. The power system block set has been designed to simulate power electronic devices.

Consider a circuit, representing one phase of static var compensator (SVC) used on a 735 KV transmission network. On the secondary of the 735 KV/ 16 KV transformers, two variable suceptance branches are connected in parallel: one thyristor controlled reactor (TCR) branch and one thyristor switched capacitor (TSC) branch.

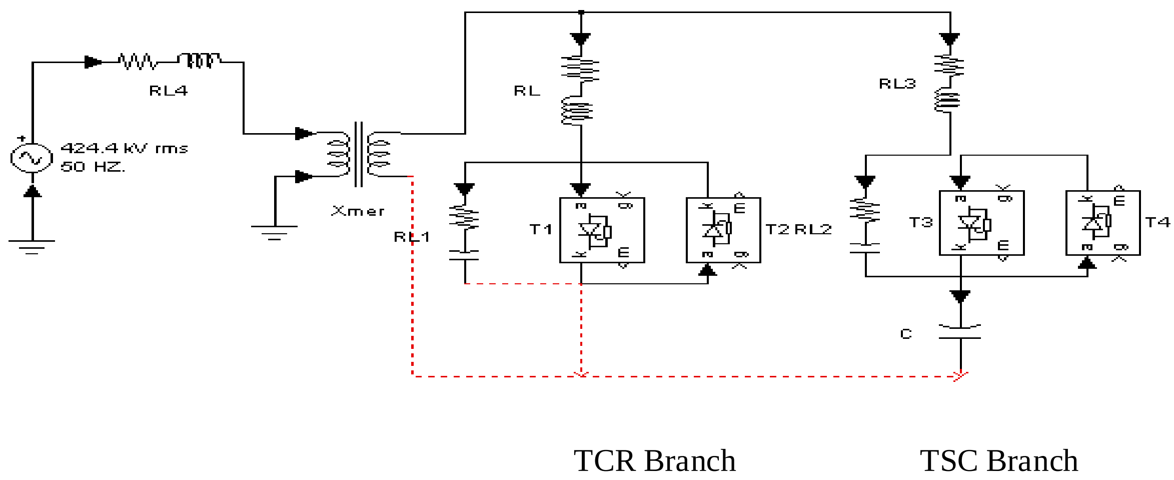


FIG.1 SINGLE LINE MODEL OF STATIC COMPENSATOR

The TCR and TSC branches are both controlled by a valve consisting of two thyristor strings connected in antiparallel. An RC snubber circuit is connected across each valve. The TSC branch is switched on/off, thus providing discrete step variation of SVC capacitive current. The TCR branch is phase controlled in order to obtain a continuous variation of the net SVC reactive current.

MODEL OF THYRISTOR SWITCHED CAPACITOR (TCR)

In a static var generator for providing reactive power compensation to a ac network and including a thyristor switched capacitor bank having a series combination of a capacitor, a bidirectional switch having gate drive, and a current limiting inductor with an applied voltage appearing across the combination with a current being conducted there through, a damping circuit and method for switching the thyristor switch to achieve damping of oscillatory transients generated by the switching of the capacitor in the network, comprising of :

- Determining the magnitude and polarity of the voltage difference between the applied voltage and the voltage across the capacitor;
- Determining the occurrence of the prepeak quadrant and the postpeak quadrant of the applied voltage.

Like, TCR this is also a shunt compensator. In this TCR modeling, the firing angle is varied and for different angles of firing angles, the value of Active & Reactive power are observed along with the voltage and current waveforms of source and thyristor and are analyzed. This can be done by applying a formula for both the thyristors i.e. for changing the firing angle of thyristors T1 & T2. Here, both the thyristors are triggered at the same time. A step wave is given as the input pulse for both the thyristors T1 and T2. The step time is defined in the step wave input. The model of TSC with the line voltage of 16 kV is shown.

$$T=1/50/4 \text{ or } T=1/50/8$$

A power GUI Discrete system is also placed in the system such that the value of initial state setting is changed. It is different for both initial and final value of capacitor voltage.

3.10 MODEL OF TCR

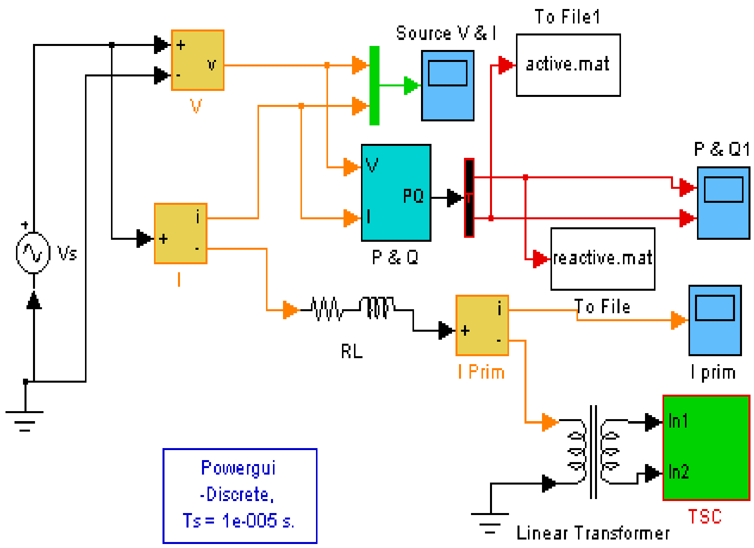


FIG.3.6 MODEL OF TSC (MAIN SYSTEM)

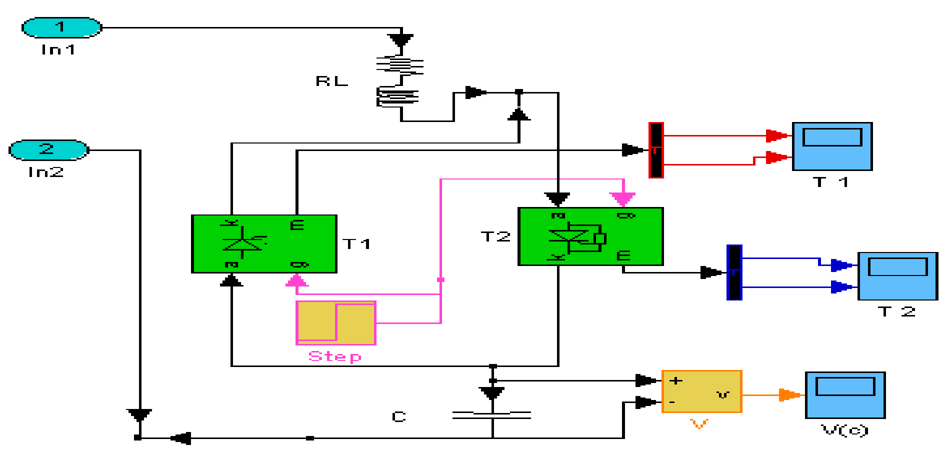


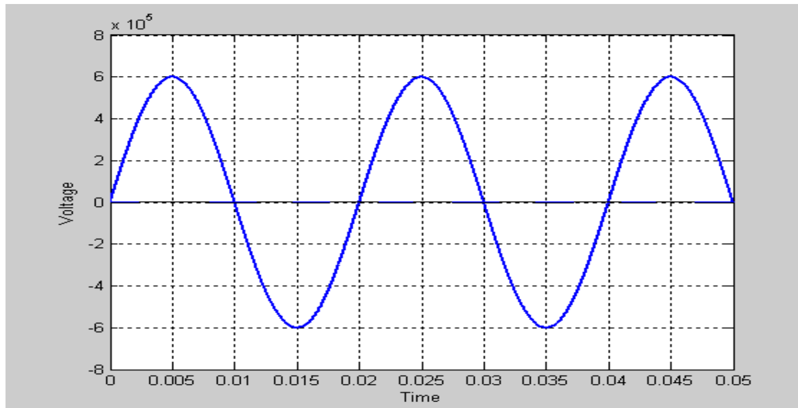
FIG.3.7 MODEL OF TSC (SUB SYSTEM)

RESULTS

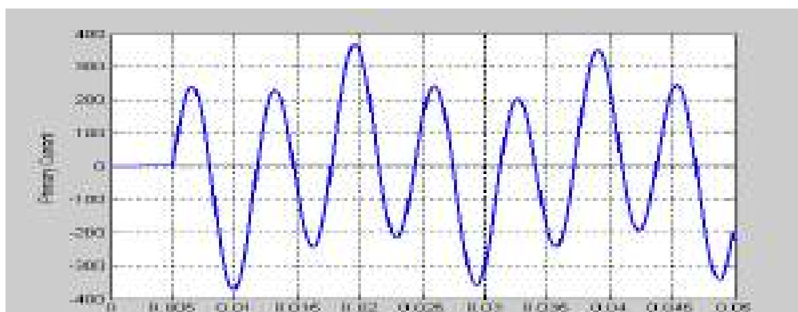
Step Pulse = 1/50/4

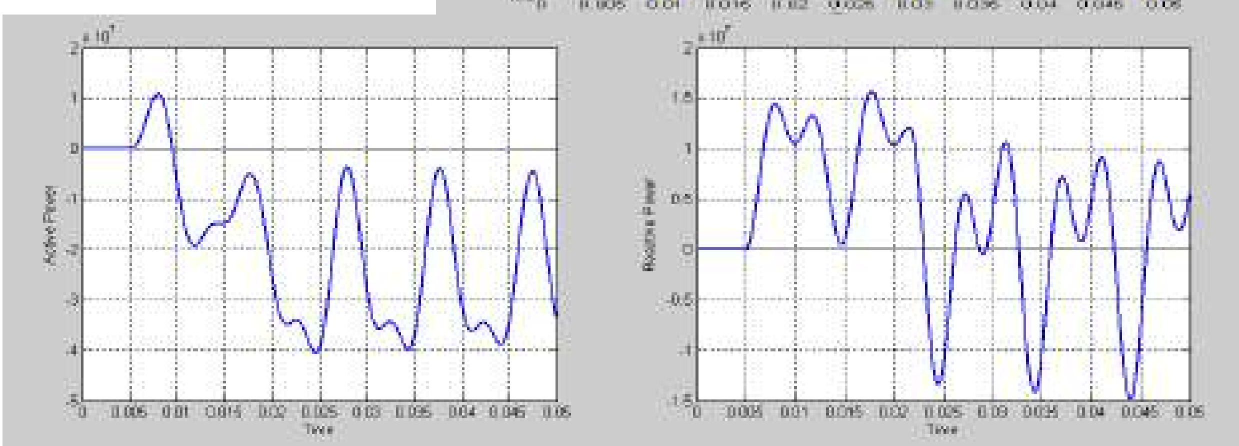
A.) Initially Charged = 24989 v

1.) Source voltage and current

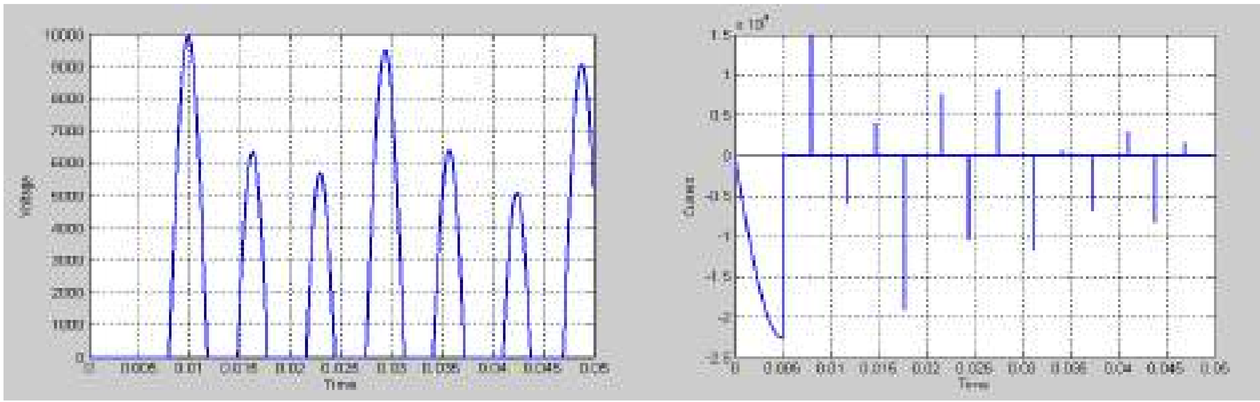


2.) Active and reactive power

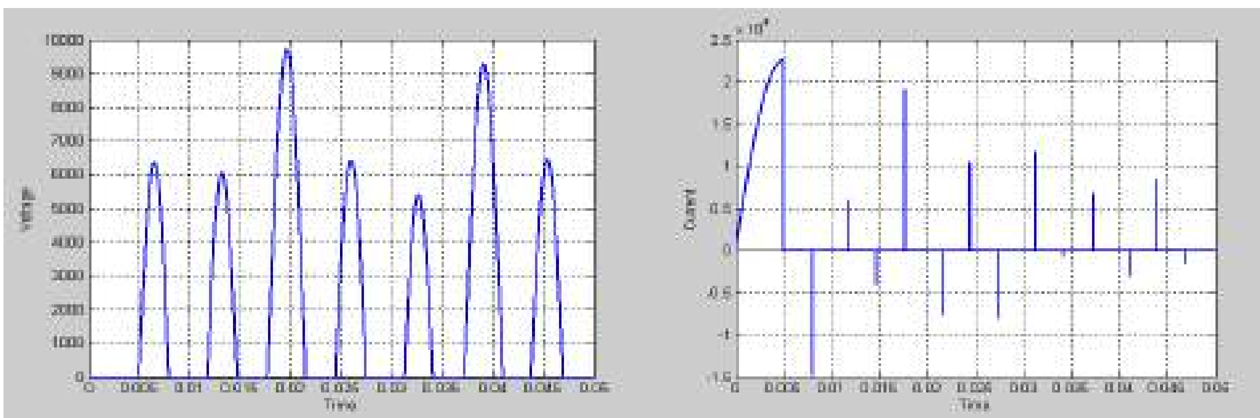




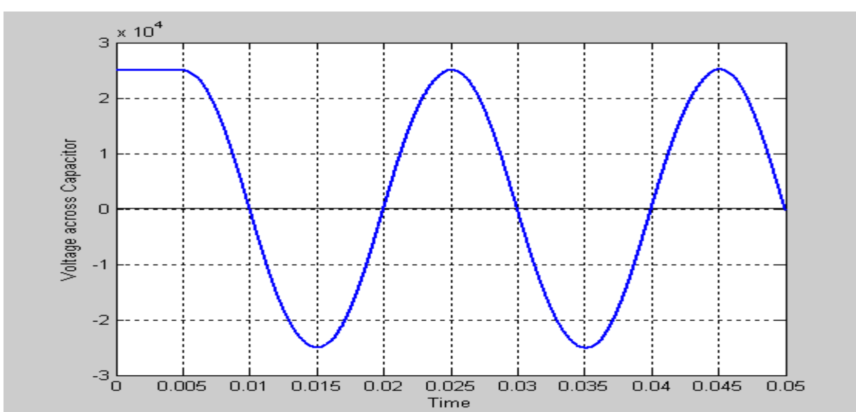
3.) Voltage & Current across Thyristor-1



4.) Voltage & Current across Thyristor-2



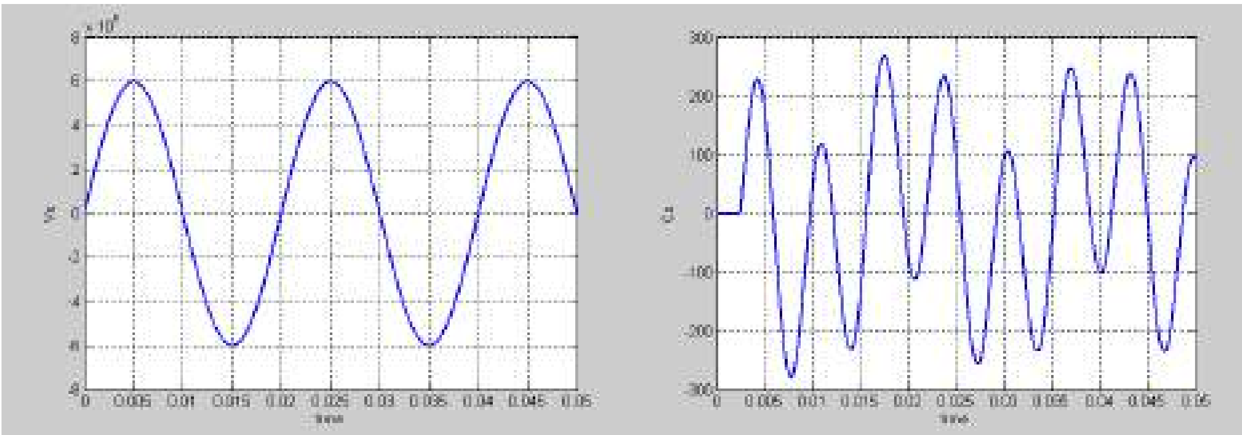
5.) Voltage across Capacitor



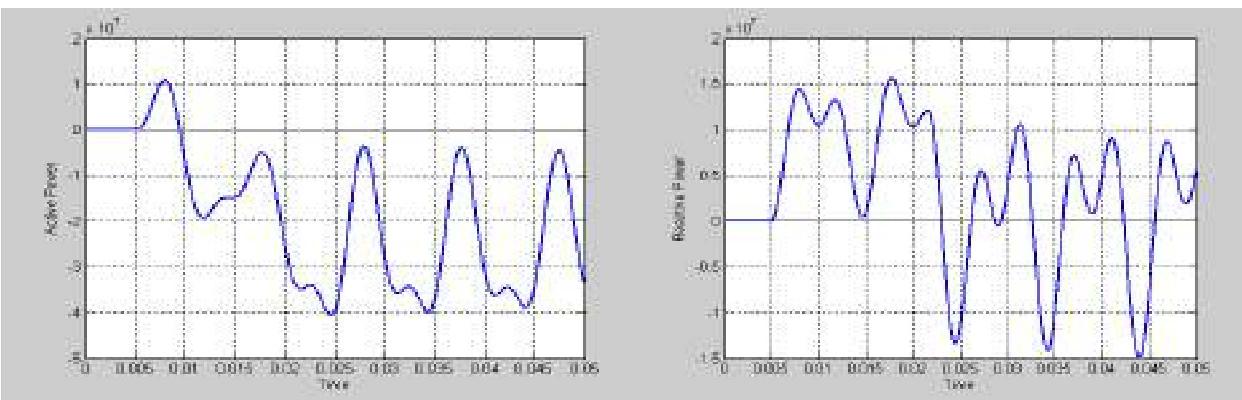
Step Pulse = 1/50/4

II.) Pre Charged = -0.314 v

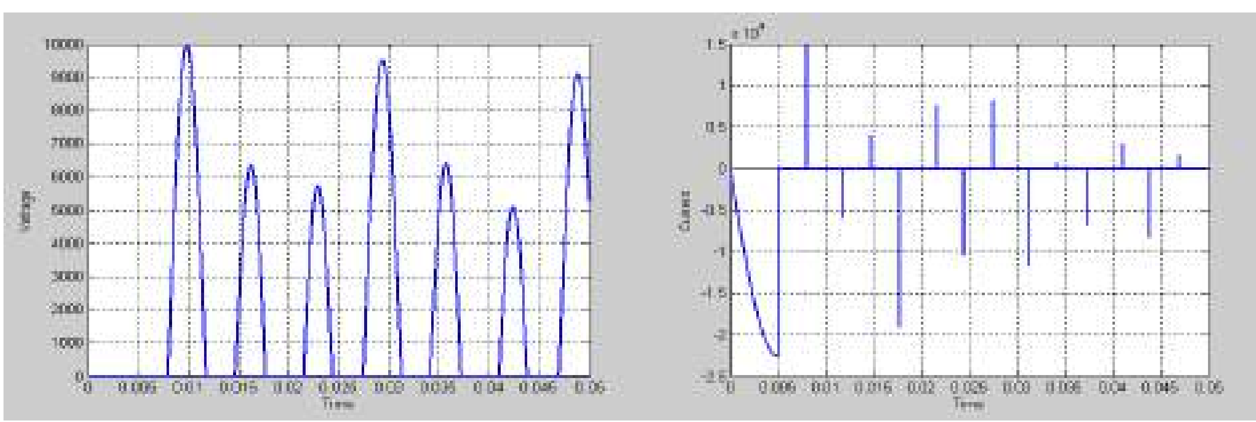
1.) Source voltage and current



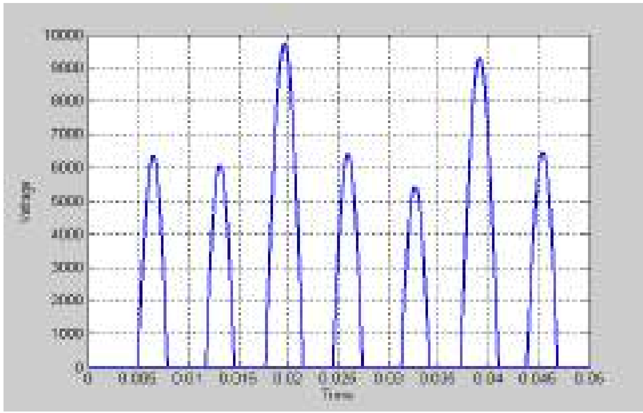
2.) Active and reactive power

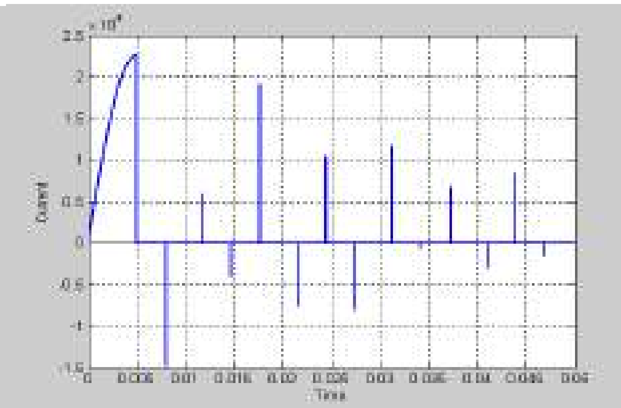


3.) Voltage & Current across Thyristor-1

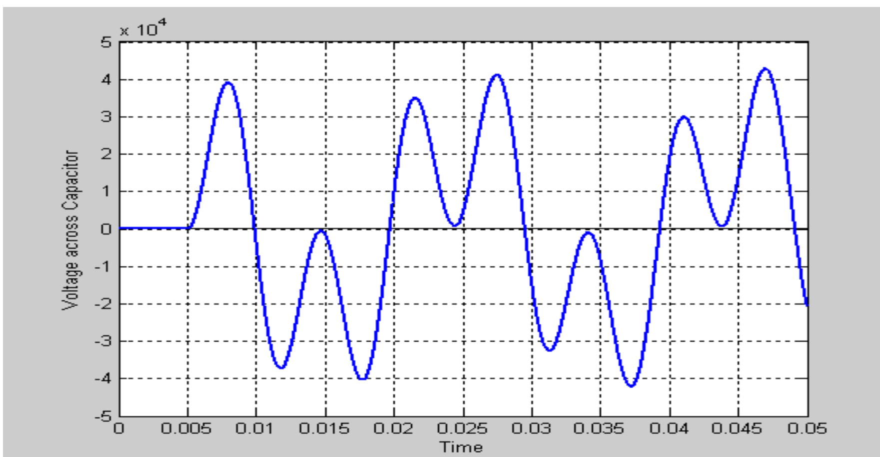


4.) Voltage & Current across Thyristor-2





5.) Voltage across Capacitor



Conclusion:

An overview of the technological development of VAR generators and compensators has been presented. Starting from the principles of VAR compensation, classical solutions using phase controlled semiconductors have been reviewed.

Heavy industrial loads having random, erratic and unbalanced changes in their real and reactive power requirement may cause considerable disturbances at the point of common coupling in the supply system. Static VAr compensators have been employed in many industrial applications throughout the world to compensate for the disturbance described and for the low lagging power factor associated with heavy inductive loads. The principal requirements of such compensators are the accurate computation of reactive power for each phase of the load and the rapid introduction of the computed swing in reactive power. Various alternative compensator designs have been used including schemes which, although effective, also introduce undesirable harmonic distortions.

EXPERIMENT NO. 9

AIM: DEVELOPING SIMULATION MODELS FOR SINGLE PHASE INVERTER, FOR DIFFERENT LOAD MODELS.

Theory:

INTRODUCTION:

SINGLE-PHASE PWM INVERTER

This demonstration illustrates use of the IGBT/Diode block in voltage-sourced converters. It also demonstrates harmonic analysis of PWM waveforms using the Powergui/FFT tool.

Contents

- **Circuit Description**
- **Demonstration**

Circuit Description

The system consists of two independent circuits illustrating single-phase PWM voltage-sourced converters (VSC).

1. Half-bridge converter

2. Full-bridge converter

The converters are built with the IGBT/Diode block which is the basic building block of all VSCs. The IGBT/Diode block is a simplified model of an IGBT (or GTO or MOSFET)/Diode pair where the forward voltages of the forced-commutated device and diode are ignored. You may replace these blocks by individual IGBT and diode blocks for a more detailed representation. VSCs are controlled in open loop with the Discrete PWM Generator block available in the Extras/Discrete Control Blocks library. The two circuits use the same DC voltage (Vdc = 400V), carrier frequency (1080 Hz) and modulation index (m = 0.8).

In order to allow further signal processing, signals displayed on the two Scope blocks (sampled at simulation sampling rate of 3240 samples/cycle) are stored in two variables named 'sps1phPWM1_str' and 'sps1phPWM2_str' (structures with time).

Demonstration

Run the simulation and observe the following two waveforms on the two Scope blocks: Current into the load (trace 1), Voltage generated by the PWM inverter (trace 2).

Once the simulation is completed, open the Powergui and select "FFT Analysis" to display the 0 - 5000 Hz frequency spectrum of signals saved in the three "sps1phPWMx_str" structures. The FFT will be performed on a 2-cycle window starting at t = 0.1 - 2/60 (last 2 cycles of recording). For each circuit, select Input labeled "V inverter" . Click on "Display" and observe the frequency spectrum of last 2 cycles.

The fundamental component of V inverter is displayed above the spectrum window. Compare the magnitude of the fundamental component of the inverter voltage with the theoretical values given in the circuit. Compare also the harmonic contents in the inverter voltage.

The half-bridge inverter generates a bipolar voltage (-200V or +200V) . Harmonics occur around the carrier frequency (1080 Hz +- k*60 Hz), with a maximum of 103% at 1080 Hz.

The full-bridge inverter generates a monopolar voltage varying between 0 and+400V for one half cycle and then between 0 and -400V for the next half cycle. For the same DC voltage and modulation index, the fundamental component magnitude is twice the value obtained with the half-bridge. Harmonics generated by the full-bridge are lower and they appear at double of the carrier frequency (maximum of 40% at 2*1080+-60 Hz) As a result, the current obtained with the full-bridge is smoother.

If you now perform a FFT on the signal "I load" you will notice that the THD of load current is 7.3% for the half-bridge inverter as compared to only 2% for the full-bridge inverter.

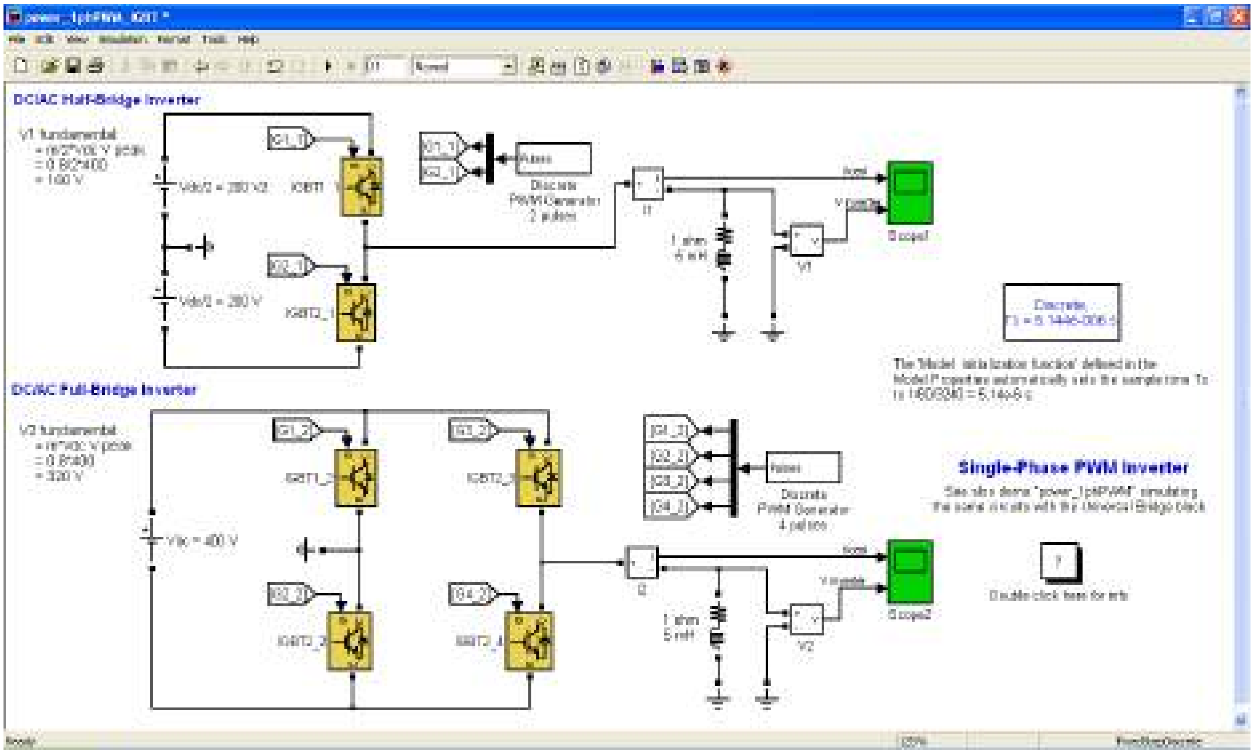


Fig. 1 Simulation Models for single phase Inverter

RESULTS:

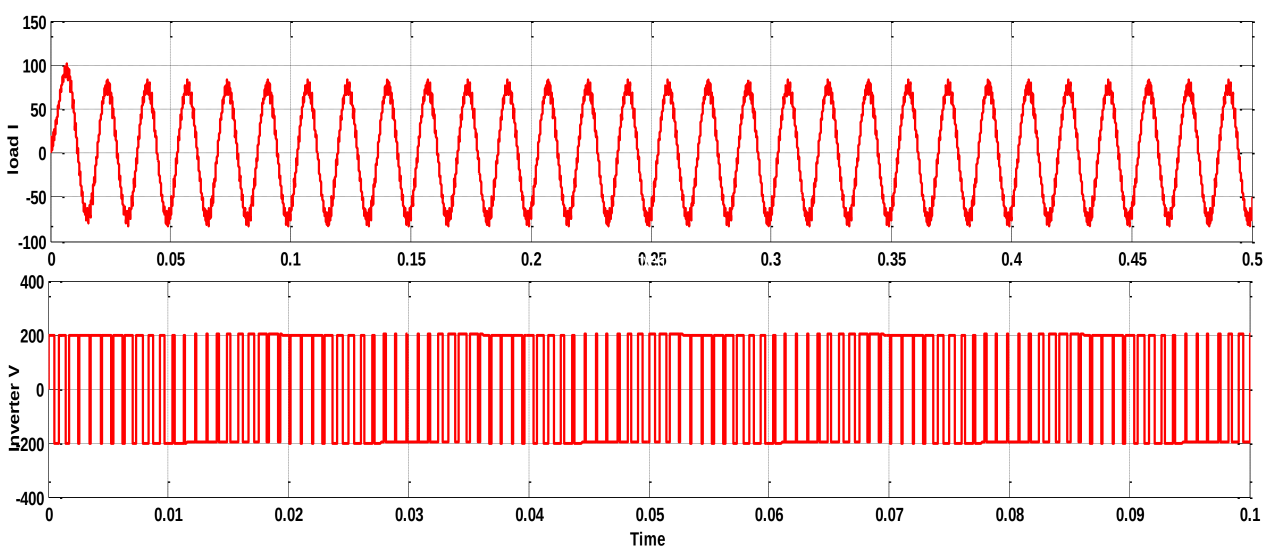


Fig. 2 DC/AC Half-Bridge Inverter

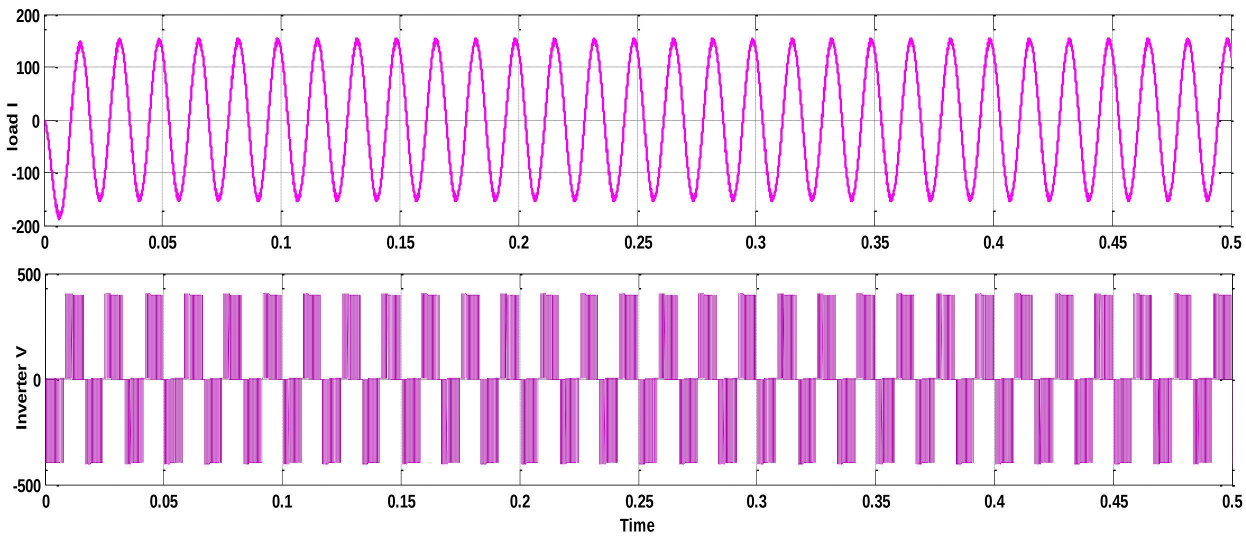


Fig. 2 DC/AC Full-Bridge Inverter

EXPERIMENT NO. 10

AIM: DEVELOPING SIMULATION MODELS USING FACTS DEVICES.

Theory:

INTRODUCTION:

STATCOM (Phasor Model)

Static Synchronous Compensator (STATCOM) Used for Midpoint Voltage Regulation on a 500-kV Transmission Line.

Pierre Giroux and Gibert Sybille (Hydro-Québec)

Contents

- **Circuit Description**
- **Demonstration**

Model Description

The Static Synchronous Compensator (STATCOM) is one of the key FACTS devices. Based on a voltage-sourced converter, the STATCOM regulates system voltage by absorbing or generating reactive power. Contrary to a thyristor-based Static Var Compensator (SVC), STATCOM output current (inductive or capacitive) can be controlled independent of the AC system voltage.

The power grid consists of two 500-kV equivalents (respectively 3000 MVA and 2500 MVA) connected by a 600-km transmission line. When the STATCOM is not in operation, the "natural" power flow on the transmission line is 930 MW from bus B1 to B3. In our demo, the STATCOM is located at the midpoint of the line (bus B2) and has a rating of +/- 100MVA. This STATCOM is a phasor model of a typical three-level PWM STATCOM. If you open the STATCOM dialog box and select "Display Power data", you will see that our model represents a STATCOM having a DC link nominal voltage of 40 kV with an equivalent capacitance of 375 uF. On the AC side, its total equivalent impedance is 0.22 pu on 100 MVA. This impedance represents the transformer leakage reactance and the phase reactor of the IGBT bridge of an actual PWM STATCOM.

Demonstration

1. STATCOM Dynamic Response

We will now verify the dynamic response of our model. Open the STATCOM dialog box and select "Display Control parameters". Verify that the "Mode of operation" is set to "Voltage regulation" and that "External control of reference voltage Vref" is selected. Also, the "droop" parameter should be set to 0.03 and the "Vac Regulator Gains" to 5 (proportional gain Kp) and 1000 (integral gain Ki). Close the STATCOM dialog block and open the "Step Vref" block (the red timer block connected to the "Vref" input of the STATCOM). This block should be programmed to modify the reference voltage Vref as follows: Initially Vref is set to 1 pu; at t=0.2 s, Vref is decreased to 0.97 pu; then at t=0.4 s, Vref is increased to 1.03; and finally at 0.6 s, Vref is set back to 1 pu. Also, make sure that the fault breaker at bus B1will not operate during the simulation (the parameters "Switching of phase A, B and C" should not be selected).

Run the simulation and look at the "VQ_STATCOM" scope. The first graph displays the Vref signal (magenta trace) along with the measured positive-sequencevoltage Vm at the STATCOM bus (yellow trace). The second graph displays the reactive power Qm (yellow trace) absorbed (positive value) or generated (negative value) by the STATCOM. The signal Qref (magenta trace) is not relevant to our simulation because the STATCOM is in "Voltage regulation" and not in "Var Control".

Looking at the Qm signal we can determine that the closed-loop time constant of the system is about 20 ms. This time constant depends primarily on the power system strength at bus B2 and on the programmed Vac Regulator gains of the STATCOM. To see the impact of the regulator gains, multiply the two gains of the Vac Regulator Gains by two and rerun the simulation. You should observe a much faster response with a small overshoot.

Looking at the Vm and Vref signals, you can see that the STATCOM does not operate as a perfect voltage regulator (Vm does not follow exactly the reference voltage Vref). This is due to the regulator droop (regulating slope) of 0.03 pu. For a given maximum capacitive/inductive range, this droop is used to extend the linear operating range of the STATCOM and also to ensure automatic load sharing with other voltage compensators (if any). Set the droop parameter to 0 and the voltage regulator gains back to 5 (Kp) and 1000 (Ki). If you then run a simulation, you will see that the measured voltage Vm now follows perfectly the reference voltage Vref.

2. STATCOM compared to a SVC under fault condition

We will now compare our STATCOM model with a SVC model having the same rating (+/- 100 MVA). If you double-click on the "SVC Power System" (the magenta block), you will see a SVC connected to a power grid similar to the power grid on which our STATCOM is connected. A remote fault will be simulated on both systems using a fault breaker in series with a fault impedance. The value of the fault impedance has been programmed to produce a 30% voltage sag at bus B2.Before running the simulation, you will first disable the "Step Vref" block by multiplying the time vector by 100. You will then program the fault breaker by selecting the parameters "Switching of phase A, B and C" and verify that the breaker is programmed (look at the "Transition times" parameter) to operate at t=0.2 s for a duration of 10 cycles. Check also that the fault breaker inside the "SVC Power System" has the same parameters. Finally, set the STATCOM droop back to its original value (0.03 pu).

Run the simulation and look at the "SVC vs STATCOM" scope. The first graph displays the measured voltage Vm on both systems (magenta trace for the SVC). The second graph displays the measured reactive power Qm generated by the SVC (magenta trace) and the STATCOM (yellow trace). During the 10-cycle fault, a key difference between the SVC and the STATCOM can be observed. The reactive power generated by the SVC is -0.48 pu and the reactive power generated by the STATCOM is -0.71 pu. We can then see that the maximum capacitive power generated by a SVC is proportional to the square of the system voltage (constantsusceptance) while the maximum capacitive power generated by a STATCOM decreases linearly with voltage decrease (constant current). This ability to provide more capacitive power during a fault is one important advantage of the STATCOM over the SVC. In addition, the STATCOM will normally exhibits a faster response than the SVC because with the voltage-sourced converter, the STATCOM has no delay associated with the thyristor firing (in the order of 4 ms for a SVC).

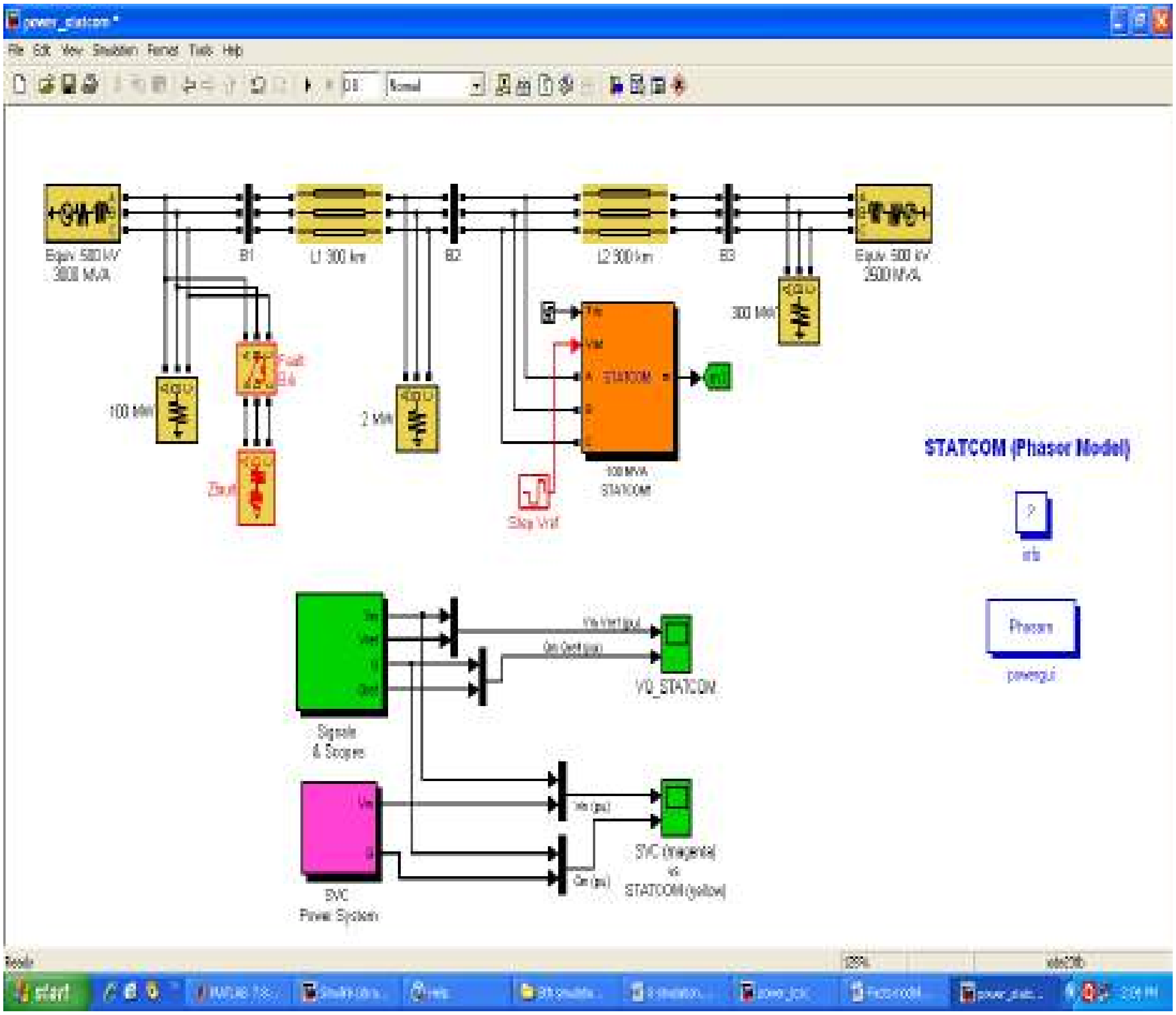
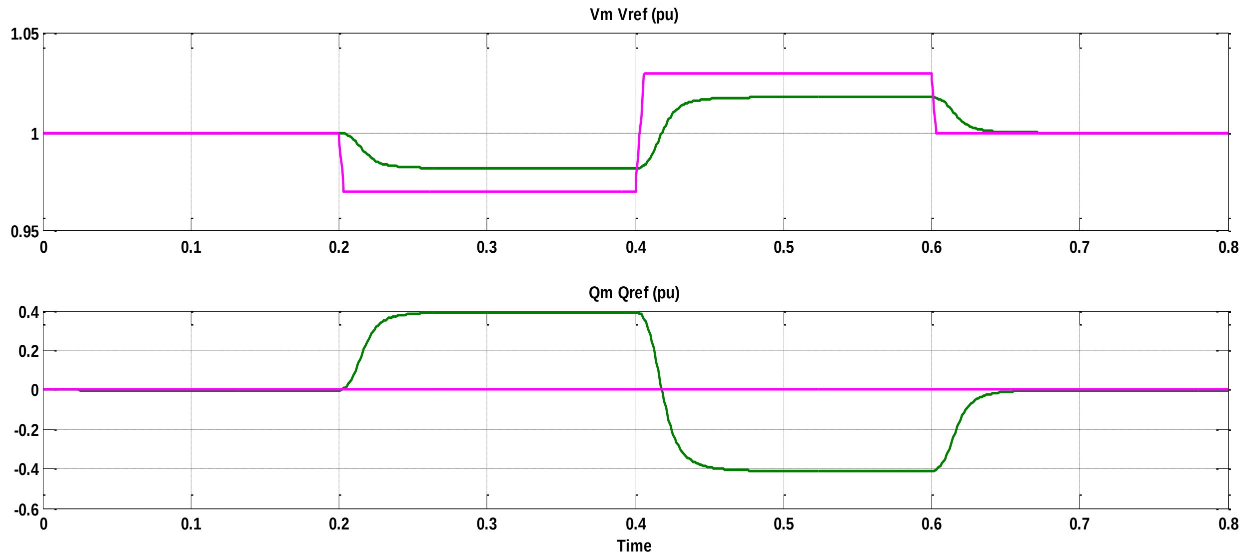
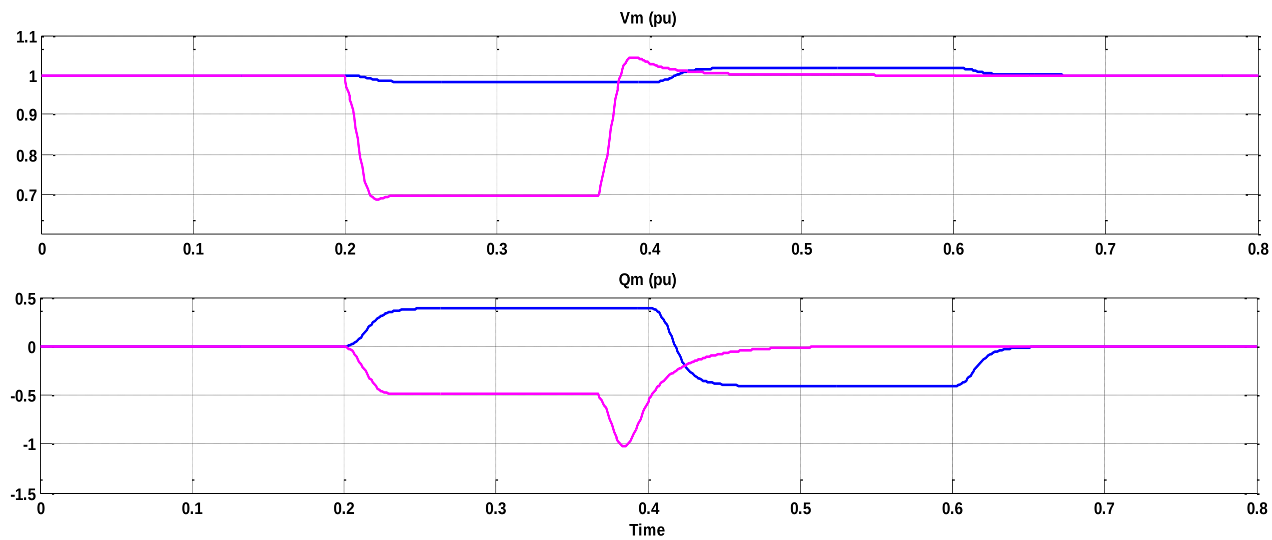


Fig.1 STATCOM (Phasor Model)

RESULTS:


Vm Vref (pu)
1.05
1
0.95
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
Qm Qref (pu)
0.4
0.2
0
-0.2
-0.4
-0.6
Time


Vm (pu)
1.1
1
0.9
0.8
0.7
0
0.1
0.2
0.3
0.4
0.5
0.6
0.7
0.8
Qm (pu)
0.5
0
-0.5
-1
-1.5
Time