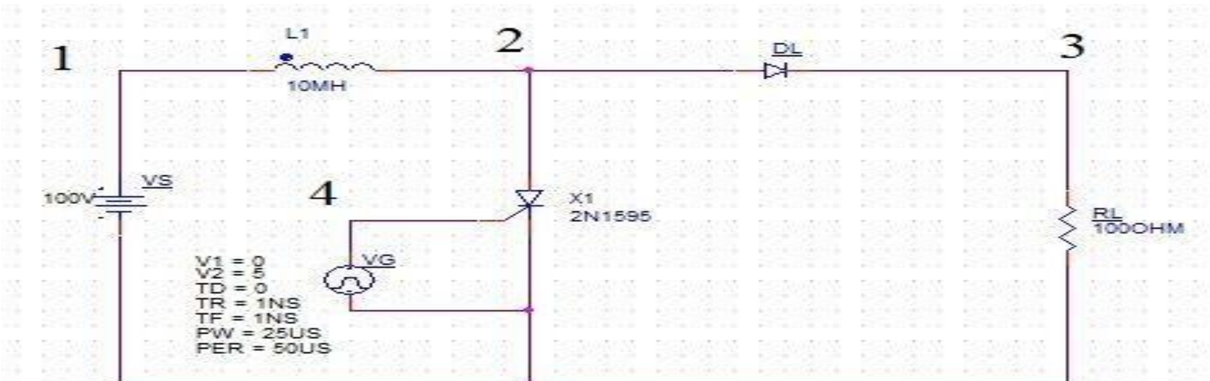
APPLICATIONS OF SOFT COMPUTING TOOLS IN ELECTRICAL ENGINEERING LAB MANUAL



Department of Electrical and Electronics Engineering

VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA

NEAR PAKALA, CHITTOOR-517112
(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

APPLICATIONS OF SOFT COMPUTING TOOLS IN ELECTRICAL ENGINEERING LAB MANUAL

Name: ______________________________

H.T.No: ______________________________

Year/Semester: ______________________________

Department of Electrical and Electronics Engineering

VEMU INSTITUTE OF TECHNOLOGY::P.KOTHAKOTA
NEAR PAKALA, CHITTOOR-517112
(Approved by AICTE, New Delhi & Affiliated to JNTUA, Anantapuramu)

VEMU INSTITUTE OF TECHNOLOGY

DEPT.OF ELECTRICAL AND ELECTRONICS ENGINEERING

VISION OF THE INSTITUTE

- To be a premier institute for professional education producing dynamic and vibrant force of technocrats with competent skills, innovative ideas and leadership qualities to serve the society with ethical and benevolent approach.

MISSION OF THE INSTITUTE

- To create a learning environment with state-of-the art infrastructure, well equipped laboratories, research facilities and qualified senior faculty to impart high quality technical education.
- To facilitate the learners to foster innovative ideas, inculcate competent research and consultancy skills through Industry-Institute Interaction.
- To develop hard work, honesty, leadership qualities and sense of direction in rural youth by providing value based education.

VISION OF THE DEPARTMENT

- To produce professionally deft and intellectually adept Electrical and Electronics Engineers and equip them with the latest technological skills, research & consultancy competencies along with social responsibility, ethics, Lifelong Learning and leadership qualities.

MISSION OF THE DEPARTMENT

- To produce competent Electrical and Electronics Engineers with strong core knowledge, design experience & exposure to research by providing quality teaching and learning environment.
- To train the students in emerging technologies through state - of - the art laboratories and thus bridge the gap between Industry and academia.
- To inculcate learners with interpersonal skills, team work, social values, leadership qualities and professional ethics for a holistic engineering professional practice through value based education.

PROGRAM EDUCATIONAL OBJECTIVES(PEOs)

Programme Educational Objectives (PEOs) of B.Tech (Electrical and Electronics Engineering) **program are:**

Within few years of graduation, the graduates will

PEO 1: Provide sound foundation in mathematics, science and engineering fundamentals to analyze, formulate and solve complex engineering problems.

PEO 2: Have multi-disciplinary Knowledge and innovative skills to design and develop Electrical & Electronics products and allied systems.

PEO 3: Acquire the latest technological skills and motivation to pursue higher studies leading to research.

PEO 4: Possess good communication skills, team spirit, ethics, modern tools usage and the life-long learning needed for a successful professional career.

PROGRAM OUTCOMES (POs)

PO-1	**Engineering knowledge:** Apply the knowledge of mathematics, science, engineering fundamentals, and an engineering specialization to the solution of complex engineering problems.
PO-2	**Problem analysis:** Identify, formulate, review research literature, and analyze complex engineering problems reaching substantiated conclusions using first principles of mathematics, natural sciences, and engineering sciences.

PO-3	**Design/development of solutions:** Design solutions for complex engineering problems and design system components or processes that meet the specified needs with appropriate consideration for the public health and safety, and the cultural, societal, and environmental considerations.
PO-4	**Conduct investigations of complex problems:** Use research-based knowledge and research methods including design of experiments, analysis and interpretation of data, and synthesis of the information to provide valid conclusions.
PO-5	**Modern tool usage:** Create, select, and apply appropriate techniques, resources, and modern engineering and IT tools including prediction and modeling to complex engineering activities with an understanding of the limitations.
PO-6	**The engineer and society:** Apply reasoning informed by the contextual knowledge to assess societal, health, safety, legal and cultural issues and the consequent responsibilities relevant to the professional engineering practice.
PO-7	**Environment and sustainability:** Understand the impact of the professional engineering solutions in societal and environmental contexts, and demonstrate the knowledge of, and need for sustainable development.
PO-8	**Ethics:** Apply ethical principles and commit to professional ethics and responsibilities and norms of the engineering practice.
PO-9	**Individual and team work:** Function effectively as an individual, and as a member or leader in diverse teams, and in multidisciplinary settings.
PO-10	**Communication:** Communicate effectively on complex engineering activities with the engineering community and with society at large, such as, being able to comprehend and write effective reports and design documentation, make effective presentations, and give and receive clear instructions.
PO-11	**Project management and finance:** Demonstrate knowledge and understanding of the engineering and management principles and apply these to one’s own work, as a member and leader in a team, to manage projects and in multidisciplinary environments.
PO-12	**Life-long learning:** Recognize the need for, and have the preparation and ability to engage in independent and life-long learning in the broadest context of technological change.

PROGRAM SPECIFIC OUTCOMES (PSOs)

On completion of the B.Tech. (Electrical and Electronics Engineering) degree, the graduates will be able to

PSO-1: Higher Education: Apply the fundamental knowledge of Mathematics, Science, Electrical and Electronics Engineering to pursue higher education in the areas of Electrical Circuits, Electrical Machines, Electrical Drives, Power Electronics, Control Systems and Power Systems.

PSO-2: Employment: Get employed in Public/Private sectors by applying the knowledge in the domains of design and operation of Electronic Systems, Microprocessor based control systems, Power systems, Energy auditing etc.

CONTENTS

(20A02606) APPLICATIONS OF SOFT COMPUTING TOOLS IN ELECTRICAL ENGINEERING LABORATORY

13	Single Phase A.C. Voltage Controller	
14	Simulation Of Three Phase Full Converter And Pwm Inverter	

JAWAHARLAL NEHRU TECHNOLOGICAL UNIVERSITY, ANANTAPUR
B. Tech III - II SEM (E.E.E)

L	C	P	C
1	0	2	2

(20A02606) APPLICATIONS OF SOFT COMPUTING TOOLS IN ELECTRICAL ENGINEERING LABORATORY

Course Objectives: The objectives of this course include:

Understand the basic concepts of Electrical Engineering

- Apply the concepts to design MATLAB models
- Analyse various Electrical engineering applications through MATLAB
- Develop real time models using MATLAB

Course Outcomes: At the end of the course the student will be able to:

Understand the basic concepts of Electrical Engineering.

- Apply the concepts to design MATLAB models
- Analyse various Electrical engineering applications through MATLAB.
- Develop real time models using MATLAB.

List of Experiments:

1. Transient analysis of given electrical network
2. Simulation of 1-phase and 3-phase transformers
3. Implementation of buck and boost dc-dc converters
4. Study on the design of PI controllers and stability analysis for a DC-DC buck Converter
5. Sine-PWM techniques for single-phase half-bridge, full-bridge and three-phase inverters
6. Economic Load Dispatch of (i) Thermal Units and (ii) Thermal Plants using Conventional method
7. Transient Stability Analysis of Power Systems using Equal Area Criterion (EAC)
8. Reactive Power Control in a transmission system (Ferranti effect, Effect of shunt Inductor)
9. Fault studies using Zbus matrix
10. Design of virtual PMU
11. Simulink Model For Two Area Load Frequency Control

12. MATLAB Program to Find Optimum Loading Of Generators with Penalty Factors

GENERAL INSTRUCTIONS FOR LABORATORY CLASSES

DO‘S

1. Without Prior permission do not enter into the Laboratory.
2. While entering into the LAB students should wear their ID cards.
3. The Students should come with proper uniform.
4. Students should sign in the LOGIN REGISTER before entering into the laboratory.
5. Students should come with observation and record note book to the laboratory.
6. Students should maintain silence inside the laboratory.
7. Circuit connections must be checked by the lab-in charge before switching the supply

DONT‘S

1. Students bringing the bags inside the laboratory.
2. Students wearing slippers/shoes insides the laboratory.
3. Students scribbling on the desk and mishandling the chairs.
4. Students using mobile phones inside the laboratory.
5. Students making noise inside the laboratory.
6. Students mishandle the devices.
7. Students write anything on the devices

SCHEME OF EVALUATION

S. NO	EXPERIMENT NAME	DATE	MARKS AWARDED				Total 30 (M)
			Record (10M)	Observation (10M)	Viva voce (5M)	Attendance (5M)	
1	Transient analysis of given electrical network						
2	Simulation of 1-phase and 3-phase transformers						
3	Implementation of buck and boost dc-dc converters						
4	Study on the design of PI controllers and stability analysis for a DC-DC buck Converter						
5	Sine-PWM techniques for single-phase half-bridge, full-bridge and three-phase inverters						
6	Economic Load Dispatch of (i) Thermal Units and (ii) Thermal Plants using Conventional method						
7	Transient Stability Analysis of Power Systems using Equal Area Criterion (EAC)						
8	Reactive Power Control in a transmission system (Ferranti effect, Effect of shunt Inductor)						
9	Fault studies using Zbus matrix						
10	Design of virtual PMU						
11	Simulink Model For Two Area Load Frequency Control						
12	MATLAB Program to Find Optimum Loading Of Generators with Penalty						

	Factors						
ADDITIONAL EXPERIMENTS							
13	Single Phase A.C. Voltage Controller						
14	Simulation Of Three Phase Full Converter And Pwm Inverter						

Signature of Lab In-charge

Exp. No.:1 **Date:**

TRANSIENT ANALYSIS

AIM:

To find out the transient response and parametric analysis by simulation of RLC circuits Using Pulse, and Step response.

SOFTWARE REQUIRED:

PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

a) Simulation of STEP RESPONSE Using PSPICE:

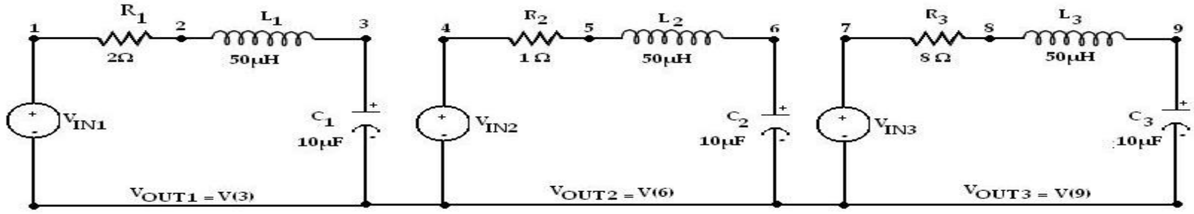
SYNTAX USED:

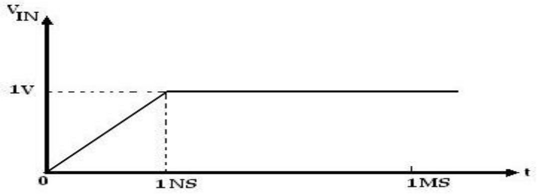
S.NO	TYPE OF SOURCE	REPRESENTATION OF SOURCE	DECLARATION FORMAT
1	STEP RESPONSE	PWL	STEP (Time at a Point) (Voltage at a Point)
2	TRANSIENT ANALYSIS	.TRAN	.TRAN TStep Tstop [TStart TMax] [UIC]
3	PROBE STATEMENT	.PROBE	It is a wave form analyzer
4	PLOT STATEMENT	.PLOT	.PLOT (Output Variables) {(Lower limit

DATA REQUIRED FOR DRAWING THE CIRCUIT DIAGRAM:

For example, Three RLC circuits with R=2Ω, 1 Ω, and 8 Ω respectively, with L having the values of 50µH each, with C having the values of 10µF each. The inputs are identical Step Response. The Step having the Time at points as 1nsec and 1msec respectively and Voltage at a point as 1V respectively. Use PSPICE to plot and calculate the transient response from 0 to 400µseconds with an increment of 1µsecond. Plot the voltages across the capacitors.

CIRCUIT DIAGRAM:





PROCEDURE:

1. Open PSpice A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the voltage, current graph of any in probe window.

PROGRAM

```
VIN1 1 0 PWL(0 0 1NS 1V 1MS 1V)
VIN2 4 0 PWL(0 0 1NS 1V 1MS 1V)
VIN3 7 0 PWL(0 0 1NS 1V 1MS 1V)
R1  1 2  2
R2   4 5  1
R3   7 8  8
L1    2 3  50UH
L2    5 6  50UH
L3    8 9 50UH
C1    3 0 10UF
C2    6 0 10UF
C3    9 0 10UF
.TRAN 1US 400US
```

```
.PLOT TRAN V(3) V(6) V(9)
.PROBE
.END
```

THEORETICAL CALCULATIONS

RESULT:

VIVA QUESITIONS:

1. Define transient response.

2. Define sinusoidal response.

3. Define time constant.

4. When Transient behavior occur in any circuits

Exp. No.: 2 **Date:**

SIMULATION OF 1-PHASE AND 3-PHASE TRANSFORMERS

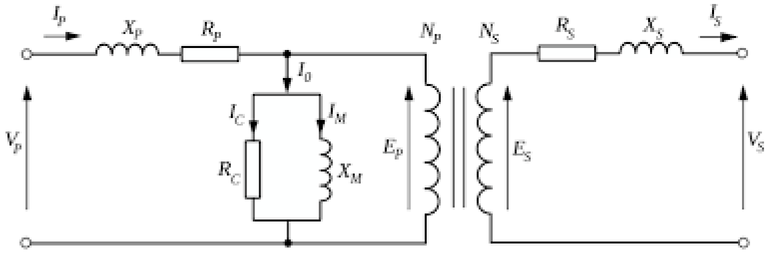
AIM:

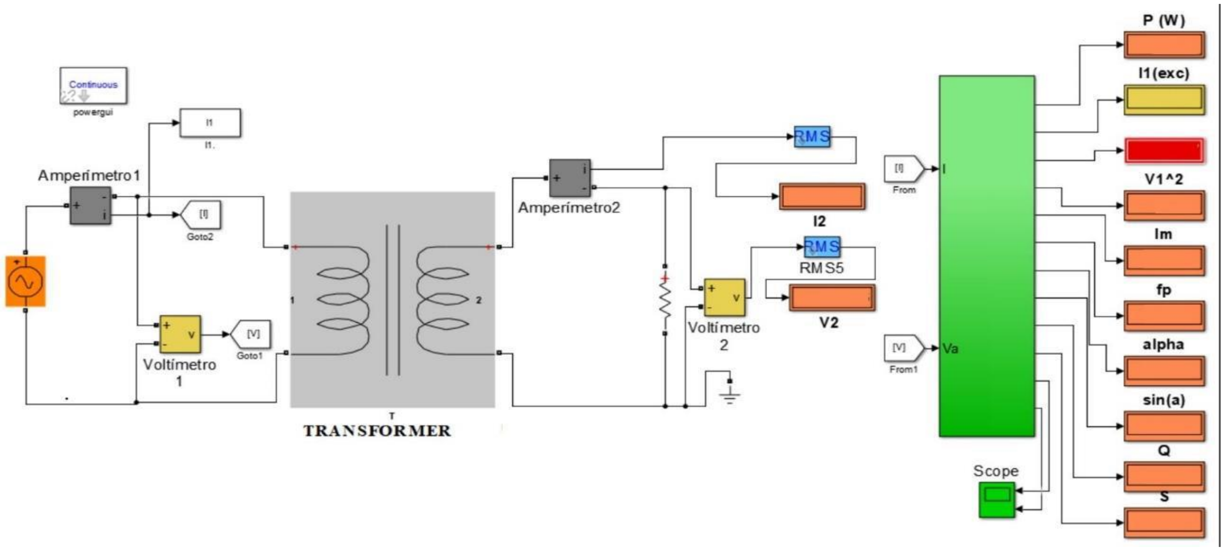
To determine currents for the given DC circuit by mesh analysis.

APPARATUS REQUIRED:

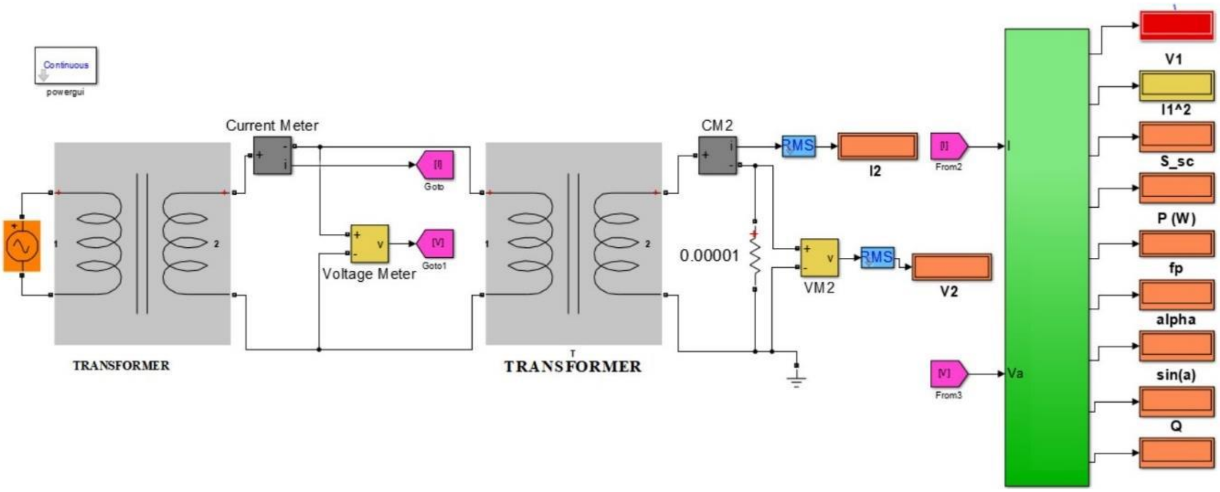
S.No	Name Of The Equipment	Quantity
1	Voltmeter	1NO
2	Ammeter	1NO
3	Wattmeter	1NO
4	Load	1NO

CIRCUIT DIAGRAM:





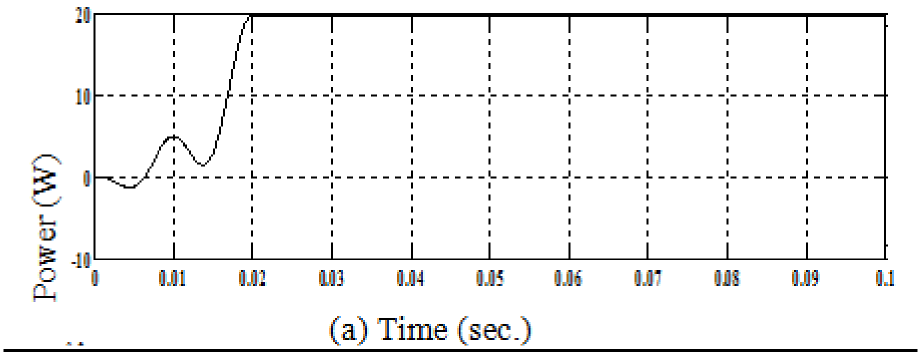
Modeling of open-circuit test on transformer

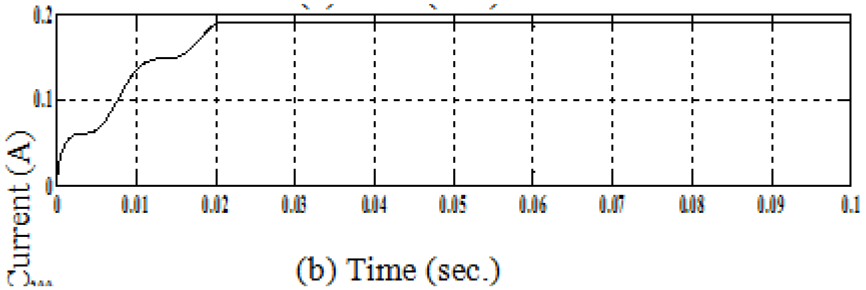


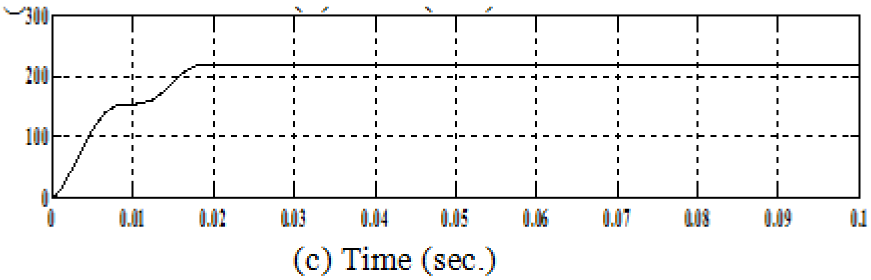
Modeling of short-circuit teston transformer

PROCEDURE:

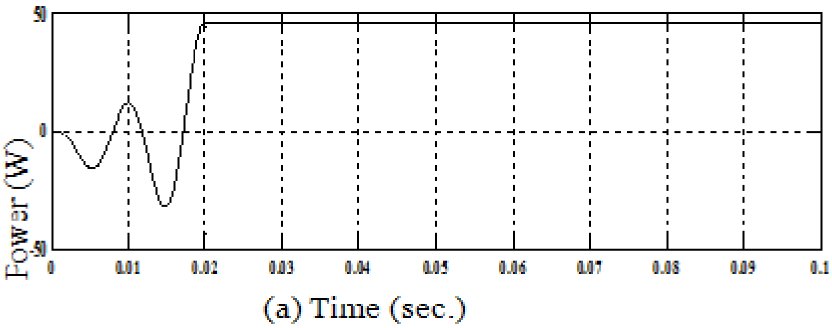
1. Open PSPICE A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the required outputs (Graphs) in output window.

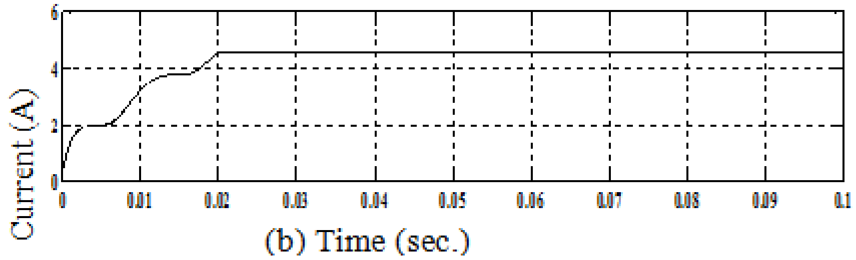


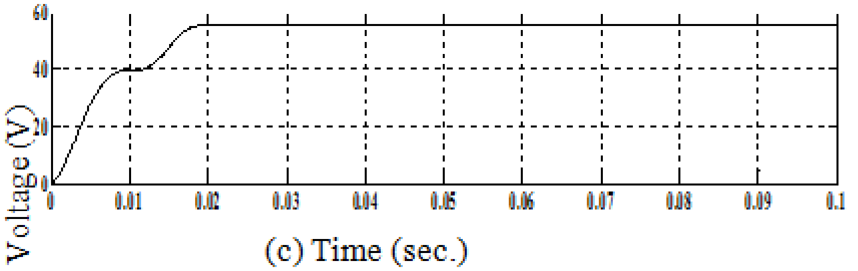




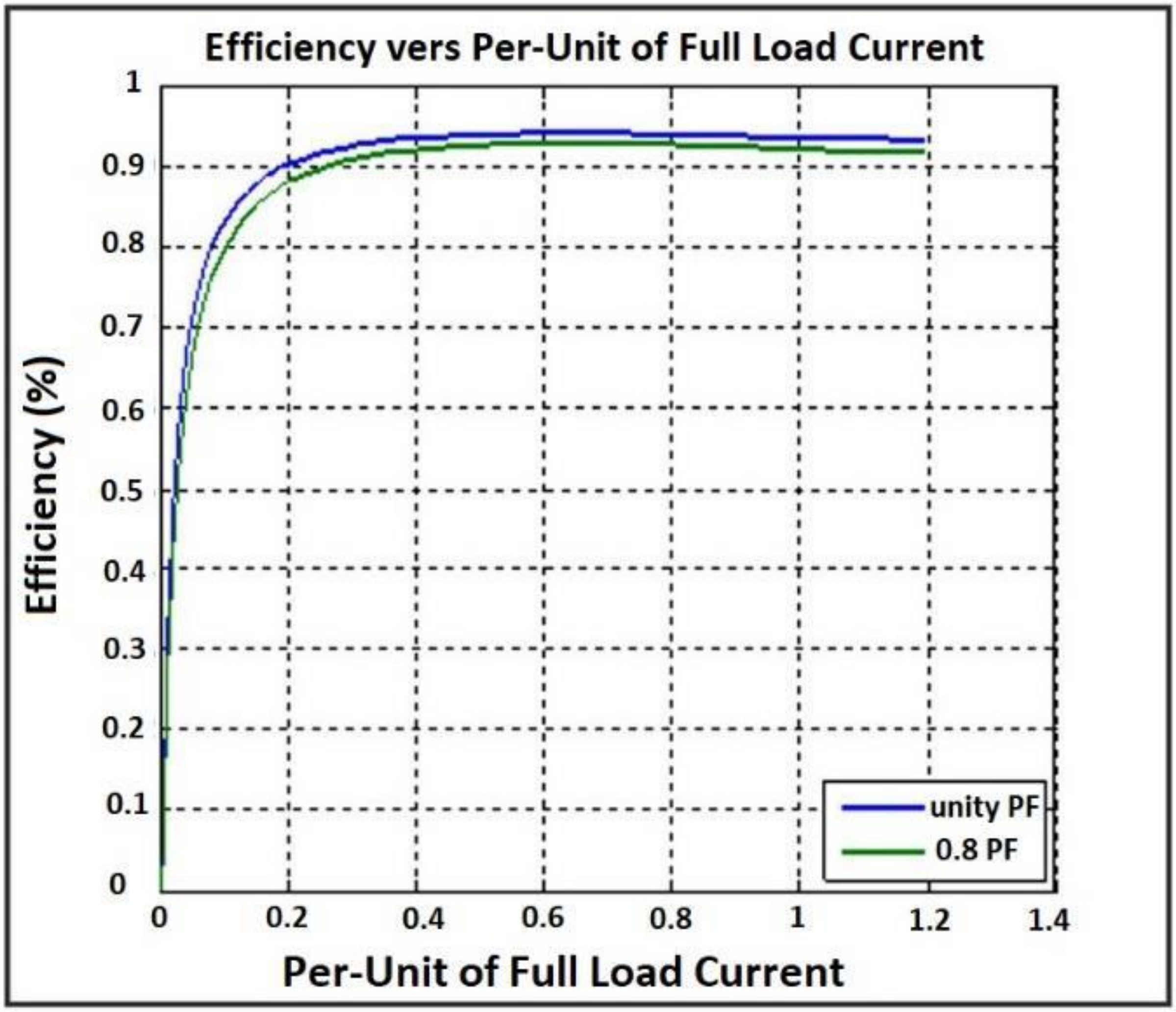
Transformer open-circuit simulation results, (a) Open-circuit power vs. time, (b) Open-circuit current vs. time, (c) Open-circuit voltage vs. time







Simulation results of transformer short-circuit test, (a) Short-circuit power vs time, b) Short-circuit current vs time, (c) Short-circuit voltage vs time



Load test (efficiency vs. load current)

RESULT:

Exp. No.: 3a **Date:**

SIMULATION OF BOOST CONVERTERS

AIM:

To simulate boost converters using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAMS:

PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

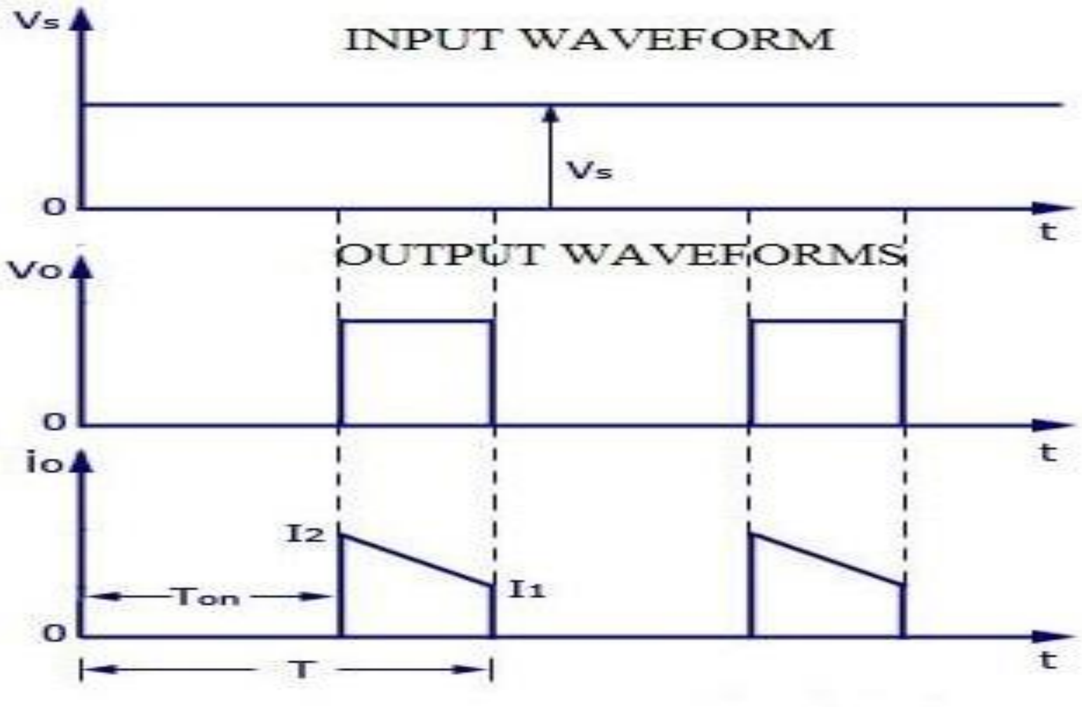
MODEL CALCULATIONS:

$$V0 = \frac{VS}{1-\alpha} \quad where\alpha = \frac{Ton}{T}$$

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Experiment No:3 b **Date:**

SIMULATION OF BUCK CONVERTERS

AIM:

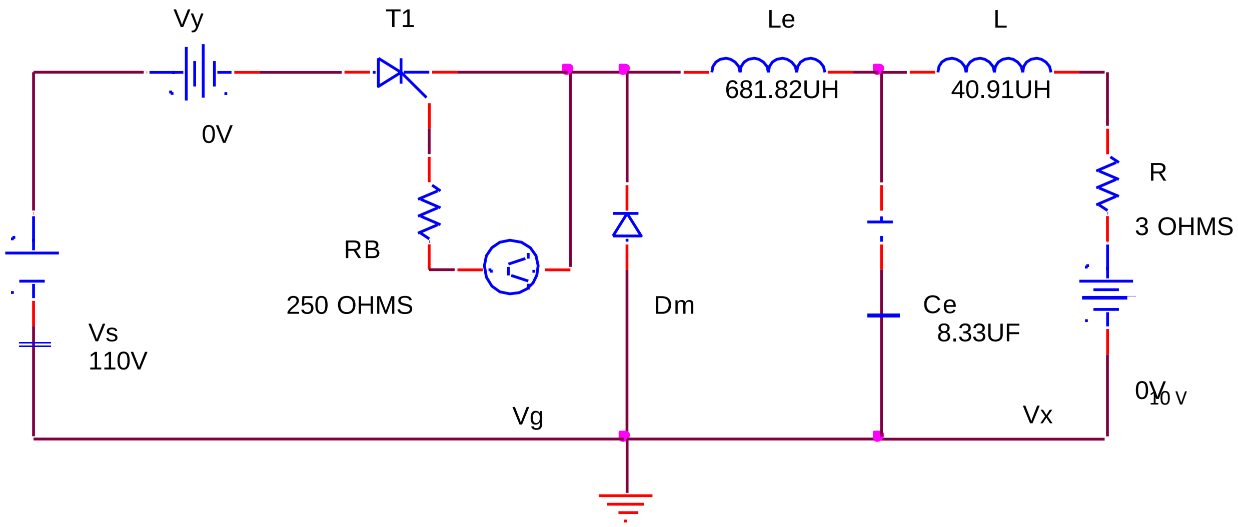
To analyze Buck chopper using Pspice.

SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAM:

Buck chopper



PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL CALCULATIONS :

$$V0 = \alpha * VS = \frac{Ton}{T} VS$$

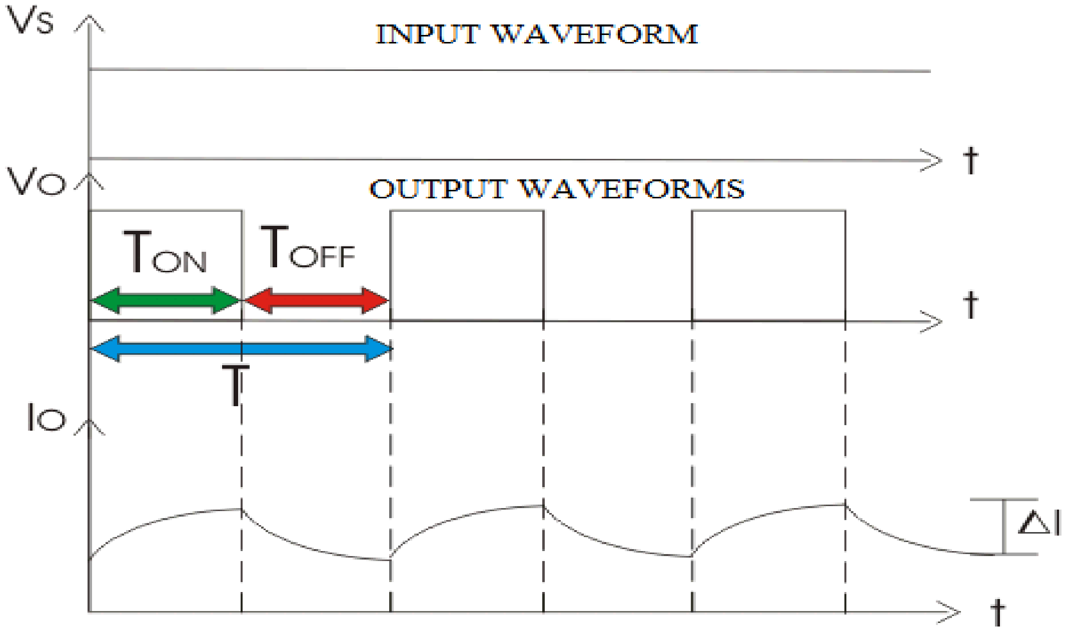
$=$

$=$

RESULT:

APPLICATIONS:

MODEL WAVEFORMS:



Exp. No.:4 **Date:**

STUDY ON THE DESIGN OF PI CONTROLLERS AND STABILITY ANALYSIS FOR A DC-DC BUCK CONVERTER

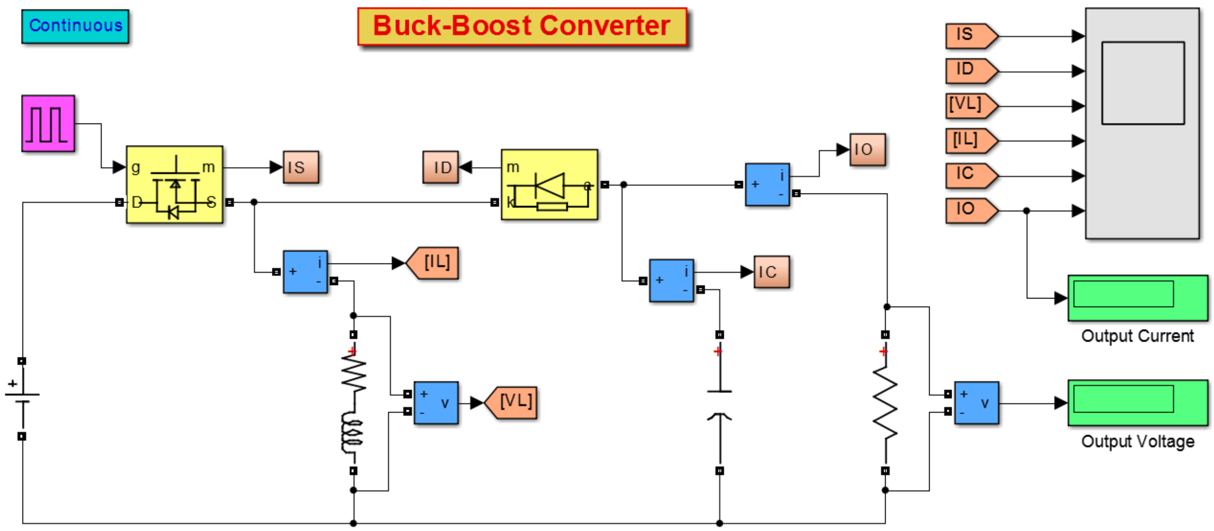
AIM:

To find out the dc-dc buck converter with digital PI-controlled is analyzed and designed considering all design parameters such as inductance current variation, output voltage ripple.

SOFTWARE REQUIRED:

PSPICE – Personal Computer Simulated Program with Integrated Circuit Emphasis.

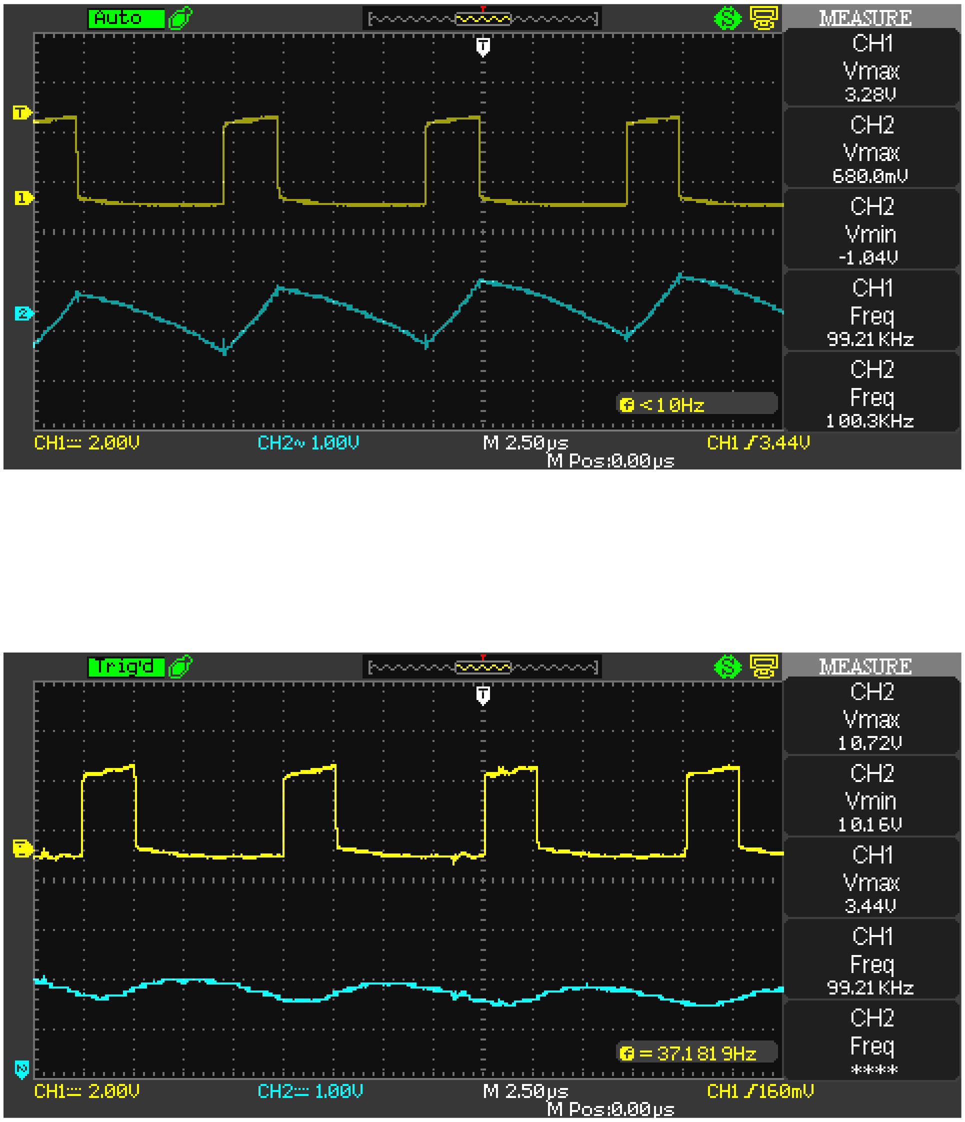
CIRCUIT DIAGRAM:



PROCEDURE:

1. Open PSpice A/D windows
2. Create a new circuit file
3. Enter the program representing the nodal interconnections of various components
4. Run the program
5. Observe the response through all the elements in the output file
6. Observe the voltage, current graph of any in probe window.

THEORETICAL CALCULATIONS



Experimental Results for Vi=48V and R=10Ω
(a) PWM, Inductance Current (b) PWM, Output Voltage

RESULT:

Expt. No: 5 **Date:**

SINE-PWM TECHNIQUES FOR SINGLE-PHASE HALF-BRIDGE, FULL-BRIDGE AND THREE-PHASE INVERTERS

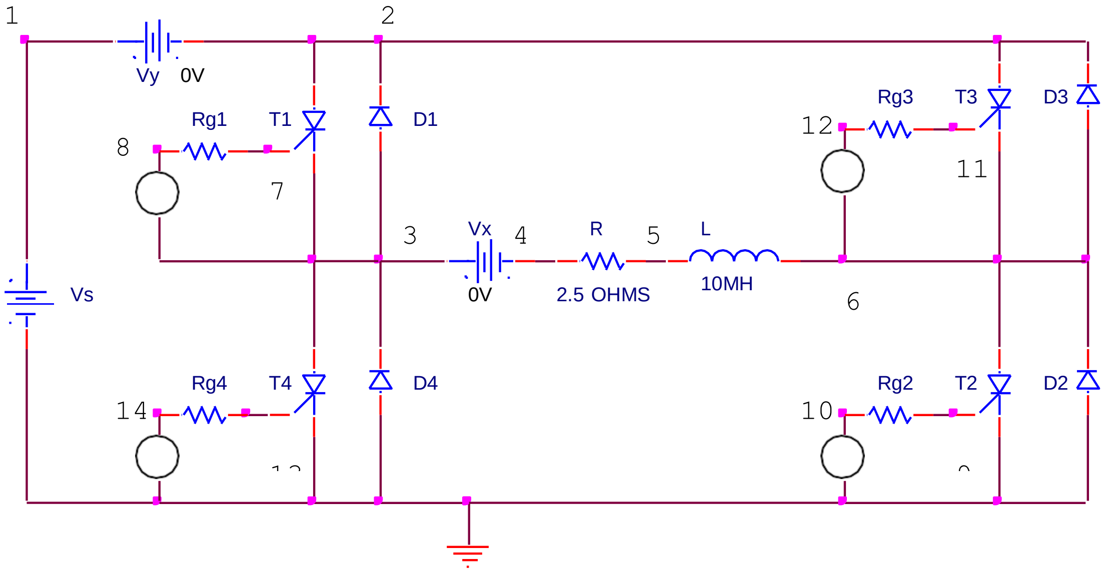
AIM:

PSpice analysis of single phase inverter with PWM control.

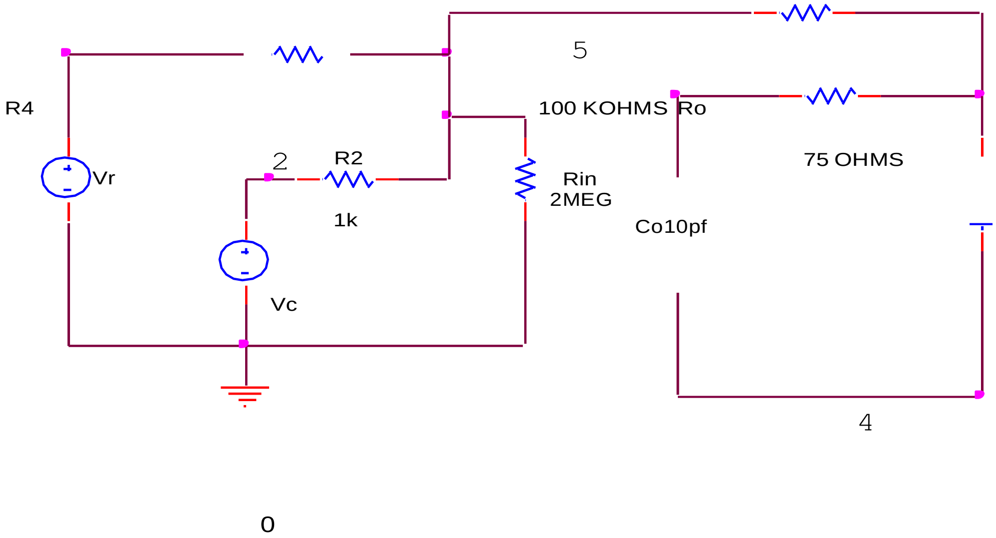
SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

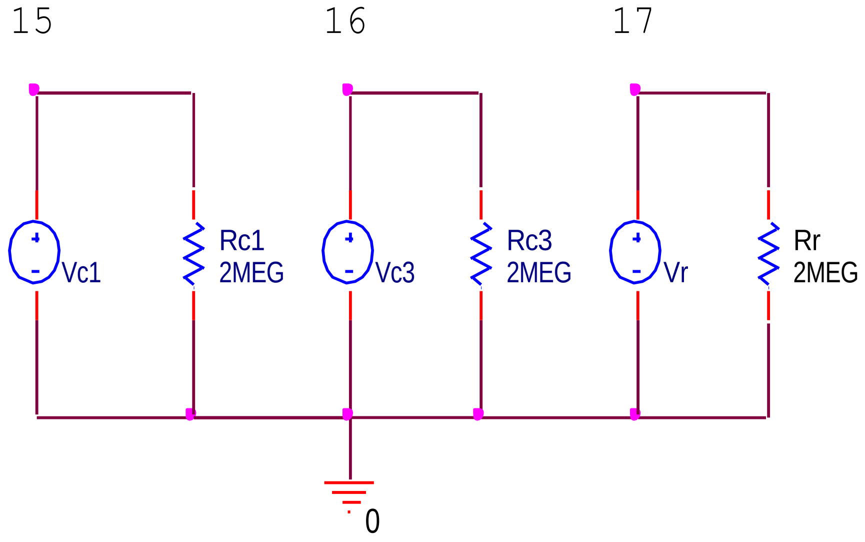
CIRCUIT DIAGRAM:



PWM Generator



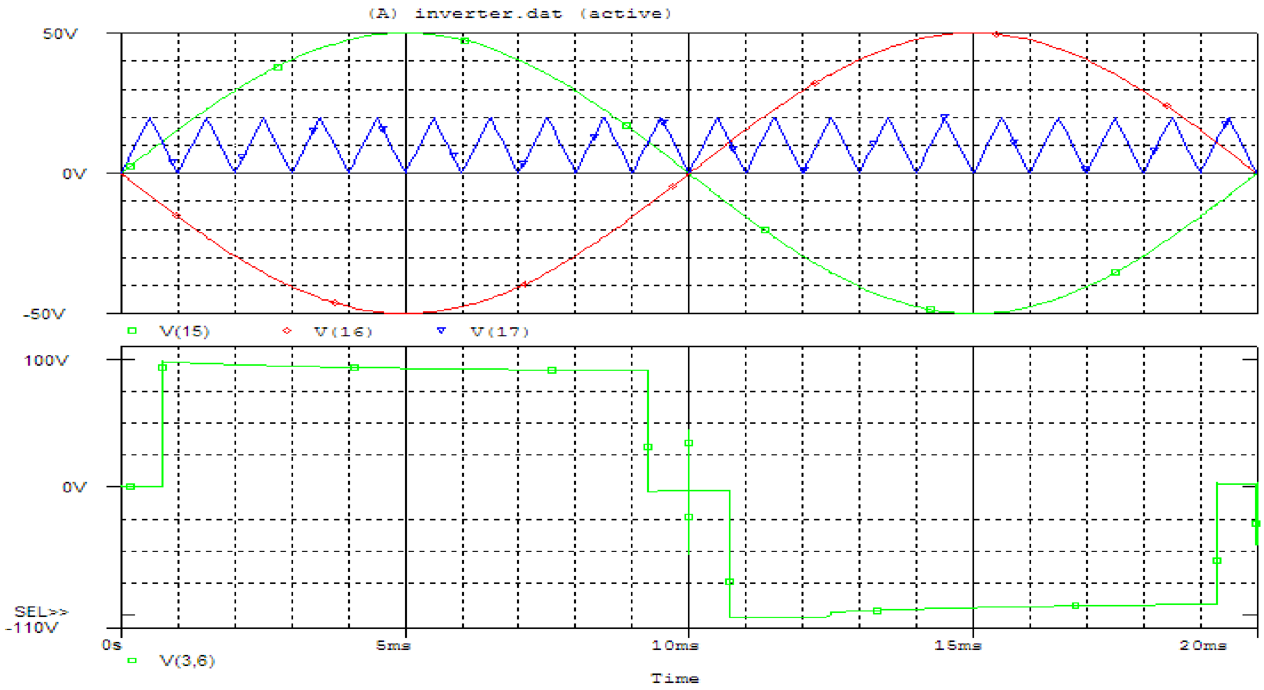
Carrier and Reference signals



PROCEDURE:

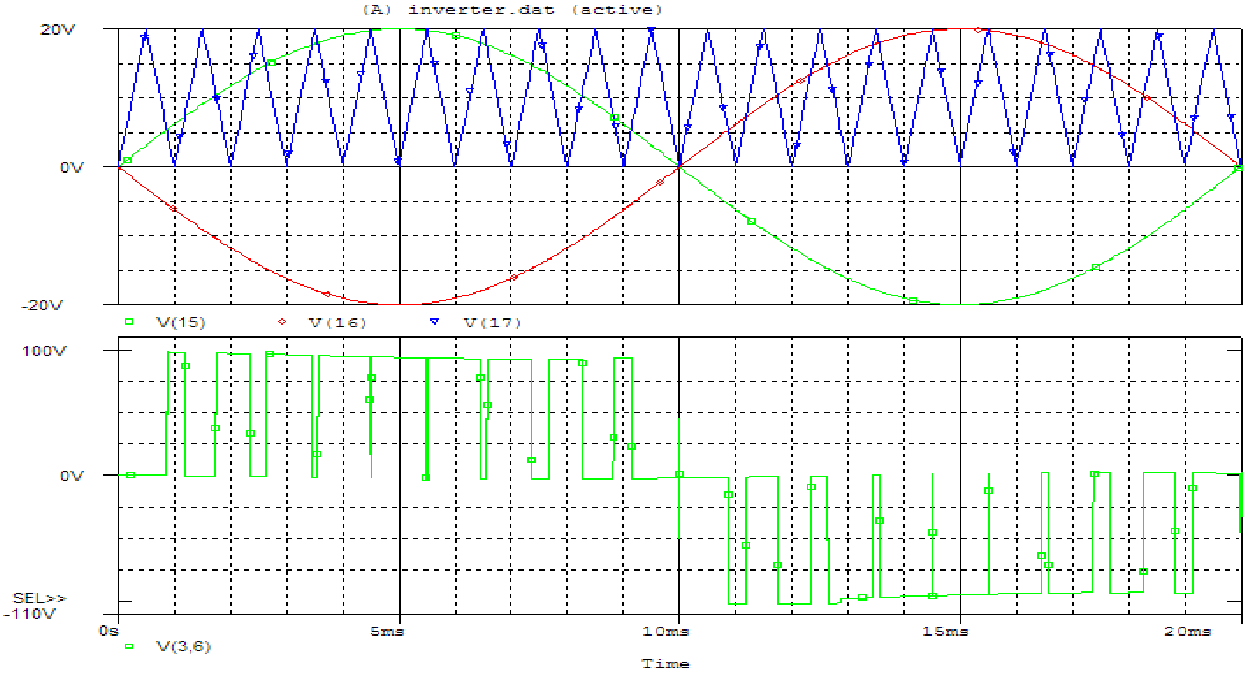
1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

MODEL WAVEFORM:



AMPLITUDE OF ‘VC1&VC2’>AMPLITUE OF ‘VR’

AMPLITUDE OF ‘VC1&VC2’=AMPLITUE OF ‘VR’



MODEL CALCULATIONS:

CASDE 'A':OVER MADULATION (Ar >Ac)

==5>1
>Mr>1
=>Ar/Ac
=

N=number of pulses per half cycle

N=1

CASDE 'A':UNDER MADULATION (Ar <Ac)

=>Mr<1
=>Ar/Ac
=
=

N-1=number of pulses per half cycle

$N = fc/(2\,fr) =$

$N - 1 =$

RESULT:

PSPICE ANALYSIS OF SINGLE PHASE FULL CONVERTERWITH RL & RLE LOADS

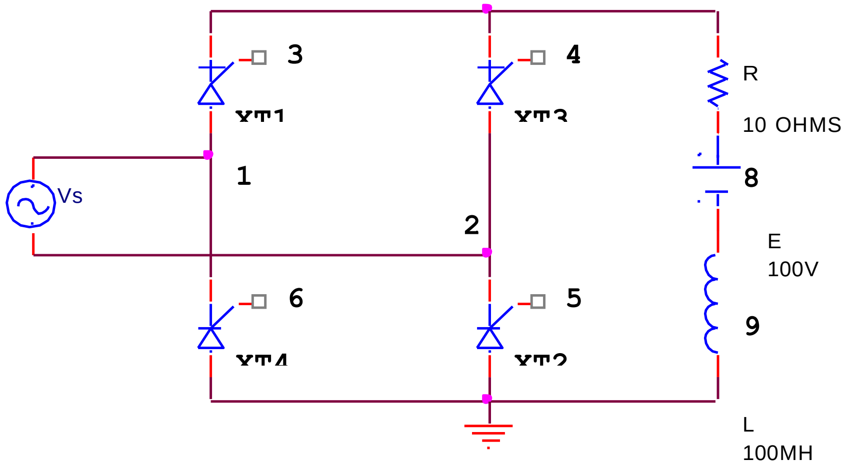
AIM:

To analyze the single phase full converter with RL and RLE Loads.

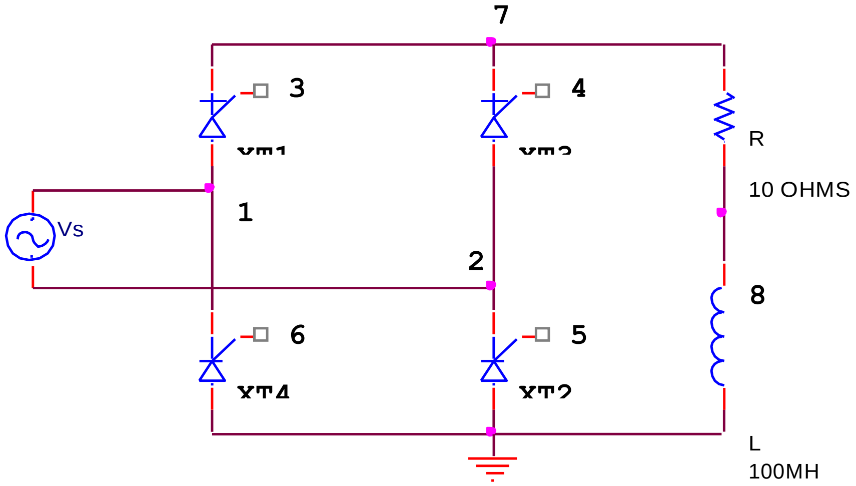
SIMULATION TOOLS REQUIRED:

- PC with PSPICE Software

CIRCUIT DIAGRAMS:



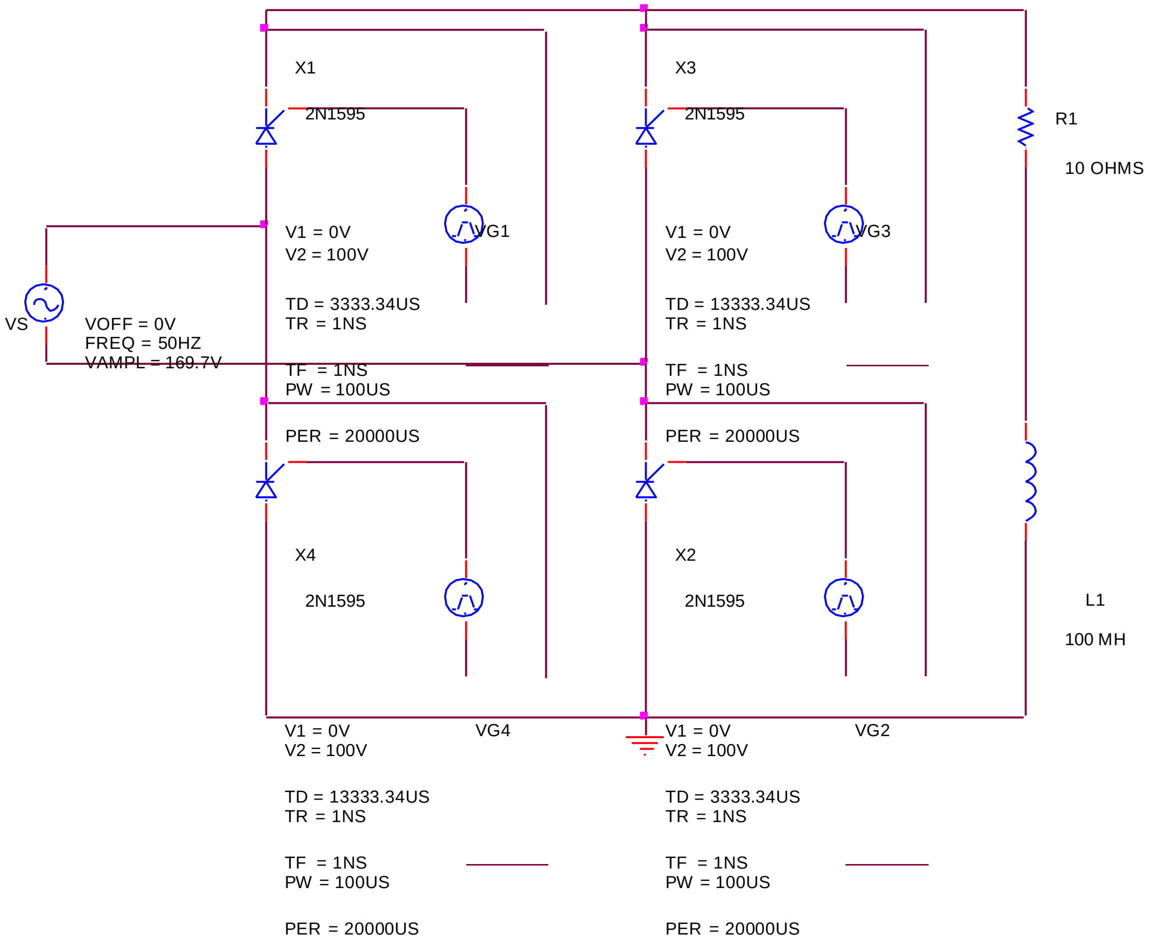
Single Phase full converter with RLE load



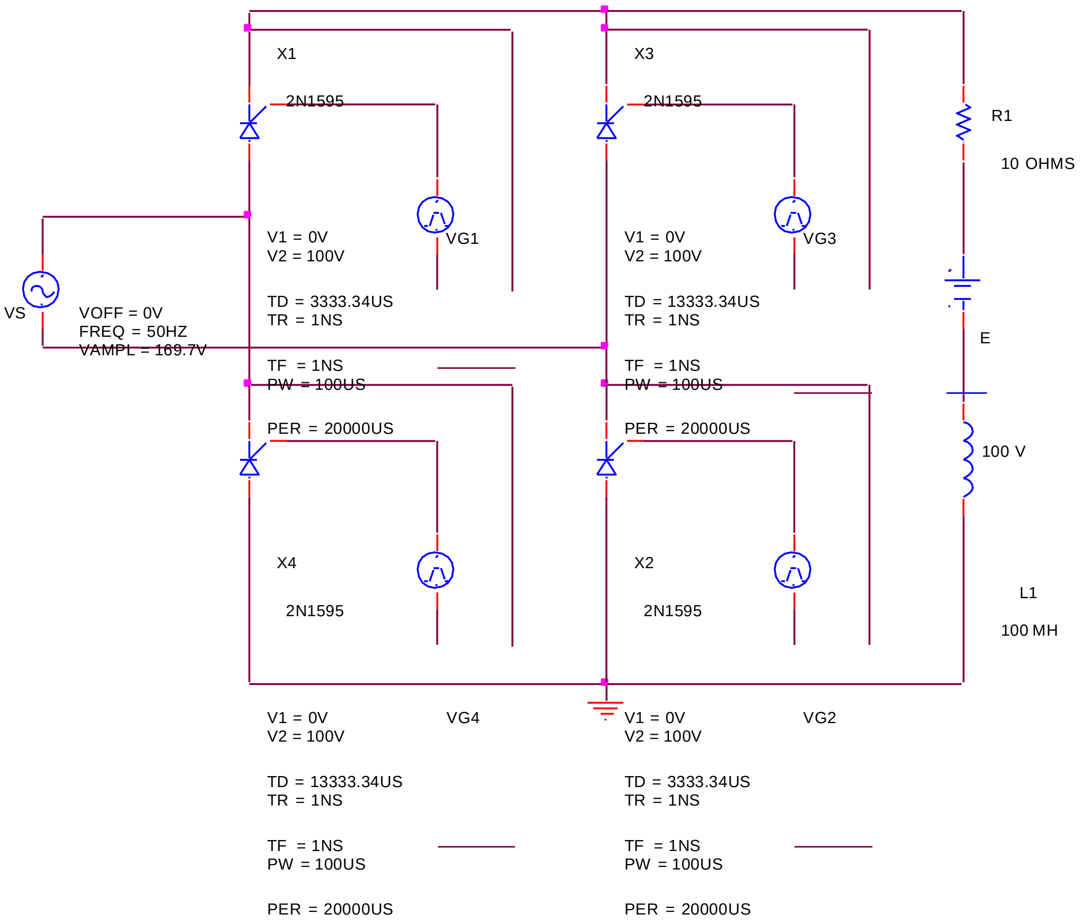
0

CIRCUIT DIAGRAMS FOR ANALYSIS USING CIRCUIT:

Single Phase full converter with RL load



Single Phase full converter with RLE load



PROCEDURE:

1. Write the program in a new text file in PSpice AD.
2. Save the file using the notation filename.cir.
3. Activate the file by opening it.
4. Run the simulation process using blue button.
5. By clicking Add Trace icon, get the required waveform.

THERITICAL CALCULATIONS:

A)FOR RL LOAD

$$V0 = \frac{2V_M}{\Pi} COS(\alpha)$$

$$V0 = \frac{2V_M}{\Pi} COS(\_\_\_\_\_\_)$$

$= \_\_\_\_\_\_\_\_ V$

B)FOR RLE LOAD

At $\omega t = \alpha$ i.e. at $t = \alpha / \omega$. $i_o = 0$

We know $\omega t = \alpha => V_m \sin(\theta) = E$

Min value of firing angle $\theta = \sin^{-1}(\frac{E}{V_m}) = \sin^{-1}(\_) = 3$

Max value of firing angle $\theta_2 = 180 - \theta_1 =$

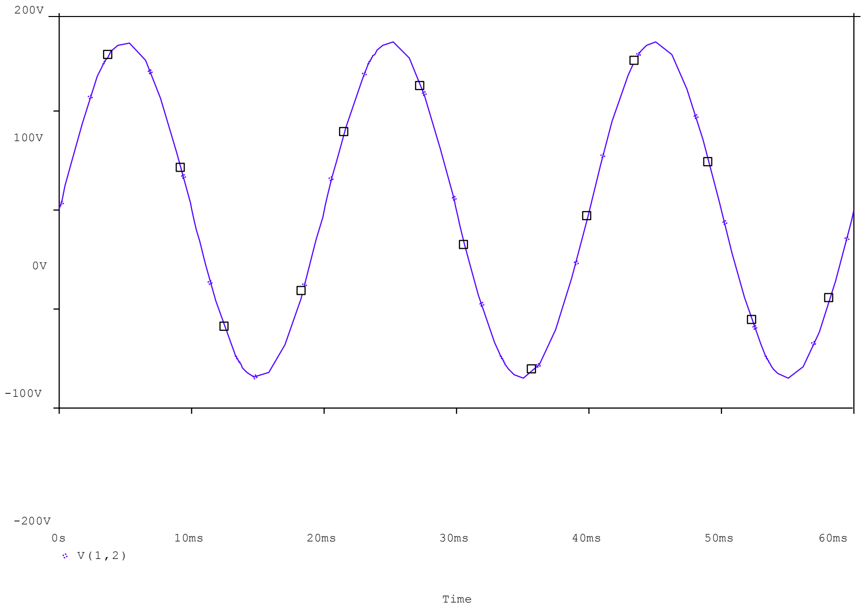
RESULT:

APPLICATIONS:

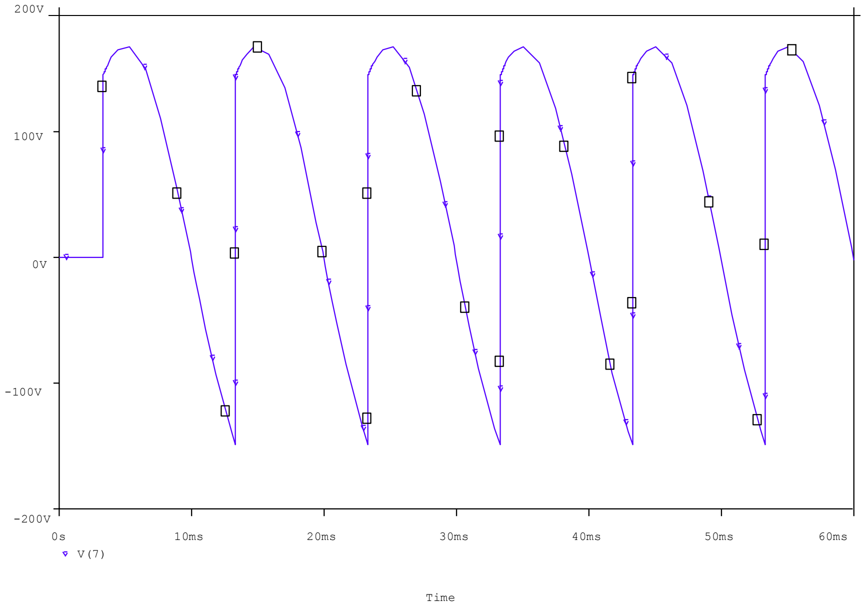
The single-phase full-wave controlled rectifier is used to control power flow in many applications (e.g., power supplies, variable-speed dc motor drives, and input stages of other converters)

MODEL WAVEFORMS FOR FULL CONVERTER:

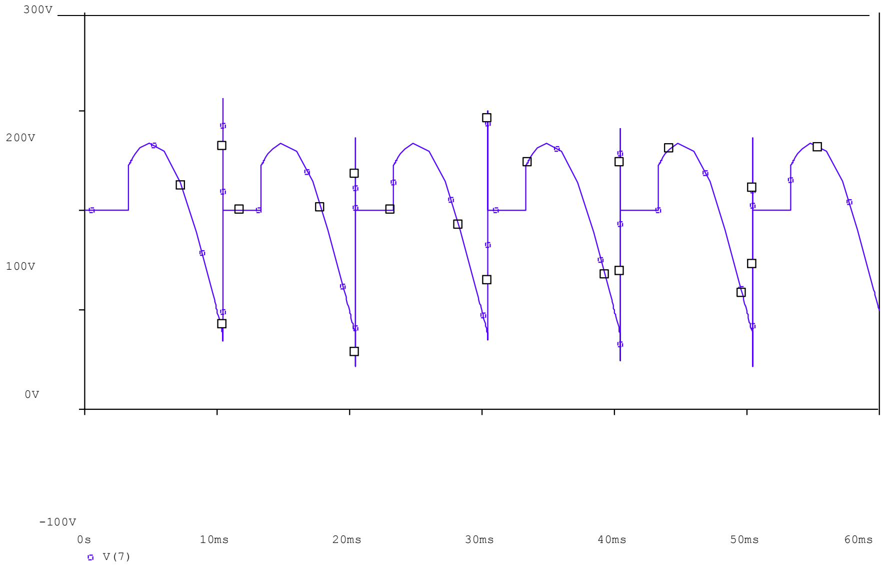
INPUT WAVEFORM



OUTPUT WAVEFORM WITH RL LOAD



OUTPUT WAVEFORM WITH RLE LOAD



Expt. No: 6 **Date:**

Economic Load Dispatch of (i) Thermal Units and (ii) Thermal Plants using Conventional method

AIM:
To develop program for economic load dispatch problem using lambda iterative method

APPARATUS:
Desktop Computers with MATLAB software

THEORY:
A modern power system is invariably fed from a number of power plants. Research and development has led to efficient power plant equipment. A generating unit added to the systemtoday is likely to be more efficient than the one added some time back. With a very large numberof generating units at hand, it is the job of the operating engineers to allocate the loads betweenthe units such that the operating costs are the minimum. The optimal load allocation is byconsidering a system with any number of units. The loads should be so allocated among thedifferent units that every unit operates at the same incremental cost. This criterion can bedeveloped mathematically by the method of lagrangian multiplier.

Statement of Economic Dispatch Problem:
In a power system, with negligible transmission losses and with N number of spinning thermal generating units the total system load PD at a particular interval can be met by different sets of generation schedules.

$P_{G1(K)}$,$P_{G2(K)}$,……..$P_{GN(K)}$ k =1,2,……….NS

Out of these NS sets of generation schedules, the system operator has to choose that setof schedule which minimizes the system operating cost which is essentially the sum of theproduction costs of all the generating units. This economic dispatch problem is mathematicallystated as an optimization problem. Given the number of available generating units Ns theirproduction cost function, their operating limits and the system load PD.

To determine the set of generating schedule PG,

Min $F_T = F_i.\ P_{Gi\ Ni}$ (1)

where$F_i.\ P_{Gi} = a_i P_{Gi2} + b_i P_{Gi} + c_i$ i=1,2……N (2)

$\Phi = P_{GiNi} - P_D = 0$ (3)

$P_{Gi\,min} \le P_{Gi} \le P_{Gimax} \cdot$ (4)

where ai, bi and ci are constants.

The ED problem is given by the equations . By omitting the inequality constraint thereduced ED problem may be restated as an unconstrained optimization problem by augmenting the objective function with the constraint function Φ multiplied by Lagrange multiplier λ to obtain the Lagrange function L as,

The solution to ED problem can be obtained by solving simultaneously the necessary conditions and which state that the economic generation schedules not only satisfy the system powerbalance equation but also demand that the incremental cost rates of all the units be equal to λwhich can be interpreted as "incremental cost of received power" when the inequality constraints are included in the ED problem the necessary condition gets modified as The solution to the ED problem with the production cost function assumed to be a quadraticfunction, equation , can be obtained by simultaneously solving and using a directmethod as given below, Substituting Equation in Equation we obtain The method of solution involves computing λ using equation and then computing theeconomic schedules PGi; i=1,2,........N using equation . In order to satisfy the operating limits the following iterative algorithm is to be used.

ALGORITHM:

Step 1: Choose appropriate value of Lagrangian multiplier λ
Step 2: Start iteration iter=0
Step 3: Iteration iter=iter+1
Step 4: Solve for power generated by i_{th} unit using equation
Step 5: Check if any Pi is beyond or below the inequality constant
If Pi <Pi,min, fix Pi= Pi,min
If Pi>Pi,max, fix Pi= Pi,max
Step 6: Calculate the power loss using the equation
Step 7: Calculate power mismatch using the formula,
Step 8: If, then increment λ, λnew=λ+0.001 and go to step 3else go to step 9
Step 9: If, then decrement λ, λnew=λ-0.001 and go to step 3else go to step 10
Step 10: If ΔP is less than tolerance value , print the values of generatedpower and losses
Step11: Stop

MATLAB Program:

```
clc;
clear all; % a b c fc max min 49 | P a g e
```

```
data= [0.00142 7.20 510 1.1 600 150
0.00194 7.85 310 1 400 100
0.00482 7.97 78 1 200 050];
ng=length(data(:,1));
a=data(:,1);
b=data(:,2);
c=data(:,3);
fc=data(:,4);
pmax=data(:,5);
pmin=data(:,6);
% loss=[0.00003 0.00009 0.00012];
loss=[ 0 0 0];
C=fc.*c; B=fc.*b; A=fc.*a;
la=1; pd=850; acc=0.2;
diff=1;
145
while acc<(abs(diff));
for i=1:ng;
p(i)= (la-B(i))/(2*(la*loss(i)+A(i)));
if p(i)<pmin(i);
p(i)=pmin(i);
end;
if p(i)>pmax(i);
p(i)=pmax(i);
end;
end;
LS=sum(((p.*p).*loss));
diff=(pd+LS-sum(p));
if diff>0
la=la+0.001;
else la=la-0.001;
end;
end;
PowerShared=p
Lambda=la
Loss=LS
```

OUTPUT:

```
a). When loss = [0.00003 0.00009 0.00012]
Power Shared = 435.1026 299.9085 130.6311
Lambda = 9.5290
Loss = 15.8222
b). When loss = 0
Power Shared = 393.0858 334.5361 122.1992
Lambda = 9.1490
Loss = 0
```

RESULT:

LAB VIVA QUESTIONS:

1. Define economic load disptach
2. State the objectives of economic load dispatch.
3. Name the methods of finding economic dispatch

EXPERIMENT – 7 DATE:

TRANSIENT STABILITY ANALYSIS

AIM:

To develop program for transient stability analysis for single machine connected to infinite bus

APPARATUS:

Desktop Computers with MATLAB software

THEORY:

Stability: Stability problem is concerned with the behavior of power system when it is subjected todisturbance and is classified into small signal stability problem if the disturbances are small andtransient stability problem when the disturbances are large.

Transient stability: When a power system is under steady state, the load plus transmission loss equals to the generation in the system. The generating units run a synchronous speed and system frequency, voltage, current and power flows are steady. When a large disturbance such as three phase fault, loss of load, loss of generation etc., occurs the power balance is upset and the generating units rotors experience either acceleration or deceleration. The system may come back to a steady state condition maintaining synchronism or it may break into subsystems or one or more machines may pull out of synchronism. In the former case the system is said to be stable and in the later case it is said to be unstable.

Reactive power $Qe = \sin(\cos_{-1}(p.f))$

Stator current, $I_t=S^*/E_{t^*}$

$= Pe-jQe/E_{t^*}$

Voltage behind transient condition

Voltage of infinite bus

Angular separation between E_1 and E_B

Pre-fault Operation:

During Fault Condition:

Find out X from the equivalent circuit during fault condition

Post fault Condition:

Find out X from the equivalent circuit during post fault condition

Critical Clearing Angle:

Critical Clearing Time:

MATLAB PROGRAM:

a)
```
Pm = 0.8; E = 1.17; V = 1.0;
X1 = 0.65; X2 = inf; X3 = 0.65;
eacfault(Pm, E, V, X1, X2, X3)
```

b)
```
Pm = 0.8; E = 1.17; V = 1.0;
X1 = 0.65; X2 = 1.8; X3 = 0.8;
eacfault(Pm, E, V, X1, X2, X3)
```

eacfault

```
function eacfault(Pm, E, V, X1, X2, X3)
if exist('Pm')~=1
Pm = input('Generator output power in p.u. Pm = '); else, end
if exist('E')~=1
E = input('Generator e.m.f. in p.u. E = '); else, end
if exist('V')~=1
V = input('Infinite bus-bar voltage in p.u. V = '); else, end
if exist('X1')~=1
X1 = input('Reactance before Fault in p.u. X1 = '); else, end
if exist('X2')~=1
X2 = input('Reactance during Fault in p.u. X2 = '); else, end
if exist('X3')~=1
X3 = input('Reactance aftere Fault in p.u. X3 = '); else, end
Pe1max = E*V/X1; Pe2max=E*V/X2; Pe3max=E*V/X3;
delta = 0:.01:pi;
Pe1 = Pe1max*sin(delta); Pe2 = Pe2max*sin(delta); Pe3 = Pe3max*sin(delta);
d0 =asin(Pm/Pe1max); dmax = pi-asin(Pm/Pe3max);
cosdc = (Pm*(dmax-d0)+Pe3max*cos(dmax)-Pe2max*cos(d0))/(Pe3max-Pe2max);
if abs(cosdc) > 1
fprintf('No critical clearing angle could be found.\n')
fprintf('system can remain stable during this disturbance.\n\n')
return
else, end
dc=acos(cosdc);
if dc >dmax
fprintf('No critical clearing angle could be found.\n')
fprintf('System can remain stable during this disturbance.\n\n')
return
else, end
Pmx=[0 pi-d0]*180/pi; Pmy=[Pm Pm];
x0=[d0 d0]*180/pi; y0=[0 Pm]; xc=[dc dc]*180/pi; yc=[0 Pe3max*sin(dc)];
xm=[dmaxdmax]*180/pi; ym=[0 Pe3max*sin(dmax)];
d0=d0*180/pi; dmax=dmax*180/pi; dc=dc*180/pi;
x=(d0:.1:dc);
y=Pe2max*sin(x*pi/180);
y1=Pe2max*sin(d0*pi/180);
y2=Pe2max*sin(dc*pi/180);
```

```
x=[d0 x dc];
y=[Pm y Pm];
xx=dc:.1:dmax;
h=Pe3max*sin(xx*pi/180);
xx=[dc xx dmax];
hh=[Pm h Pm];
delta=delta*180/pi;
if X2 == inf
fprintf('\nFor this case tc can be found from analytical formula. \n')
H=input('To find tc enter Inertia Constant H, (or 0 to skip) H = ');
if H ~= 0
d0r=d0*pi/180; dcr=dc*pi/180;
tc = sqrt(2*H*(dcr-d0r)/(pi*60*Pm));
else, end
else, end
%clc
fprintf('\nInitial power angle = %7.3f \n', d0)
fprintf('Maximum angle swing = %7.3f \n', dmax)
fprintf('Critical clearing angle = %7.3f \n\n', dc)
if X2==inf & H~=0
fprintf('Critical clearing time = %7.3f sec. \n\n', tc)
else, end
h = figure; figure(h);
fill(x,y,'m')
hold;
fill(xx,hh,'c')
plot(delta, Pe1,'-', delta, Pe2,'r-', delta, Pe3,'g-', Pmx, Pmy,'b-', x0,y0,
xc,yc, xm,ym), grid
Title('Application of equal area criterion to a critically cleared system')
xlabel('Power angle, degree'), ylabel(' Power, per unit')
text(5, 1.07*Pm, 'Pm')
text(50, 1.05*Pe1max,['Critical clearing angle = ',num2str(dc)])
axis([0 180 0 1.1*Pe1max])
hold off;
```

OUTPUT:

```
a) To find tc enter Inertia Constant H, (or 0 to skip) H = 5
Initial power angle = 26.388
Maximum angle swing = 153.612
Critical clearing angle = 84.775
Critical clearing time = 0.260 sec.
```

MATLAB PROGRAM:

```
E=1.35; V=1.0; H=9.94; X=0.65; Pm=0.6; D=0.138; f0=60;
Pmax=E*V/X, d0=asin(Pm/Pmax)
Ps=Pmax*cos(d0)
wn=sqrt(pi*60/H*Ps)
z=D/2*sqrt(pi*60/(H*Ps))
wd=wn*sqrt(1-z^2),fd=wd/(2*pi)
tau=1/(z*wn)
th=acos(z)
```

```
Dd0=10*pi/180;
t=0:.01:3;
Dd=Dd0/sqrt(1-z^2)*exp(-z*wn*t).*sin(wd*t+th);
d=(d0+Dd)*180/pi;
Dw=-wn*Dd0/sqrt(1-z^2)*exp(-z*wn*t).*sin(wd*t);
f=f0+Dw/(2*pi);
subplot(2,1,1),plot(t,d),grid
xlabel('t sec'),ylabel('Delta degree')
```

```
subplot(2,1,2),plot(t,f),grid
xlabel('t sec'),ylabel('frquency hertz')
subplot(111)
```

OUTPUT:

```
Pmax = 2.0769
d0 = 0.2931
Ps = 1.9884
wn = 6.1405
z = 0.2131
wd = 5.9995
fd = 0.9549
tau = 0.7643
th = 1.3561
```

EXPERIMENT – 8 DATE:

REACTIVE POWER CONTROL IN A TRANSMISSION SYSTEM (FERRANTI EFFECT, EFFECT OF SHUNT INDUCTOR)

AIM: To find Ferranti effect of a 5000 kM transmission line and to plot the locus of voltage for the given problem and verify results in MATLAB.

PROBLEM:

A 3-Phase 50 Hz transmission line is 5000 kM long. The line parameters are R= 0.125Ω/km,X= 0.4 Ω/km and Y= 2.8*10-6 mho/km. If the line is open circuited with a receiving end voltage of 220KV, find the rms value and phase angle of the following. Use the receiving-end line to neutral voltage as reference.

(a) The incident and reflected voltages to neutral at the receiving-end.

(b) The incident and reflected voltages to neutral at 200 km from the receivingend.

(c) The resultant voltage at 200 km from the receiving end.

Solve the problem theoretically. Vary the length of the long transmission line in steps of 10 KM from zero (receiving end) to 5000KM (sending end), and plot the sending end voltage phasor using MATLAB.

THEORETICAL SOLUTION:

MATLAB PROGRAM:

```
%Program to illustrate Ferranti effect
%it simulates the effect by varying the length of transmission line from
%zero(receiving end) to 5000km in steps of 10km
%and plots the sending end voltage phasor
clc
clear all
VR=220e3/sqrt(3);
alpha=0.163e-3;
beta=1.0683e-3;
L=5000;
k=1;
for i=0:10:L,
VS=(VR/2)*exp(alpha*i)*exp(j*beta*i)+(VR/2)*exp(-alpha*i)*exp(-j*beta*i);
X(k)=real(VS);
Y(k)=imag(VS);
k=k+1;
p(k)=VS;
q(k)=i;
end
figure(1);
plot(p,q)
figure(2);
plot(X,Y)
```

EXPECTED OUTPUT:

RESULT

Expt. No: 9 **DATE:**

FAULT STUDIES USING ZBUS MATRIX

AIM:

Develop program for Z_{BUS} building algorithm.

APPARATUS:

Desktop Computers with MATLAB software

THEORY:

The Y_{bus} /Z_{bus} matrix constitutes the models of the passive portions of the power network. The impedance matrix is a full matrix and is most useful for short circuit studies. An algorithm for formulating [Z_{bus}] is described in terms of modifying an existing bus impedance matrixdesignated as [Z_{bus}]old. The modified matrix is designated as [Z_{bus}]new. The network consists of areference bus and a number of other buses. When a new element having self-impedance,Z_b isadded, a new bus may be created (if the new element is a tree branch) or a new bus may not becreated (if the new element is a link). Each of these two cases can be subdivided into two typesso that Z_b may be added in the following ways:

1. Adding Z_b from a new bus to reference bus.
2. Adding Z_b from a new bus to an existing bus.
3. Adding Z_b from an existing bus to reference bus.
4. Adding Z_b between two existing buses.

Type 1 modification:

In type 1 modification, an impedance Z_b is added between a new bus p and the reference bus asshown in Figure 1

Figure 1: Addition of impedance between new bus and reference bus

Let the current through bus „p" be Ip, then the voltage across the bus p is given by,

$V_p = I_p Z_b$

The potential at other buses remains unaltered and the system equations can be written as,

Type 2 modification:

In type 2 modification, an impedance Z_b is added between a new bus *p* and an existing bus *k* as shown in Figure 2. The voltages across the bus *k* and *p* can be expressed as,

$V_k(new) = V_k + Ip\ Z_{kk}$

$Vp = V_k(new) + Ip\ Zp = V_k + Ip(Z_b + Z_{kk})$

where, V_k is the voltage across bus *k* before the addition of impedance Z_b

Z_{kk} is the sum of all impedance connected to bus *k*.

Figure 2: Addition of impedance between new bus andexisting bus

The system of equations can be expressed as,

Type 3 Modification:

In this modification, an impedance Z_b is added between a existing bus k and a reference bus. Then the following steps are to be followed:

1. Add Z_b between a new bus p and the existing bus k and the modifications are done as in type 2.
2. Connect bus p to the reference bus by letting Vp = 0.

To retain the symmetry of the Bus Impedance Matrix, network reduction technique can beused to remove the excess row or column.

Type 4 Modification:

In this type of modification, an impedance Z_b is added between two existing buses j and k asshown in Figure 3. From Figure 3, the relation between the voltages of bus k and j can be writtenas,

$V_k – V_j = I_bZ_b$

Figure 3: Addition of impedance between two existing buses

The voltages across all the buses connected to the network changes due to the addition of impedance Zb and they can be expressed as,

$V_1 = Z_{11}I_1 + Z_{12}I_2 + \text{- - - - - - - - -} + Z_{1j}(I_j + I_b) + Z_{1k}(I_k – I_b) + \text{- - -}$

$V_2 = Z_{21}I_1 + Z_{22}I_2 + \text{- - - - - - - - -} + Z_{2j}(I_j + I_b) + Z_{2k}(I_k – I_b) + \text{- - -}$

$V_j = Z_{j1}I_1 + Z_{j2}I_2 + \text{- - - - - - - - -} + Z_{jj}(I_j + I_b) + Z_{jk}(I_k – I_b) + \text{- - -}$

$V_k = Z_{k1}I_1 + Z_{k2}I_2 + \text{- - - - - - - - -} + Z_{kj}(I_j + I_b) + Z_{kk}(I_k – I_b) + \text{- - -}$

$V_n = Z_{n1}I_1 + Z_{n2}I_2 + \text{- - - - - - - - -} + Z_{nj}(I_j + I_b) + Z_{nk}(I_k – I_b) + \text{- - -}$

The system of equations can be rewritten as,

where,

$Z_{bb} = Z_{jj} + Z_{kk} – 2\,Z_{jk} + Z_b$

Procedure for formation of Z_{bus}matrix:

Step1: Number the nodes of the given network, starting with those nodes at the ends of branches connected to the reference node.

Step2: Start with a network composed of all those branches connected to thereference node.

Step3: Add a new node to the ith node of the existing network.

Step4: Add a branch between ith and jth nodes. Continue until all the remainingbranches are connected.

MATLAB PROGRAM:

```
clc;
clear all;
nb=input('Enter the number of buses');
Zbusm=null(nb,nb);
ele=input('Enter the number of elements:');
dim=0;
for i=1:ele
disp(' ');
disp('1.Addition of Branch 2.Addition of Link');
```

```
ch=input('Enter your choice:');
p=input('enter p value');
q=input('enter q value');
val=input('Enter value to be added;');
switch(ch)
case 1
dim=dim+1;
if p==0||q==0
for row=1:dim
for col=1:dim
Zbusm(dim,dim)=val;
Zbusm(dim+1:end,dim+1:end)=0;
end;
end;
else
for i=1:dim
Zbusm(q,i)=Zbusm(p,i);
Zbusm(i,q)=Zbusm(q,i);
end;
Zbusm(q,q)=Zbusm(p,q)+val;
end;
%end;
case 2
Zbusrm=null(dim,dim);
li=dim+1;
if p==0
for i=1:li
Zbusm(li,i)=-Zbusm(q,i);
Zbusm(i,li)=Zbusm(li,i);
end;
%end
Zbusm(li,li)=-Zbusm(q,li)+val;
else
for i=1:li
Zbusm(li,i)=Zbusm(p,i)-Zbusm(q,i);
Zbusm(i,li)=Zbusm(li,i);
end;
Zbusm(li,li)=Zbusm(p,li)-Zbusm(q,li)+val
for i=1:dim
for j=1:dim
Zbusrm(i,j)=Zbusm(i,j)-(((Zbusm(i,li))*Zbusm(li,j))/Zbusm(li,li));
end;
end;
disp(Zbusrm);
Zbusm=Zbusrm
end;
end;
end
```

OUTPUT:

```
Enter the number of buses: 3
Enter the number of elements:5
1. Addition of Branch 2.Addition of Link
```

```
Enter your choice:1
```

```
enter p value0
enter q value1 26 | P a g e
```

```
Enter value to be added: 0.2
1.Addition of Branch 2.Addition of Link
Enter your choice:1
enter p value0
enter q value2
Enter value to be added;0.4
1.Addition of Branch 2.Addition of Link
Enter your choice:1
enter p value1
enter q value3
Enter value to be added;0.4
Zbusm =
0.2000 0 0.2000
0 0.4000 0
0.2000 0 0.6000
1.Addition of Branch 2.Addition of Link
Enter your choice:2
enter p value1
enter q value2
Enter value to be added;0.8
Zbusm =
0.2000 0 0.2000 0.2000
0 0.4000 0 -0.4000
0.2000 0 0.6000 0.2000
0.2000 -0.4000 0.2000 1.4000
0.1714 0.0571 0.1714
0.0571 0.2857 0.0571
0.1714 0.0571 0.5714
Zbusm =
0.1714 0.0571 0.1714
0.0571 0.2857 0.0571
0.1714 0.0571 0.5714
1.Addition of Branch 2.Addition of Link
Enter your choice:2
enter p value2
enter q value3
Enter value to be added;0.4
Zbusm =
0.1714 0.0571 0.1714 -0.1143
0.0571 0.2857 0.0571 0.2286
0.1714 0.0571 0.5714 -0.5143
-0.1143 0.2286 -0.5143 1.1429
0.1600 0.0800 0.1200
0.0800 0.2400 0.1600
0.1200 0.1600 0.3400
Zbusm =
0.1600 0.0800 0.1200
0.0800 0.2400 0.1600
0.1200 0.1600 0.3400
```

RESULT:

LAB VIVA QUESTIONS:

1. What are the diagonal elements of bus impedance matrix called?

2. When a branch of impedance Z_b is added from a new bus to the reference bus, what will be the order of the bus impedance matrix?

3. What are the off diagonal elements in Y_{bus} called?

Expt. No: 10 **DATE:**

DESIGN OF VIRTUAL PMU

AIM: To this purpose, a virtual phasor measurement unit (PMU) is tested and characterized to understand its uncertainty contribution. To achieve that, firstly, the characterization of a virtual PMU calibrator is described. Afterward, the virtual PMU calibration is performed, and the results clearly highlightits key role in the overall uncertainty.

THEORY:

To this purpose, and considering that such an aspect is not always considered or sufficiently treated, this article aims at emphasizing the importance of treating the RTS exactly as the other devices. In fact, RTSs and virtual models of physical devices should be characterized and assessed as well in order to avoid unexpected sources to the overall uncertainty.Therefore, this work has a double added value. First, a calibrator for RTS systems is presented and characterized. Second, the described calibrator is used to characterize a virtual phasor measurement unit (PMU) developed inside an RTS (the OPAL). Note that Sensors **2021**, 21, 6133 3 of 25 this study focuses on PMUs and the electrical world; however, the main concept can be extended and implemented in all fields in which simulations and DT are being used.

1. Characterization of the Calibrator

As mentioned in the previous section, the first operation to be accomplished is the characterization of the calibrator, which is presented herein. The final goal is to assess the uncertainty that can be reasonably associated with the reference test waveform generated by the calibrator. Once this operation has been performed, the virtual PMU performance can finally be evaluated. The present article only deals with the steady-state characterization for the M-class PMU because the P-class has less strict requirements, hence it is included in the M-class. The virtual PMU, in accordance with its standard IEC 60255-118-1 [47], has a reporting rate of 1 frame per second (fps); consequently, the dynamic performance requirements and the out-of-band requirements do not apply.

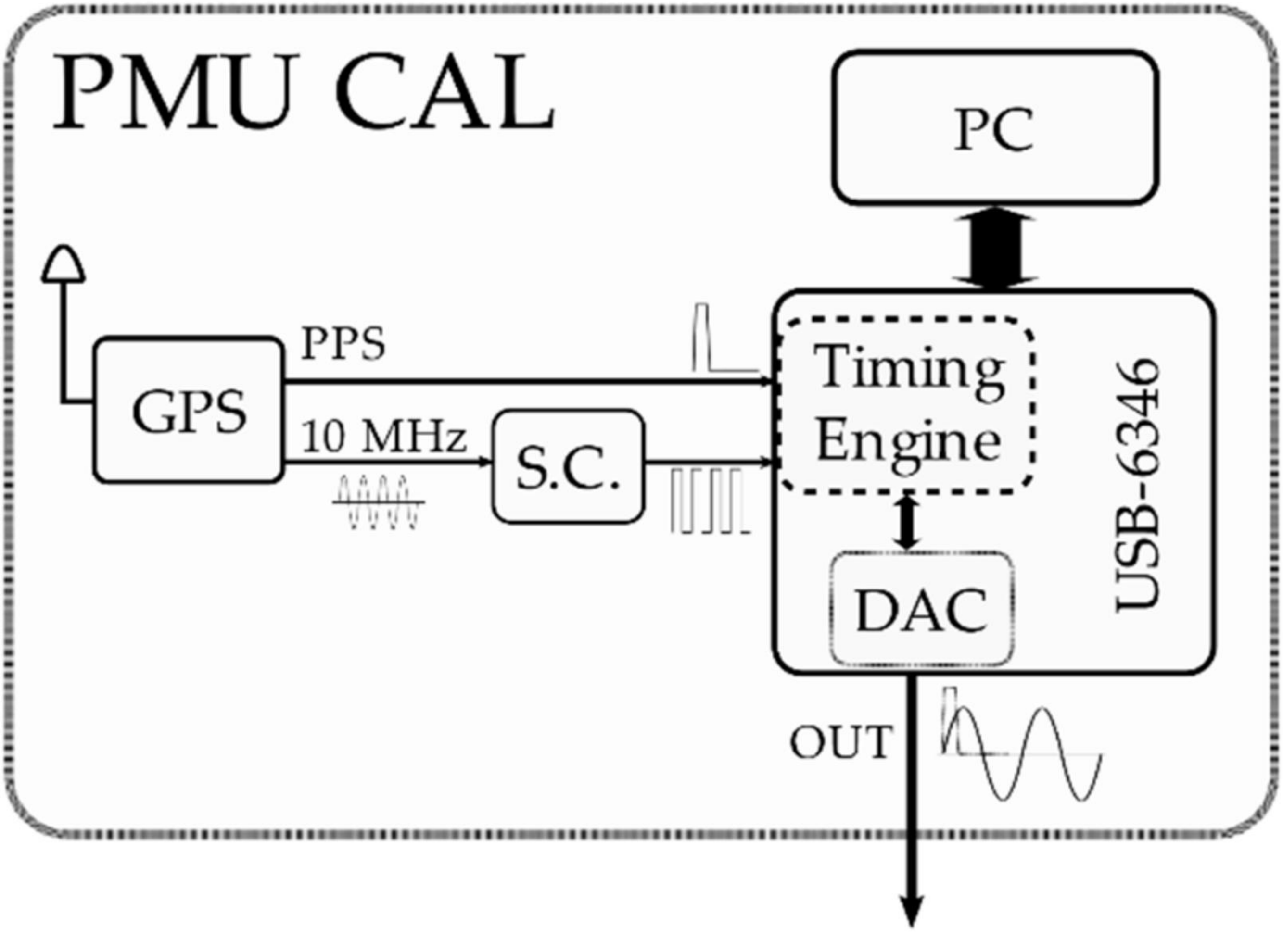
The remainder of this section contains: first, the calibrator concept and hardware are presented in

Section 1.1; second, the designed characterization tests for the calibrator are described in Section 2.2; finally, the characterization results are commented on and arranged to provide the accuracy specifications of the calibrator in Section 2.3.

2.1. The Calibrator Hardware Architecture

The standard prescribes the verification of some parameters that quantify the deviation of the synchrophasor measured by the tested PMU from the reference one. These parameters are the Total Vector Error (TVE), the Frequency Error (FE), and the Rate Of Change Of Frequency (ROCOF) Error (RFE). The uncertainty affecting the reference synchrophasor should be at least one order of magnitude smaller than the one expected from the tested PMU. To conduct this kind of evaluation, a PMU test system (or PMU calibrator) is needed. In recent years, many have faced the problem of PMU calibration. Besides the research on the definition of accurate phasor estimation algorithms , the implementation of reference class hardware test systems is of key importance to the successful deployment of PMUs in Smart Grids. Researchers and national metrological institutes have developed in-house test facilities and employed off-the-shelf solutions . The architecture of PMU calibrators is quite consolidated. Basically, the PMU calibrator generates the test waveforms through an analog output stage and feeds them to the PMU under test; simultaneously, the calibrator returns the generated waveform to produce the reference synchrophasor. Both the generation and the acquisition stages are driven by a timing stage distinguished by stable clocks and triggers referenced to an absolute timing source, such as a GPS clock or an atomic clock. The main reason behind this kind of design, in which a reference PMU is actually implemented, is the fact that it also allows the calibration of other calibrators

For the purpose of the present work, this aspect is not necessary: thus, the calibrator architecture has been kept as simple as possible. In fact, it is a generator capable of producing an accurate waveform from the magnitude and timing point of view, equivalent to the reference synchrophasor. The components are sketched in **Figure 2**: (i) an accurate GPS disciplined oscillator Trimble Thunderbolt E [**63**] providing the pulse-per second (PPS) signal and a disciplined 10 MHz reference clock signal; (ii) an NI USB-6346 multifunction I/O device , which employs as timing and synchronization sources both the PPS and the 10 MHz signals and outputs the test waveforms from the analog output channel; (iii) a PC running the calibrator software and the calibrator characterization test software, both developed in LabVIEW. The characteristics of the analog output channel (OUT) DAC are summarized in **Table 1**, whereas the characteristics of the GPS disciplined oscillator are shown in **Table 2**.



2.3. Results of the Characterization Tests

This section presents the results obtained from each of the test cases described above.

2.3.1. Signal Magnitude Test Results

In **Table 5**, the following quantities are reported: �1 RMS* is the ideal RMS value of the sinusoidal test waveform set on the calibrator user interface corresponding to the test points shown in **Table 3**; ��1 RMS is the average of the RMS values measured by the DMM and ��1 RMS is the standard deviation; ��, �1 RMS and ��, �1 RMS are the uncertainties evaluated according to Type A and Type B methods, respectively, as described in [67]. Lastly, ��1 RMS is a parameter defined as:

(1)

Table 5. Steady-state signal magnitude characterization results.

The main contribution to the uncertainty comes from the DMM a priori evaluation of the measurement uncertainty; in fact, the deviation in the measurement is negligible compared to the former. Moreover, the computed values of ��1 RMS show that the deviation between the RMS value of the sinusoidal test waveform produced by the calibrator and the ideal RMS value set on the calibrator user interface is of the same order of magnitude of the uncertainty affecting the DMM measurement.

2.3.2. Harmonic Distortion Test Results

Given the considerable amount of test points, for readability's sake, only the single harmonic cases which produced the best and the worst results have been reported in **Table 6**. The quantities other than the harmonic order h are the same as the ones in **Table 3**, but this time they refer to the tested single harmonic component.

Table 6. Steady-state single harmonic component signal characterization results.

The order of magnitude of the ��h RMS values for all the others h cases is 10−5. Not surprisingly, considerations analogous to the ones deduced in **Section 2.3.1** can be made. In fact, as in the previous case, the DMM acquires sinusoidal signals whose frequency is contained in a bandwidth in which the DMM maintains almost the same accuracy.

Table 7 is analogous to **Table 6**, but the values for the test signal composed by the power frequency component and the harmonic disturbance are reported.

Table 7. Steady-state standard harmonic disturbance signal characterization results.

Again, the results obtained are in line with the previous ones, confirming the consistency of the operations.

2.3.3. Synchronization Test Results

In **Figure 5**, the PPS signal, the 100 kHz square waveform generated from the analog output (OUT) of the calibrator and the digital square wave reproduced by the digital counter (CTR) are plotted. **Figure 6**, instead, shows the histograms representing the distribution of the delay measurement between the (a) PPS and the CTR rising fronts and (b) the PPS and the OUT rising fronts.

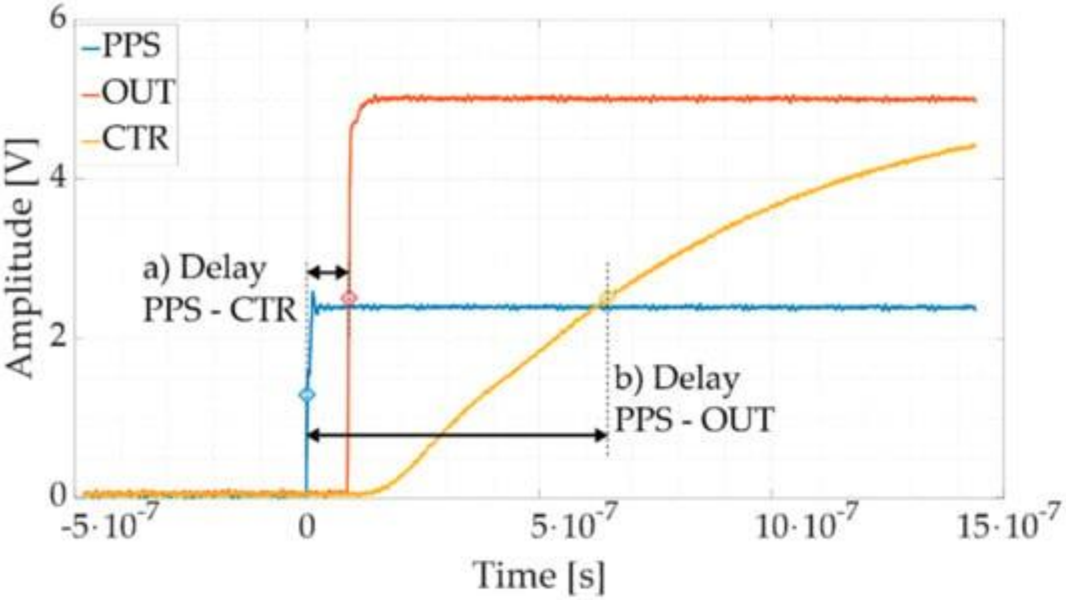


Figure 5. Oscilloscope waveform acquisitions for the board synchronization evaluation. The PPS signal (**blue**), the CTR signal (**red**), the OUT signal (**yellow**).

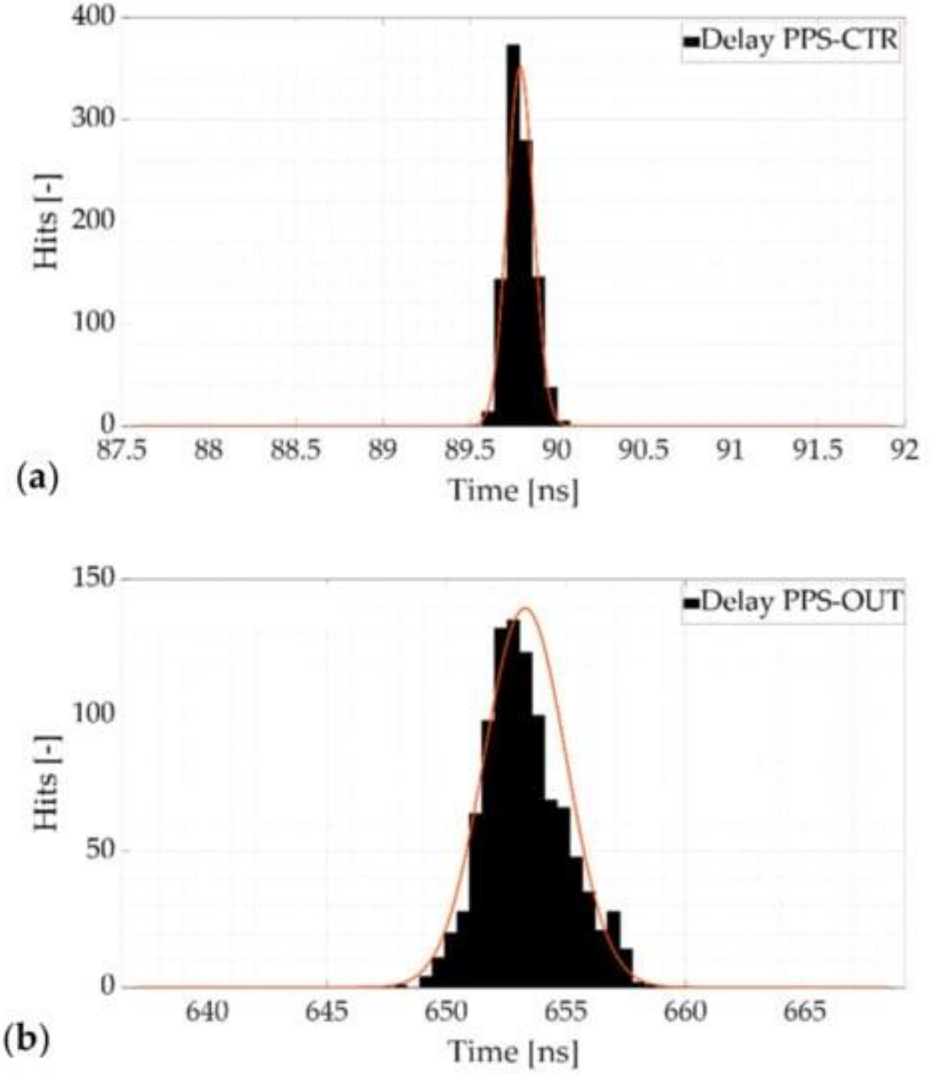


Figure 6. Distribution histograms of the delay measurement between the (**a**) PPS and the CTR rising fronts and (**b**) the PPS and the OUT rising fronts.

Other than the locked phase relation among the three waveforms, it is evident that the board outputs the requested signals in a very responsive way. The CTR signal rises almost a single time-base clock tick (≈90 ns) after the detection of the trigger, with low dispersion (less than 100 ps), denoting an immediate response of the board. The OUT signal response behavior shows a systematic delay contribution of ≈ 650 ns and a standard deviation of less than 2 ns (equivalent to ≈0.6 �rad).

The results for the case of the 50-Hz square waveforms are not reported since they do not differ significantly from the ones already shown.

2.3.6. Characterization Conclusions

After the presentation of the calibrator characterization results in this sub-section, it is possible to summarize them with the goal of drawing an overall picture of the device's performances.

Concerning magnitude, the calibrator has proved to be very accurate and precise in terms of reproducing waveforms with the desired RMS value. In the sinusoidal steady-state case, there are no appreciable deviations between the DMM measurement result and the ideal value set on the calibrator user interface. Moreover, the worst relative uncertainty is 1 ×10−4. The results are also similar in the harmonic test cases. This fact ensures us that the calibrator can provide the virtual PMU with the designed harmonic test waveform.

Analogously, the calibrator performs well also under the frequency point of view. No significant deviation has been observed from the frequency counter measurement results, and the worst relative standard uncertainty is 6 × 10−7. Instead, the ROCOF absolute standard uncertainty is 4 × 10−5 Hz.

Instead, different main contributions shall be considered for the phase accuracy. First, the 15 ns (≈5 �rad) introduced by the GPS disciplined oscillator; second, the 2 ns deviation (≈0.7 �rad) measured between the rising front of the PPS signal and the analog output step waveform; third, the 0.4 �rad deviation measured with the DAQ; finally, since errors on frequency directly affect the phase, it is possible to also add a contribution which translates the worst frequency uncertainty in an angle, resulting in a ≈6 �rad term. All the conversions from time and frequency to angles have been carried out considering the most precautionary scenario: for example, a 1 ns variation at 55 Hz corresponds to a bigger phase angle rather than the one at 45 Hz, whereas a 10 �Hz variation corresponds to a bigger phase angle variation at 45 Hz rather than at 55 Hz. Combining all these components as the root of the sum of the squares and considering a 3� interval, the phase uncertainty is equivalent to ≈2 × 10−5 rad.

Let us take under examination the equation below shown in [47]:

$$\mathrm{TVE}=21+\mathrm{ME}1-\cos \mathrm{PE}+\mathrm{ME}2$$

(6)

where ME is the synchrophasor magnitude error and PE is the synchrophasor phase error. Given the conclusions of the analysis presented above, if ME = 1 × 10−4 and PE = 2 × 10−5 rad, then the equivalent TVE of the calibrator is approximately ≈0.01 %. This result is compliant with what is recommended in [47] for PMU test systems.

3. RTS Environment

This section aims at briefly presenting the RTS environment selected for being tested. In particular, the main features of the OPAL simulator are summarized in **Section 3.1**, while in **Section 3.2**, the virtual PMU to be tested is presented and commented on.

3.1. Description of the RTS

The RTS adopted in this work is the OPAL-RT OP 4510 Simulator, which allows running real-time simulations and HIL applications via RT-LAB software. Its main components are depicted in **Figure 7**, in which each color has a specific meaning. Green is used for the internal components and yellow for the interfaces and connections.

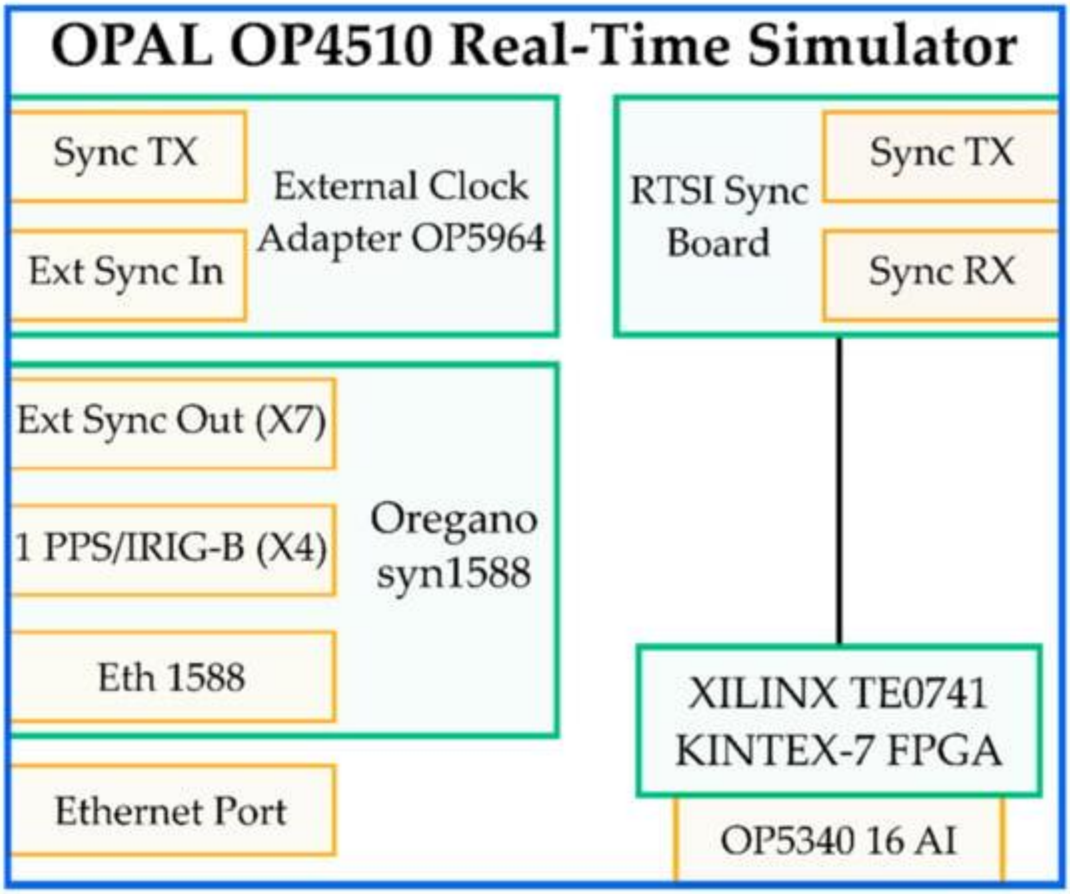


Figure 7. Main components of the OPAL OP4510 RTS.

A brief description of the main components is given in what follows:

- Oregano syn1588 PCIe NIC. It is a PCI Express Ethernet network interface that provides highly accurate clock synchronization via the IEEE 1588 Standard (accuracy of its oscillator higher than 0.05 ppm). The Oregano card can be synchronized with either a PPS signal or an IRIG-B signal from a GPS source (3.3 V signal).
- External Clock Adapter OP5964. It is used to receive and transmit the synchronization signal from the outside to the interfaces.
- RTSI (Real-Time System Interface) Synchronization Board. This board directly communicated with the FPGA, as can be seen from Figure 7 (black line).
- XILINX TE0741 KINTEX-7 FPGA. It accepts either OPAL-RT boards or RS422 signals. The types of synchronization allowed are LVDS and fiber optic.
- Analog input (AI) card OP5340. It features 16 synchronous differential analog input channels with a maximum voltage range of ±20 V, sampled at 400 kSa/s. The analog to digital converter (ADC) has a 16-bit resolution, and the minimum acquisition time is 2.5 µs per channel. The declared maximum noise of the analog card is 20 mV peak-to-peak. The ADC already includes anti-aliasing filters to remove frequencies higher than 600 kHz.
- Ethernet port. It is used to interface a laptop to the OP4510 RTS.

3.2. Description of the PMU

A PMU is a digital device that provides synchronized voltage and current phasor measurements referred to as synchrophasors. At the installation bus, instrument transformers, such as CTs and VTs, are needed to measure the three-phase quantities. Their analog signals are converted into digital by means of an ADC with a sampling rate usually varying from 12 to 128 samples per cycle of the rated power frequency.

In a PMU, the sampling clock could be phase-locked with a single time reference, which is given by the GPS pulse per second (PPS). The clock provided by the GPS system is also referred to as Universal Time Coordinated (UTC), and it is used as a time reference to time-tag the outputs.

At this stage, the phasor must be retrieved, and the easiest method to perform this action consists of using the Discrete Fourier Transform (DFT), which allows obtaining the magnitude and phase of the signal. Nevertheless, it must be highlighted that the relevant international standards, such as IEC 60255 , do not provide a specific phasor algorithm to be implemented in PMUs. Likewise, the window length, the sampling rate, the phasor estimates reporting rate, the communication protocol, as well as the measurement accuracy are all distinctive to each PMU device. Therefore, alternatives to the DFT technique have been investigated and reported in the scientific literature.

Two main algorithm categories can be distinguished: DFT-based and non-DFT-based algorithms. Examining the first category, the classic DFT-based methods work well when the system frequency is close to the nominal frequency but, due to spectral leakage, significant errors arise when the frequency drifts from its rated value. For this reason, much effort has been made in order to improve the accuracy of DFT-based estimation algorithms under off-nominal frequency conditions. Among these, it is worth mentioning interpolated-DFT approaches (IpDFT) and dynamic DFT ones .

On the other hand, the majority of non-DFT-based algorithms are based on Kalman filters (KFs). It is worth remarking on Taylor-Kalman Filters (TKFs) [71] and Adaptive and Extended Kalman Filters (AKFs, EKFs) . A small number of other approaches are based on different techniques such as Taylor Weighted Least-Squares (TWLS) , Space Vectors (SVs) [75], and wavelets [76]. Given the broadness of the topic enveloping several techniques, comparative studies between PMU estimation algorithms have also been presented in the literature.

Assessments between DFT and KF-based algorithms are typically performed, as in . The analyses are based on simulations in accordance with to evaluate and compare the performance of the estimators. It is shown that KFs are optimal for harmonic rejection and for tracking large-frequency deviations occurring in power systems, contrary to DFT-based ones, which suffer from leakage issues as aforementioned . On the contrary, DFT approaches do not suffer from instabilities, in contrast to KFs, and they are generally simpler than the latter, resulting in a significantly reduced computational burden.

For this reason, in this work, a simple DFT algorithm is chosen as the estimator in the PMU implemented in real-time. The selected method consists of applying the algorithm to a single-phase signal. Given that the objective of this research involves the characterization of a virtual PMU, then the use of a strictly three-phase algorithm requiring more resources is discarded—for instance, Clarke transformation-based and positive sequence estimation algorithms [80]. Moreover, this choice means that the algorithm could be easily duplicated for a real-case three-phase signal. In addition, as above-mentioned, this choice implies a limited computational burden aiming at having the least impact on the cores of the CPU of the RTS, which would be able to perform the phasor evaluation within fixed time steps.

Even though parallelization could be possible, our choice of a single-phase DFT algorithm is also based on the objective of this paper. Indeed, the proposed work aims at highlighting the importance of the characterization of virtual PMUs for HIL applications; hence, the achievement of the best estimating algorithm is out of the scope of this research.

Finally, according to PMUs can be classified into two classes of performances: P-class (protection applications requiring a fast response) and M-class (measurement applications requiring high accuracy). The latter is considered the type of virtual device implemented in the RTS, considering that it also includes the performance limits of the P-class.

Tests that will be described in the next **Section 4.1** are based on [**47**], which specifies both steady-state and the dynamic performance compliance criteria for each class of PMUs.

4. Tests and Results

This section contains the core of the work. As previously mentioned, a virtual PMU is tested to highlight its significant contribution to the overall uncertainty of a system. The set of tests designed for assessing the performance of a virtual PMU considering the RTS contribution is fully described in accordance with the standard IEC 60255 . The goal of the tests is an increased awareness by more RTSs users in performing preliminary characterization of their virtual models.

4.1. PMU Testing

The tests performed on the virtual PMU hosted inside the RTS aim to assess the amplitude, the phase, the frequency, and the distorted signals. For each aspect, a specific test has been performed. For all tests, the time step of the simulator is 50 �s, and the sampling frequency of the analog input DAQ is 20 kSa/s. A picture of the test setup is shown in **Figure 8**. To better clarify the testing idea, the virtual PMU inside the RTS is tested according to the PMU standard [**47**]. This is to treat the virtual PMU like a physical one when its testing is concerned. Therefore, amplitude, phase, harmonic, and frequency tests are described in what follows.

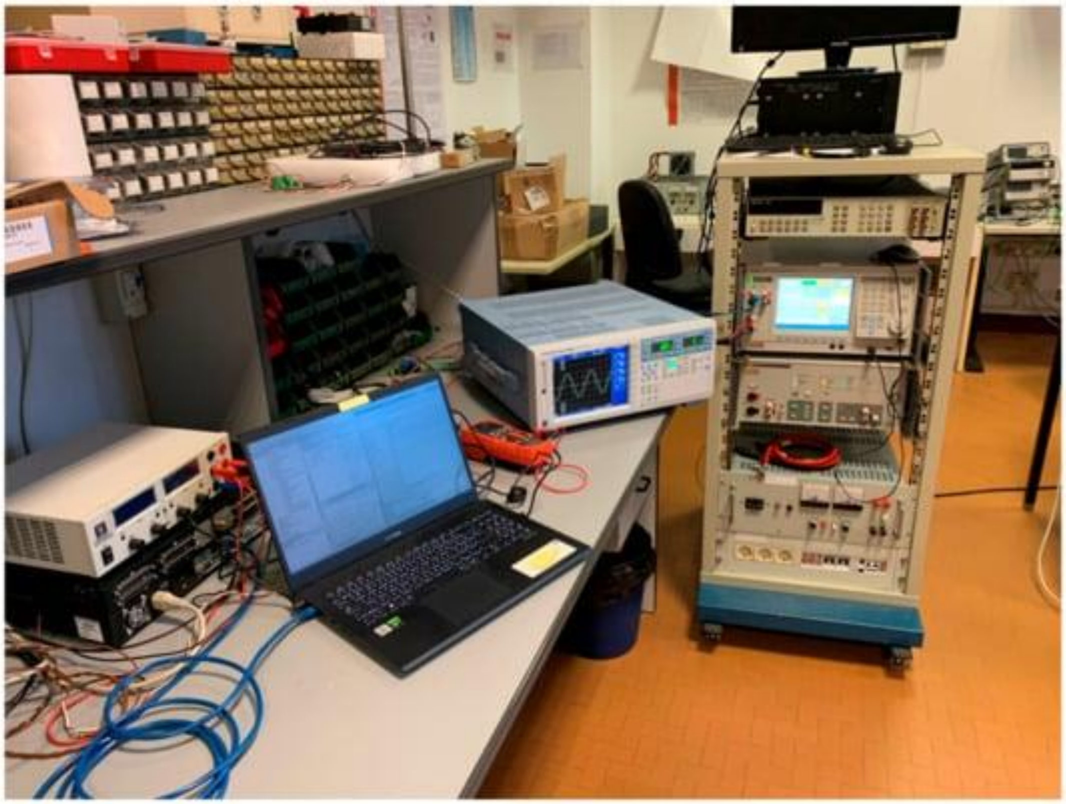


Figure 8. Picture taken of the laboratory environment during the RTS testing.

4.1.1. Amplitude Tests

The amplitude tests have been designed considering both the current and voltage limits defined in [**47**]. In fact, the current variation is wider (from 10% to 200%); therefore, the RTS DAQ max input (10 V) has been associated with 200% of the rated signal. Consequently, five tests, referred to as A1 to A5, from 10% to 200% of the rated signal, are designed and described in Table 11. The table contains the phase, the frequency, and the amplitude (peak and RMS) of the adopted signals.

Table 11. Settings of the amplitude tests.

4.1.2. Frequency Tests

In accordance with [47], nine different tests, referred to as F1 to F9, have been designed for testing the virtual PMU behavior vs. frequency. They are collected in Table 12. Adopting 100% of the rated signal, nine frequency values from 48 Hz to 52 Hz, with steps of 0.5 Hz, have been used. For the frequency tests, the initial phase of the signals is always set to zero.

Table 12. Settings of the frequency tests.

4.1.3. Harmonic Tests

The aim of these tests is to verify the performance of a PMU when a single harmonic component is superimposed to the main frequency signal. In detail, the harmonic component has an amplitude corresponding to the 10% of the main signal.

The defined tests are listed in Table 13. For each test, referred to as H1 to H16, the table contains the amplitude of both the main signal and the harmonic component. The tested harmonic components range from the third to the forty-ninth odd harmonics.

Table 13. Settings of the harmonic tests.

4.1.4. Phase Tests

Typical laboratory tests use 0 as the initial phase. However, considering real applications, a set of tests tackling the performance of the virtual PMUs when the phase is not null is necessary. Therefore, Table 14 presents 11 tests, referred to as P1 to P11, in which the phase varies from 0° to 100° with steps of 10°. The table also contains the amplitude of the signal (always 100% of the rated) and the frequency (50 Hz).

Table 14. Settings of the phase tests.

4.2. Tests Results

Each test described in **Section 4.1** had a duration of 20 s, during which the quantities have been collected and then averaged to obtain the final results. The average values and their standard deviation of the mean are collected in **Table 15**, **Table 16**, **Table 17** and **Table 18** for the amplitude, frequency, harmonic, and phase, respectively. Every table contains the RMS value of the measured voltage RMS, its standard deviation of the mean the phase, its standard deviation of the mean h, the measured frequency, its standard deviation of the mean the ROCOF, and its standard deviation of the mean .

Table 15. Measurement results of the amplitude tests.

Table 16. Measurement results of the frequency tests.

Table 17. Measurement results of the harmonic tests.

Table 18. Measurement results of the phase tests.

From the tables, it can be observed that the results are quite coherent, and, in particular, the standard deviation of the mean (the absolute one) is always in the order of 10^{-8}, 10^{-7}, 10^{-7}, and 10^{-5} for the amplitude, phase, frequency, and ROCOF, respectively.

For the sake of readability of the results, **Figure 9** and **Figure 10** show the RMS vs. frequency and the RMS for each harmonic test taken from **Table 16** and **Table 17**, respectively.

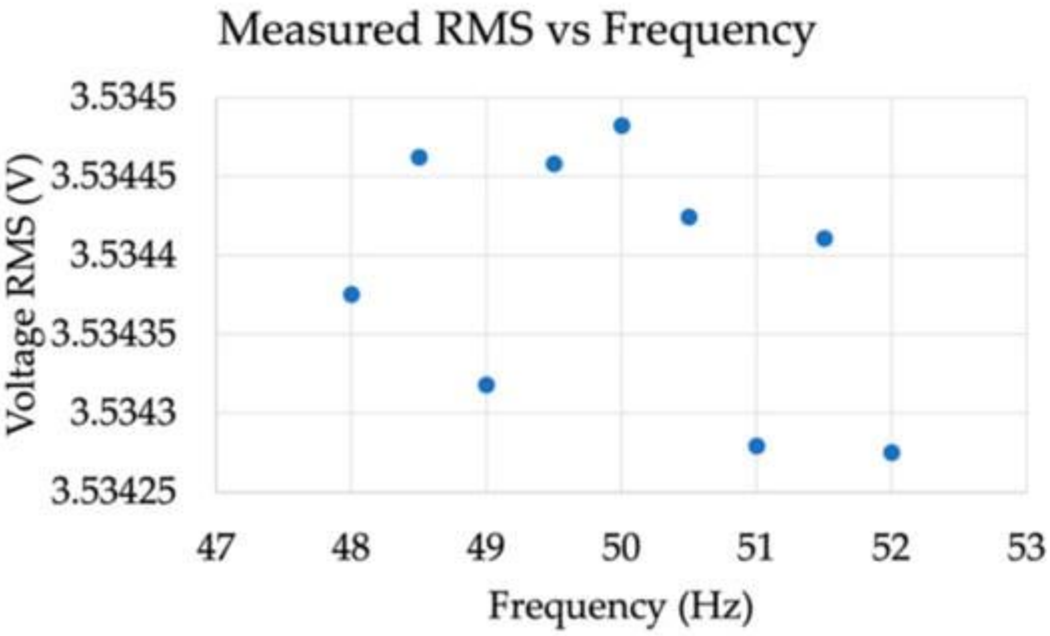


Figure 9. RMS vs. frequency of results in Table 16.

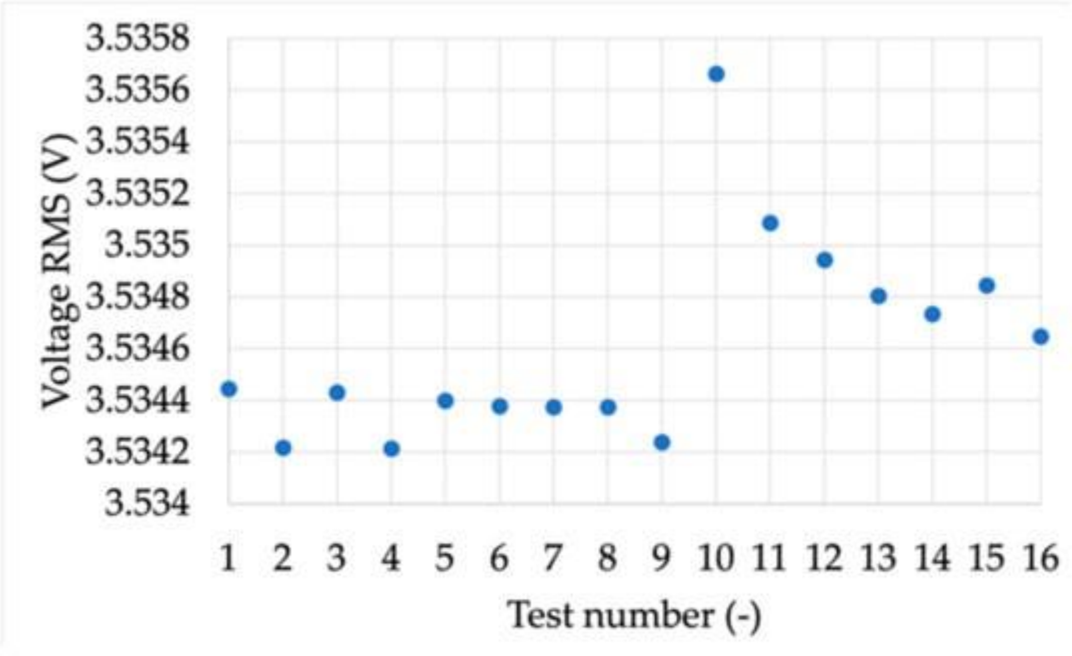


Figure 10. RMS for each harmonic test (Table 17).

Despite the good preliminary results, it is necessary to evaluate them according to the indices defined in [47]. Furthermore, the propagation of the uncertainty process is fundamental to quantify and assess the accuracy of the presented results.

To this purpose, the results presented in Table 15, Table 16, Table 17 and Table 18, along with the reference values provided and set by the reference calibrator, are used to compute the TVE, FE, and RFE for each of the performed tests.

The obtained indices are collected in Table 19, Table 20, Table 21 and Table 22 for the amplitude, frequency, harmonics, and phase, respectively. The tables are coherent among each other, and they contain: TVE, FE, RFE, and their combined uncertainties and respectively.

Table 19. Indices of the amplitude tests.

Table 20. Measurement results of the frequency tests.

Table 21. Measurement results of the harmonic tests.

Table 22. Measurement results of the phase tests.

Note that the combined uncertainties have been computed considering (i) the standard deviation of the measured quantities (Table 15, Table 16, Table 17 and Table 18), (ii) the standard deviation of the computed indices, and (iii) the formula used to compute the index. Concerning the last aspect, the uncertainty propagation is straightforward in the case of FE and RFE. In fact, the mathematical operation is subtraction. On the contrary, the TVE expression is quite complicated compared to the previous two; therefore, the Monte Carlo method has been used to obtain the uncertainty associated with TVE (100,000 trials).

Starting from Table 19, different comments arise. In the table, but also for the flowing ones, when the mean value is lower than the obtained combined uncertainty, the choice has been of leaving the full number and not putting zero. Such a choice is supported by the aim of showing the discrepancies between the order of magnitude of the quantity and its combined uncertainty. Therefore, the significant digits are coherent only in the case of combined uncertainty lower than the mean value.

Another comment involves the indices. While FE and RFE are always below the limits defined by [47] (even if a test involving the amplitude variation is not defined), the TVE exceeds the limits in test A5. Such a test is the one with the smallest input test signal (0.1 V). It is important then to understand the capabilities of the RTS system before using them for simulating purposes. As in the case of the tests, it is demonstrated from the results of Table 19 that the characterization of the virtual PMU in terms of amplitude is fundamental to know and correct the values during normal operations.

For the results in Table 20, what is stated for RFE and FE in the case of Table 19 still applies. However, the TVE does not always remain within the limits of the standard. In particular, it exceeds 1% in test F2 and is 0.921 in test F3. On average, all frequency test results are not really satisfactory, and the reason can be attributed to the acquisition process of the RTS.

Table 21 lists the indices computed in the case of the harmonic tests. From the results, it emerges that FE and RFE are far below the defined limits. As for TVE, it is below the 1% limits in all tests, but on average is always higher than 0.6%. For a better overview of the TVE results, considering the number of digits involved, Figure 11 has been used.

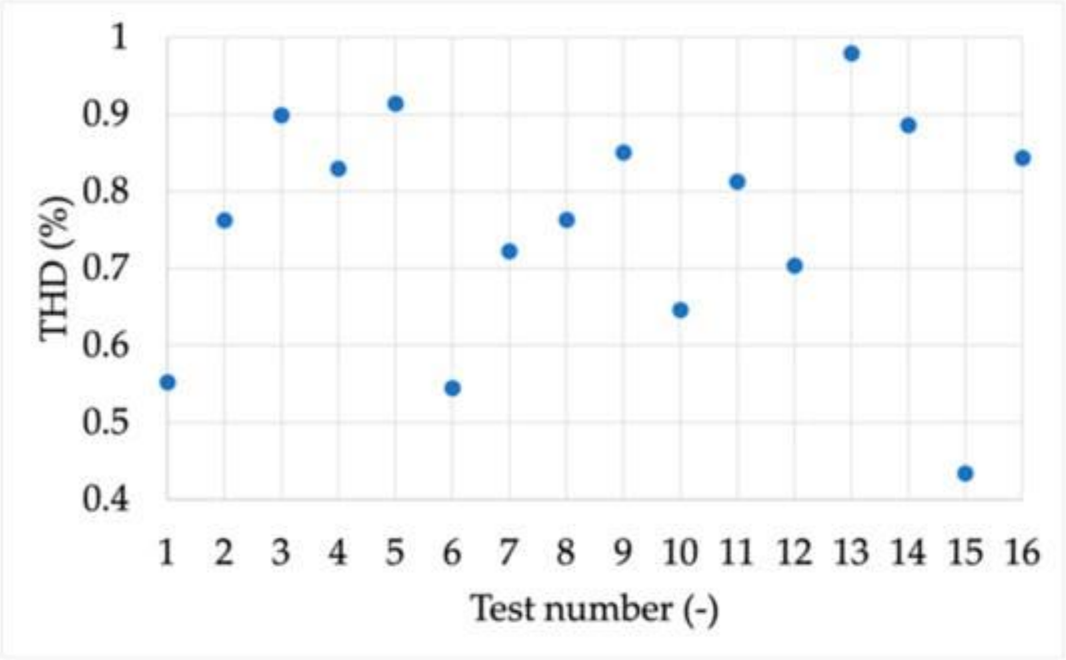


Figure 11. THD obtained in each harmonic test from H1 to H16.

Identical comments can be made for the results in Table 22, which contain the indices computed for the phase tests.

In light of all results presented in the previous tables, it can be concluded that the characterization of a virtual PMU inside an RTS environment is fundamental for accuracy purposes (see Figure 12 for a simplified block diagram of the research). In fact, what was obtained clearly emphasizes the need for a priori knowledge of the performance of each component to be used within a measurement setup. This is to avoid unnecessary propagation of uncertainties from one component to another.

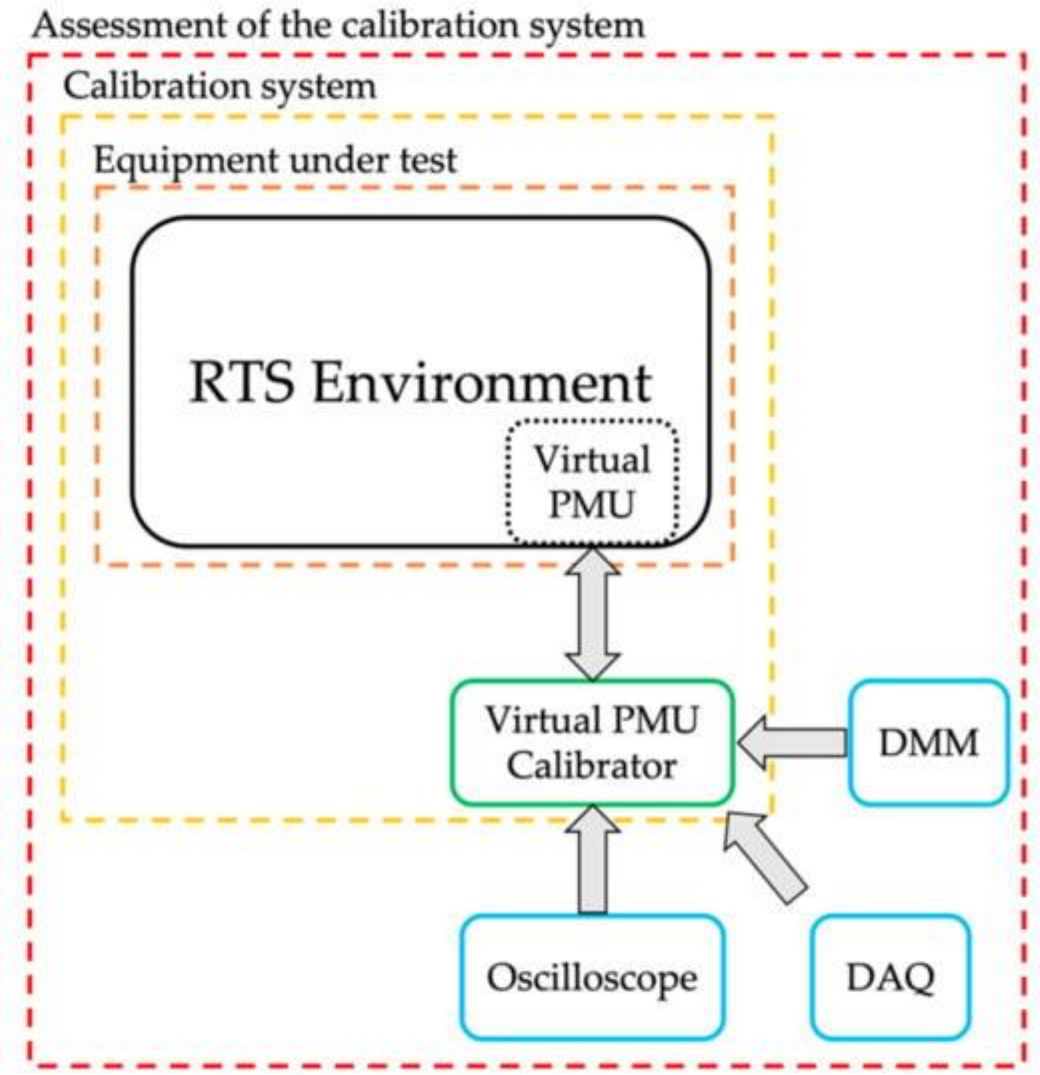
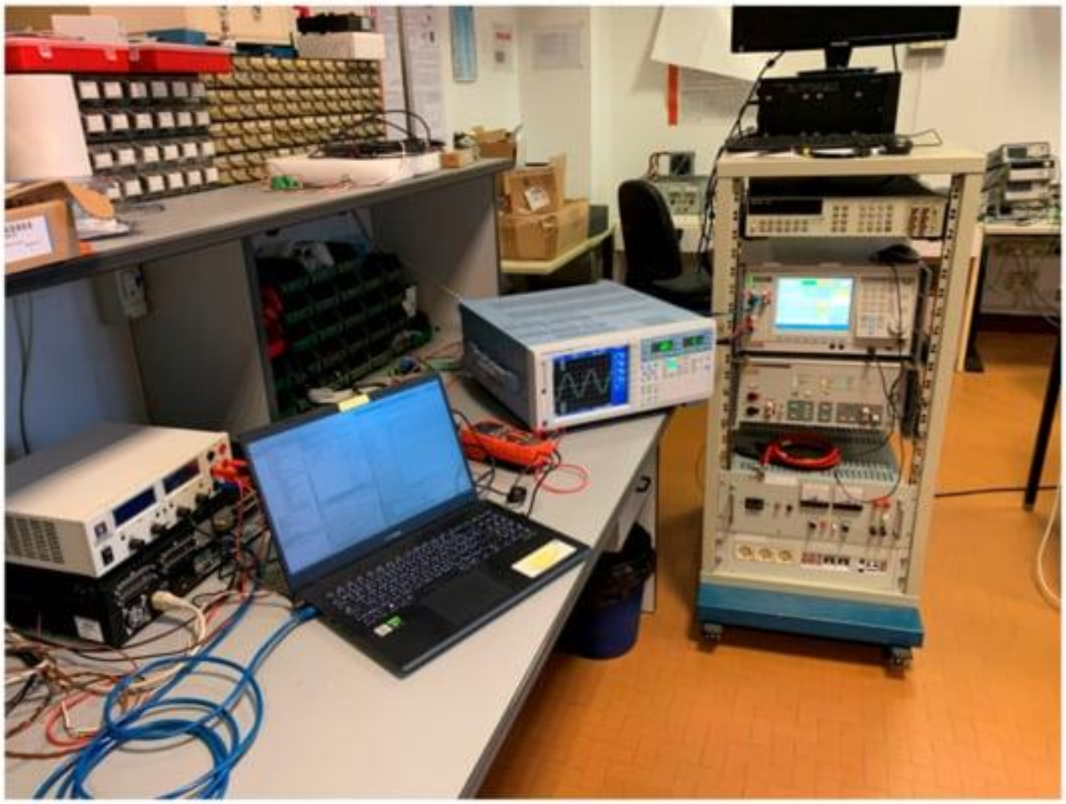


Figure 12. Block diagram of the complete research.

In the specific case considered, the single contribution of the RTS is not negligible, and in some cases, the limits defined by the standard [47] are not satisfied. In addition, the results must be assessed considering that the source of the test signals was generated by an accurate calibrator (see **Section 2.3.6**). Consequently, if real sensors with typical accuracy ranging from accuracy class 0.5–1 are considered, the overall uncertainty propagated in the final results would significantly increase.

5. Conclusions

The use of real-time simulators among researchers and utilities is increasing day after day. This allows enhancing the simulation capabilities, including the possibility to recreate complete digital models of the power network. However, the RTSs must ensure a high level of accuracy for their results to be reliable enough for the final users. This aspect is not always considered and sufficiently treated. The article has the aim of emphasizing and supporting, with rigorous experimental activity, the lack of methodology found in the literature. A virtual PMU is then characterized by testing it like a physical PMU, hence by using the same tests defined in the dedicated standard. Furthermore, to complete the discussion, the complete characterization process of a PMU calibrator is described. The main result of the virtual PMU characterization is not the index remaining within or exceeding the limits. The main result is the experimental proof that the preliminary characterization of the virtual PMU is mandatory when an RTS environment must be used for simulating power networks. Such a result is confirmed by all the performed tests, and its importance can be extended to all activities involving an RTS, not necessarily correlated to power networks or electrical engineering in general.



Expt. No: 11

SIMULINK MODEL FOR TWO AREA LOAD FREQUENCY CONTROL

AIM: To find dynamic response of the given two - area load frequency control problem theoretically and to plot and verify the results in SIMULINK

PROBLEM:

The parameters for load frequency control of a two area are:
Speed governor gain Ksg=1
Time constant of speed governor Tsg=0.4
Speed regulation of speed governor R=3
Gain of turbine Kt=1
Time constant of turbine Tt=0.5
Gain of power system Kps=100
Time constant of power system Tps=10
Proportional plus integral gain Ki=0.07
Synchronizing co-efficient Tr=0.05
Frequency bias 0.425s
Develop a SIMULINK model for two area load frequency control with PI controller and obtain the frequency deviations in both areas and tie-line power deviations for a load change of 1pu in Area-2

THEORY:

SIMULINK MODEL:

1/3
1
Transfer Fcn9
0.425
1
Transfer Fcn8
1
0.4s+1
Transfer Fcn7
1
0.5s+1
Transfer Fcn6
100
10s+1
Transfer Fcn5
100
10s+1
Transfer Fcn4
1
0.5s+1
Transfer Fcn3
1
0.4s+1
Transfer Fcn2
0.05

s
Transfer Fcn12
0.425
1
Transfer Fcn11
1/3
1
Transfer Fcn10
-0.07
s
Transfer Fcn1
-0.07
s
Transfer Fcn
Step Step2
Scope2
Scope1
Scope
-1
Gain

EXPECTED OUTPUT:

RESULT:

Expt. No: 12

MATLAB Program to Find Optimum Loading Of Generators with Penalty Factors

AIM: To find optimum loading of two units for the given load with penalty factors and verify using MATLAB.

PROBLEM:

A two-bus system is shown in figure. If 100 MW is transmitted from plant 1 to the load, a transmission loss of 10 MW is incurred. Find the required generation for each plant and the power received by load when the system λ is Rs 25/MWh. The incremental fuel costs of the two plants are given below:

1
1
$_G dP$
dC = 0.02 P_{G1} + 16.0 Rs/MWh
2
2
$_G dP$
dC = 0.04 P_{G2} + 20.0 Rs/MWh

Solve the problem theoretically. Use the data in the following MATLAB program

MATLAB PROGRAM:

```
% this program finds the optimal loading of generators including penalty factors
% Pd stands for load demand, alpha and beta arrays denote alpha beta coefficients
%for given generators, and n is the no of generators
clc
clear
n=2;Pd=237.04;
alpha=[0.020 0.04];
beta=[16 20];
% initial guess for lamda is 20;tolerance is eps and increment in lamda is deltalamda
lamda = 20; lamdaprev = lamda ; eps = 1; deltalamda = 0.25;
% the min. and max. limits of each generating unit are stored in arrays Pgmin and
Pgmax
Pgmax=[200 200];Pgmin=[0 0];
B = [0.0010 0
0 0];
noofiter=0;PL=0;Pg = zeros(n,1);
while abs(sum(Pg)-Pd-PL)>eps
for i=1:n,
sigma=B(i,:)*Pg-B(i,i)*Pg(i);
Pg(i)=(1-beta(i)/(lamda-(2*sigma)))/(alpha(i)/lamda+2*B(i,i));
%PL=Pg'*B*Pg;
if Pg(i)>Pgmax (i)
Pg(i)=Pgmax (i);
end
if Pg(i)<Pgmin(i)
Pg(i)=Pgmin(i);
end
end
PL = Pg'*B*Pg;
if (sum(Pg)-Pd-PL)<0
lamdaprev=lamda;
```

```
lamda=lamda+deltalamda;
else
lamdaprev=lamda;
lamda=lamda-deltalamda;
end
noofiter=noofiter + 1;
Pg;
end
disp ('The no of iterations required are')
noofiter
disp ('The final value of lamda is')
lamdaprev
disp ('The optimal loading of generators including penalty factors is')
Pg
disp('The losses are')
PL
```

EXPECTED OUTPUT:

```
The no of iterations required are
noofiter = 21
The final value of lamda is
lamdaprev = 25
The optimal loading of generators including penalty factors is
Pg =
128.5714
125.0000
The losses are
PL =
16.5306
```

RESULT:

EXPERIMENT –13

SINGLE PHASE A.C. VOLTAGE CONTROLLER

AIM:

To study the single phase AC voltage controller with R and RL Load

APPARATUS:

S. No	Equipment	Range	Type	Quantity
1	Single phase AC voltage controller power circuit and firing circuit			
2	CRO with deferential module			
3	Patch chords and probes			
4	Isolation Transformer			
5	Variable Rheostat			
6	Inductor			
7	AC Voltmeter			
8	AC Ammeter			

CIRCUIT DIAGRAM:

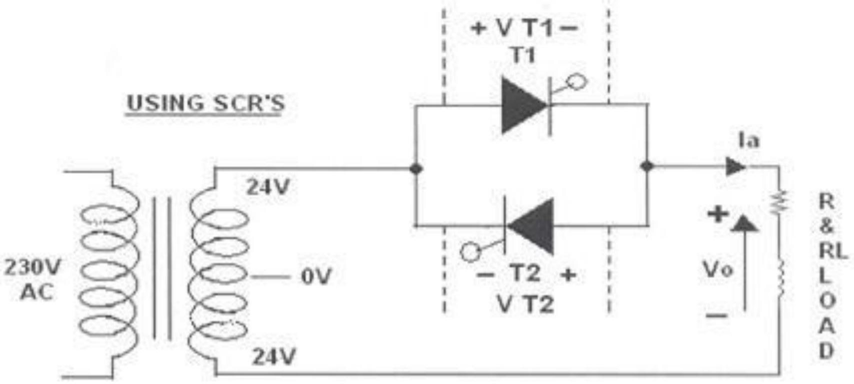


Fig - 8.1 Single Phase AC Voltage Controller with Thyristors

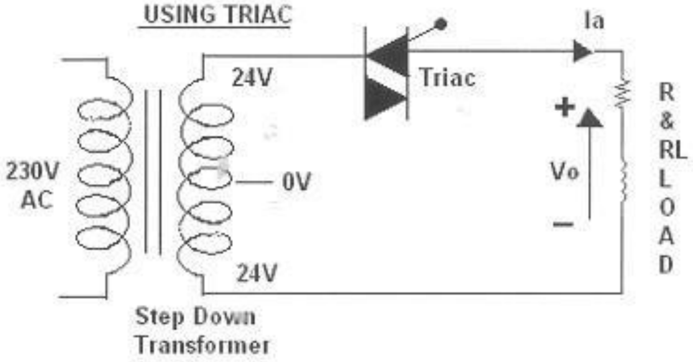


Fig - 8.2 Single Phase AC Voltage Controller with Traic

PROCEDURE:

AC VOLTAGE CONTROLLER WITH TWO THYRISTORS:

1. Make all connections as per the circuit diagram.
2. Connect firstly 30V AC supply from Isolation Transformer to circuit.
3. Connect firing pulses from firing circuit to Thyristors as indication in circuit.
4. Connect resistive load 200Ω / 5A to load terminals and switch ON the MCB and IRS switch and trigger output ON switch.
5. Observe waveforms in CRO, across load by varying firing angle gradually up to 180^0.
6. Measure output voltage and current by connecting AC voltmeter & Ammeter.
7. Tabulate all readings for various firing angles.
8. For RL Load connect a large inductance load in series with Resistance and observe all waveforms and readings as same as above.
9. Observe the various waveforms at different points in circuit by varying the Resistive Load and Inductive Load.
10. Calculate the output voltage and current by theoretically and compare with it practically obtainedvalues.

A.C. VOLTAGE CONTROLLER WITH TRIAC:

1. Make all connections as per the circuit diagram.
2. Connect firstly 30V AC supply from Isolation Transformer to circuit.
3. Connect firing pulse from firing circuit to TRIAC as indication in circuit.
4. Connect resistive load 200Ω / 5A to load terminals and switch ON the MCB and IRS switch and trigger output ON switch.

6. Observe waveforms in CRO, across load by varying firing angle gradually up to 180^0.
7. Measure output voltage and current by connecting AC voltmeter & Ammeter.
8. Tabulate all readings for various firing angles.
9. For RL Load connect a large inductance load in series with Resistance and observe all waveforms and readings as same as above.
10. Observe the various waveforms at different points in circuit by varying the Resistive Load and Inductive Load.

Calculate the output voltage and current by theoretically and compare with it practically obtained values.

TABULAR COLUMN:

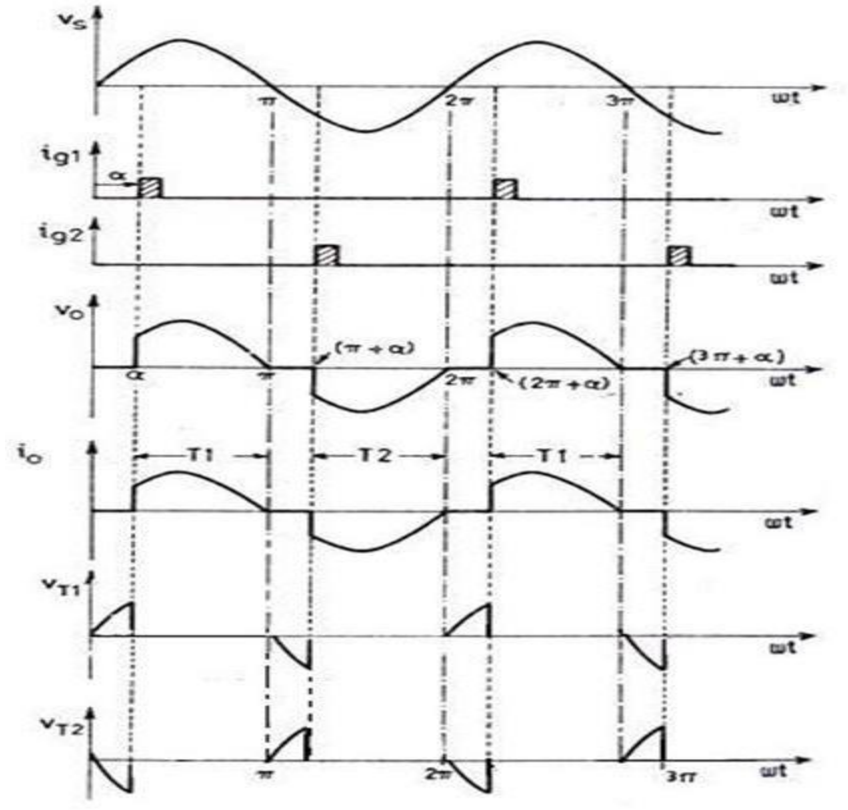
S. No.	Input Voltage (V_{in})	Firing angle in Degrees	Output voltage (V_{0r})		Output Current (I_{0r})	
			Theoretical	**Practical**	**Theoretical**	**Practical**
1						
2						
3						
4						
5						
6						

MODULE CALCULATIONS:

$I_{0r} = V_{0r} / R$

α = Firing Angle

MODEL GRAPH:



Single Phase AC Voltage controller with R-Load

VIVA QUESTIONS:

What type of commutation is used in this circuit?

What are the effects of load inductance on the performance of AC voltage controllers?

What is extinction angle?

What are the disadvantages of unidirectional controllers?

What are the advantages of ON-OFF control?

EXPERIMENT – 14

SIMULATION OF THREE PHASE FULL CONVERTER AND PWM INVERTER

AIM:

Simulation of three phase full converter and PWM inverter with R and RL loads by using MATLAB.

APPARATUS:

S. No	Equipment	Quantity
1	Desktop With MATLAB	1

THEORY:

Three phase full converter:

Three phase bridge controlled rectifier consist of upper group (T_1, T_3, T_5) and lower group (T_2,T_4,T_5) of thyristors. Thyristor T1 is forward biased ad can be triggered for conduction only when V_A is greater than both V_B and V_C. From figure this condition occurs at $w_t=30^0$. Hence T_1 can be triggered only at $w_t=30^0$.If firing angle is α, then T_1 starts conduction at $w_t=30^0 + \alpha$ and conducts for 120^0 where it get commutated by turning on of next thyristor ie,T_3.Similarly triggering instant for T_3 and T_5 are determined when considering V_B and V_C respectively. For lower group T_4,T_6 and T_2, negative voltages,-V_A,-V_B and -V_C respectively are considered.

Average Value of output voltage is given by

$$V_{avg} = \frac{3\sqrt{3}}{\Pi} V_m \text{Cos } \alpha$$

Three Phase PWM Inverter:

Three phase inverter consists of on and off controlled switches such as MOSFET or IGBT. Sine PWM pulses are used to gate the switches. Upper switches are gated with signals obtained by comparing three reference sine waves each are phase shifted with 1200 with a high frequency triangular carrier wave. Thus, switches are ON when amplitude of corresponding reference sine wave is greater than amplitude of triangular carrier wave. Lower switches are gated with a gate signal which is complement of upper switches of same leg.

Rms Value of phase to neutral output voltage is given by

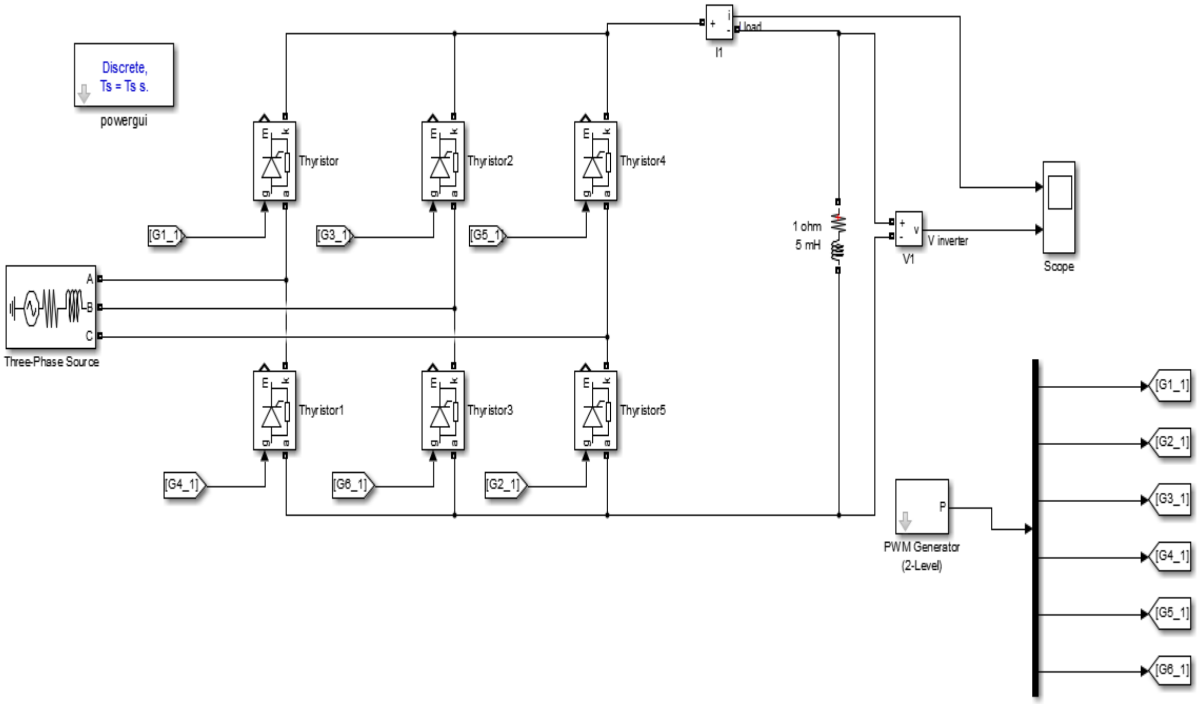
$$V_{Ph\,rms} = \frac{mV_{dc}}{2\sqrt{2}}$$

Rms Value of line to line output voltage is given by

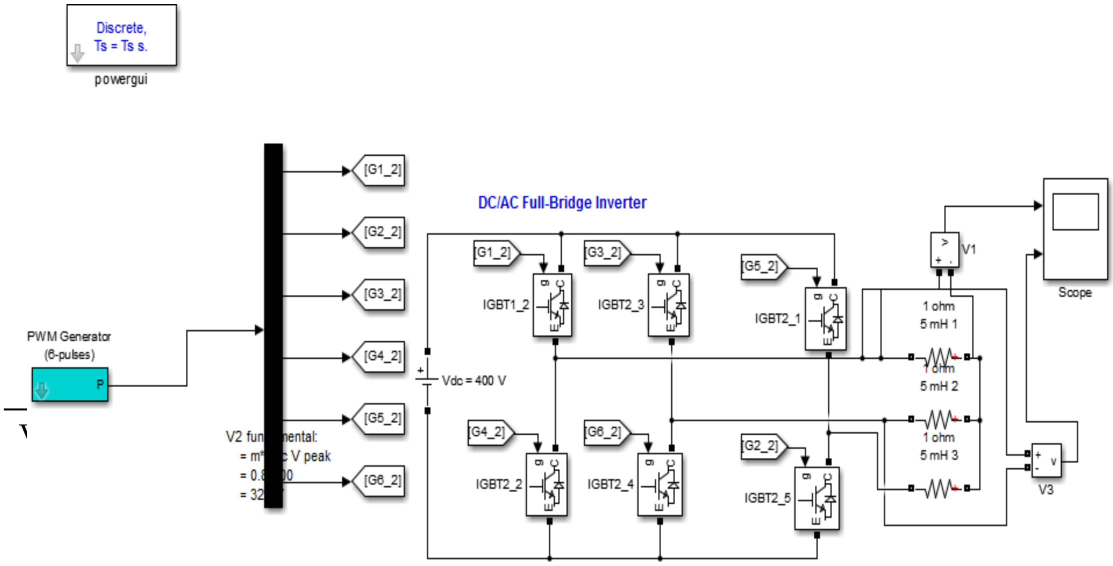
$$V_{Ph\,rms} = \frac{\sqrt{3}mV_{dc}}{2\sqrt{2}}$$

CIRCUIT DIAGRAM:

Three phase full converter:



Circuit diagram for three phase full converterThree Phase PWM Inverter:



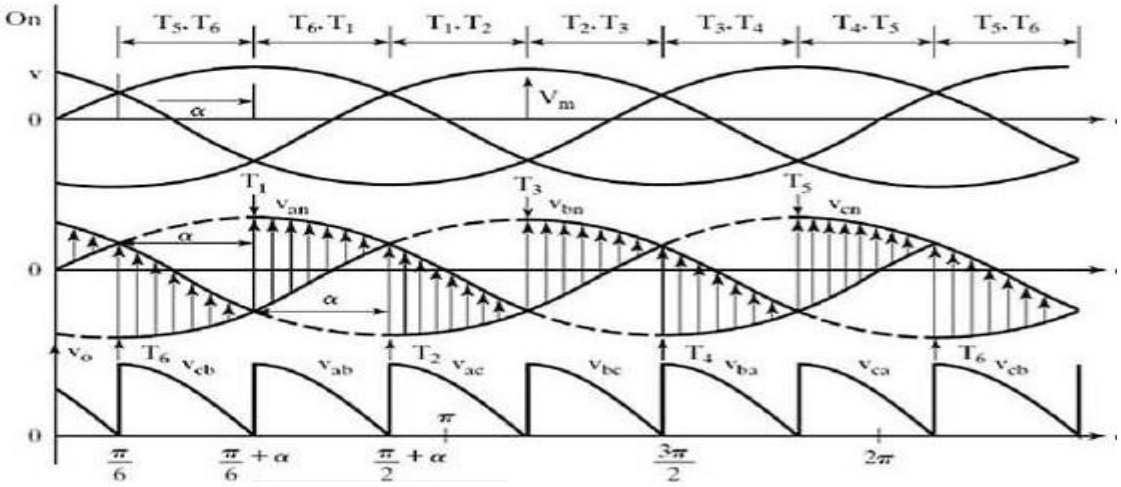
circuit diagram for Three phase PWM Inverter

PROCEDURE:

1. Make the connections as shown in the figures 13.1 and 13.2 by using MATLAB Simulink.
2. Set the parameters in PWM generator for firing the switches, set the values for load and inputvoltage.
3. Check the scope wave forms in each circuit.

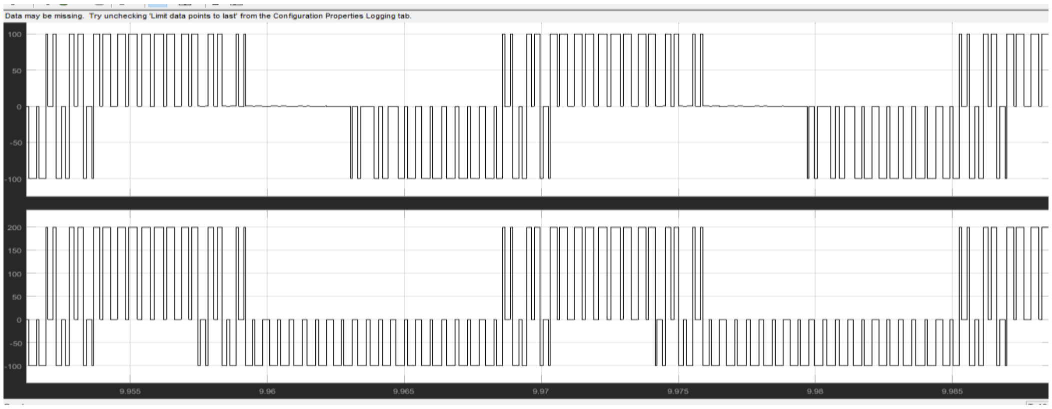
EXPECTED GRAPH

Three phase full converter:



output voltage and current waveforms of Three Phase Full ConverterThree

Phase PWM Inverter:



Data may be missing. Try unchecking 'Limit data points to last' from the Configuration Properties Logging tab.

output voltage and current waveforms of Three Phase PWM Inverter

RESULT:

PRE-LAB VIVA QUESTIONS:

1. What is PWM?
2. What is Duty cycle?
3. What is three phase converter?
4. What is an inverter?

POST LAB VIVA QUESTIONS:

1. What are the advantages of PWM inverters?
2. What is the difference between three phase and single phase inverters?
3. What is the time delay for each thyristor conduction in three phase full converter?

Exp. No.: 15 **Date:**

MATLAB Program to Solve Swing Equation using Point-by-Point Method

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 Xd=0.35p.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. **Solve the swing equation by point-bypoint method theoretically and verify using MATLAB Program. Comment on system stability.**

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:

Commands used:

RESULT:

0 0.1 0.2 0.3 0.4 0.5
0
20
40
60
80
100
120
140
160
180
200
swing equation by point by point method
time in seconds
angle in degrees
stable for 6.25 cycle CB
unstable for sustained fault (tc=5sec)

Exp. No.: 16

SIMULINK MODEL OF SINGLE AREA LOAD FREQUENC CONTROL WITHOUT AND WITH PI CONTROLLER

AIM: To find dynamic response of the given single area load frequency control problem theoretically and to plot and verify the results in SIMULINK.

PROBLEM:
The parameters for load frequency control of a single area are:
Speed governor gain Kg=10
Time constant of speed governor Tg=0.4
Speed regulation of speed governor R=3
Gain of turbine Kt=0.1
Time constant of turbine Tt=0.5
Gain of power system Kp=100
Time constant of power system Tp=20
Changes in the load ΔPD=0.01 pu
An integral controller with gain Ki=0.09 is now used to reduce steady state error.
What is the dynamic response of the system with and without the controller?
Obtain the dynamic response of the system with and without the PI controller by developing a SIMULINK model and verify the responses

SIMULINK MODEL WITHOUT & WITH PI CONTROLLER:
-0.09
s
Transfer Fcn6
100
20s+1
Transfer Fcn5
0.1
0.5s+1
Transfer Fcn4
0.4s+1
10
Transfer Fcn3
100
20s+1
Transfer Fcn2
0.1
0.5s+1
Transfer Fcn1
0.4s+1
10
Transfer Fcn
Subtract2 Subtract3
Subtract1
Subtract
Step1

Step
Scope1
1/3
Gain1
1/3
Gain
0
Constant

EXPECTED OUTPUT:

0 2 4 6 8 10 12 14 16 18 20
-0.05
-0.04
-0.03
-0.02
-0.01
0
0.01
X: 20
Y: -0.0002159
Time in sec
Frequency deviation in PU
Dynamic response of single area load frequency control
X: 20
Y: -0.02915
WithPI Controller
Without PI controller

RESULT: