Modeling of Ungrounded Shipboard Power System in Pspice

H.Zhang        K.L.Butler        N.D.R.Sarma

Power System Automation Lab
Electrical Engineering Department
Texas A&M University

Abstract: Navy shipboard power systems have different characteristics when compared to utility power systems. To conduct system studies on shipboard power systems, an effective simulation tool is required to model shipboard electric power systems. PSpice is a very powerful tool for analog/digital circuit simulation, but it is rarely used for power systems. This paper presents modeling shipboard power systems with PSpice. The detailed model for system components like generators, cables, loads, etc. are provided. Transient simulation results are presented.

Keywords: Shipboard power systems, transient simulation, fault analysis, PSpice

I. INTRODUCTION

Ungrounded delta type system configuration is widely used in U.S. Navy shipboard electric power distribution systems with the objective of providing continuity of power supply. To conduct system analysis, such as fault analysis, system reconfiguration/restoration, etc., for Shipboard Power Systems (SPSs), an effective simulation tool is needed. In the literature, there is no information discussing the appropriate tool for simulating SPSs. PSpice[1] is a popular simulation tool used for digital circuits [2,3]. In this paper, modeling and simulation of SPSs using PSpice are discussed.

In Section II, SPSs are briefly described. Section III explains PSpice modeling. Section IV gives the test system and the case studies conducted using the PSpice simulation tool. Concluding remarks are given in Section V.

II. SHIPBOARD POWER SYSTEMS (SPSs)

Navy Ships use three phase power generated and distributed in an ungrounded delta configuration which is used to ensure continued operation of the electrical system in the presence of a single phase ground fault [5,6]. The voltages are generated at levels of 450 volts a.c. at 60 hertz. The most popular topology used in Navy electrical systems is a ring bus configuration which allows any generator to provide power to any load. This feature is of great importance in order to ensure supply of power to vital loads if failure of an operating generating unit occurs. Generators are connected to switchboards called generator switchboards, which are composed of one or more switchgear units and breakers for various devices like the generator, loads and load center switchboards. These generator switchboards are located close to their associated generators. Bus tie circuits interconnect the generator switchboards which allow for the transfer of power from one switchboard to another.

Ship generators/switchboards are placed on different levels (floors) and are horizontally displaced along the ship to prevent total outage during battle. There are three major types of navy ships utilized during battle: submarines, surface combatants and aircraft carriers. Generally, a combatant type ship consists of a three generator system in the ring configuration; two of the generators would be used for normal operation with the remaining generator serving as emergency supply.

The other components of navy electric power systems are load center switchboards, power distribution panels, bus transfer units, transformers, and interconnecting cable used for delivering power to the loads. Load center distribution, which is a modification of radial distribution, is used below the generator switchboard level. One or
more load center switchboards are connected to each generator switchboard to supply power to load concentrations in various areas of the ship. The load center switchboards supply power to power panels or individual loads, either directly or via automatic bus transfers (ABTs) or manual bus transfers (MBTs). Power distribution panels are centrally located to the loads that they feed. They provide control and protection of selected portions of the power for lighting distribution systems and special power distribution systems.

Loads are fed directly from generator switchboards, load centers or power distribution panels. The loads require power at 440, 115, and 4,160 volts at 60 hertz, and 440 and 115 volts at 400 hertz. The loads requiring 400 hertz are typically part of the command and surveillance systems, weapons systems, and aircraft and aviation support equipment. The 4,160 volt loads are typically associated with aircraft carriers. Transformers are used to convert supply voltage of the distribution system from one level to another. The interfaces used between the 60 hertz and 400 hertz systems are either motor-generator sets or static solid-state frequency converters (SSFCs).

Although power systems found in navy ships appear simpler than their commercial terrestrial counterparts, shipboard electrical systems have unique characteristics that demand special attention during design and when performing other kinds of investigations, such as fault studies.

The protection system must perform efficiently in selectively isolating only loads affected by fault conditions and provide for reconfiguration to restore service as quickly as possible.

In the studies presented in this paper, a simple test system composed of navy power system component models was developed to conduct simulations.

### III. PSPICE MODELING

PSpice was used to simulate a simple test system which includes components common to SPSs. These components include navy cable, delta-connected three-phase generators, delta-connected three-phase transformer banks, ABTs and MBTs, circuit breakers, loads, buses (generator switchboards, load center switchboards, and power panels), and monitoring elements. The following sections will show how these components are modeled in PSpice.

PSpice, a computer software developed by Microsim Corporation, is considered to be one of the most powerful and popular analog circuit simulators available at present. PSpice uses linear, discrete, passive and active electrical elements, etc. for network simulation. Systems are built by combining components in an electrical model, called a PSpice model.

To simulate a navy power system network, the appropriate PSpice models of the components of the SPSs were created from the available elements. There are a few ways to make the PSpice models. The method adopted in this paper is the so called “manually create symbol using the symbol editor” method. The procedure is:

- Create the PSpice subcircuit code (in the schematics editor or in any text editor) which gives the circuit connection and save it in a ‘.lib’ file
- Draw the schematic object and save it in a ‘.slb’ file, the name for ‘.slb’ file should be the same as that of ‘.lib’ file
- Assign input/output “pins”, definition, and system attributes to the object.

### A. Modeling of SPSs components in PSpice

1. Generator model
The voltage sources available in PSpice were used directly to build the generator sets. The voltage sources are connected in delta to get the generator model. SPSs are ungrounded system and do not have a common reference. However, PSpice requires a common reference and doesn’t allow the floating conditions. Therefore, shunt resistors were connected across each phase and ground to avoid the floating condition. The parameters of the generator model include: voltage magnitude, frequency, source impedance of each phase and shunt impedance of each phase which are assigned by the users. The ports of the generator symbol are the three phase generator terminals.

(2) Transformer model

Three-phase delta-connected transformer banks were modeled as three delta-connected single phase transformer. PSpice provides modeling of a single phase transformer with only an inductance. Hence a resistance was added to the basic transformer model provided in PSpice. A large resistance ($10^9$ ohms) is connected across line and ground in each phase on the primary and secondary sides of the transformer to avoid the floating condition. The parameters for the transformer model include: primary/secondary inductance/resistance, coupling coefficient and shunt resistance.

(3) Cable model

Cables of SPSs were modeled as a lumped π-model whose components are calculated using the distributed resistance $r$, self-inductance $l_{self}$, self-capacitance $c_{self}$, mutual inductance $l_{mutual}$ and mutual capacitance $c_{mutual}$. The value of these parameters depend on the type of cable and its length. The parameters of the cable model include: distributed $r$, $l_{self}$, $l_{mutual}$, $c_{self}$, $c_{mutual}$ and the cable length.

(4) Three phase switches

PSpice provides two types of single phase time-controlled switches, one initially closed and another initially opened. In SPSs, there are three phase switches. Accordingly, two types of three phase switches are modeled using single phase time-controlled switches available in PSpice. One is normally closed and another is normally opened. The parameters of the three phase switches include the open / close time for each phase, transient time and the resistance across the switch when it is open or closed. In our simulation, a very large value of the resistance ($10^9$ ohms) is selected when opened and very small value ($10^{-9}$ ohms) when closed.

(5) ABT/MBT model

Automatic Bus Transfer switches (ABT) or Manual Bus Transfer switches provide the normal and alternate path to vital loads. The bus transfer symbol is shown in Fig.1. It is composed of one 3-phase normally closed switch S1 (connecting node 1 and 2 as normal path) and one 3-phase normally opened switch S2 (connecting node 1 and 3 as alternate path). When the path through node 2 is not available to node 1, switch S1 and S2 will exchange their status at the same time; the opened one will be closed and the closed one will be opened. Thus, the supply to node 1 will now come through node 3 instead of node2. The parameters of the ABT/MBT model include the switch time, transient time, the resistance in open state and the resistance in closed state.
(6) Switchboard, load center and power panel models

Generator switchboards are composed of one or more switchgear units and breakers for various devices like
the generator, loads, and connected load center switchboards. These generator switchboards are located close
to their associated generators. Bus tie switches interconnect the generator switchboards, thereby allowing
the transfer of power from one switchboard to another. In the current study, a switchboard along with the associated
circuit breakers/switches and bus tie breakers is modeled as a single unit. The parameters of the switchboard
include the parameters of each circuit breaker or switch. For a load center and power panel, the model is
similar to that of the generator switchboard but without the bus tie switches.

(7) Load model

The loads are represented by the real and reactive impedance components connected in delta. It is a three
phase symmetrical constant impedance load. The parameters include resistance and inductance of the load.
For the pure resistive load, only resistance is included. Other loads such as motors will be modeled in future
work.

To build a system from the component models in PSpice, the appropriate models are dragged to the work
space and are interconnected to create the system network. Input parameters are entered for each component
model to appropriately define it according to the specifications of the system.

B. Monitoring of Parameters

In PSpice, the parameters of interest can be marked in the schematics for monitoring. After or during
simulation, a graph software called Probe can be activated to give the waveforms of the marked parameters.
Even the parameters that are not marked in the schematic are available for monitoring in the probe parameter
trace list, which consists of all branch currents and all node voltages. A parameter of interest may be added for
monitoring even after the simulation is concluded by simply selecting the parameter from the trace list. It is not
necessary to run the simulation again which is one of the important advantages of PSpice. Further, it is possible
to define analog trace expressions, including any combination of analog simulation output variables, arithmetic
operators and functions. For example, the difference of two node voltages can be easily monitored.

IV TEST SYSTEM AND CASE STUDIES

A test system consisting of components of a typical SPS as shown in Fig. 2(a) was designed for simulation in
PSpice. The test system includes three-phase delta-connected voltage sources (generator G1) of rating 450V
line-to-line, connected to a switchboard by a 6.1-meter long cable of gauge 500 mcm, and a 12.2-meter long
cable of gauge 350 mcm connecting the load center LC1 to the generator switchboard. Both cables were three-
phase, and the cable system was modeled as two meters above the ground. A lumped three-phase LR load, L1,
(R=10.173Ω, L=15.338mH) is connected to load center LC1. A three-phase, 450/115 V transformer T1 with
three-phase delta-connected resistive load L2 (R=132.25 ohms) is also connected to the load center. The Fig
2(a) is the single-line diagram of the test system and Fig.2(b) is the PSpice schematic of the test system. Here
the generator switchboard and the load center switchboard are simplified as buses.

Simulations for both a normal operating condition and a fault condition were conducted. Since the single-line-to-
ground (SLG) fault happens most frequently in both utility power systems and shipboard power systems, this
fault analysis was conducted in present simulations. The time-controlled switch S1 is used to simulate single-
line faults. This switch is connected between phase and ground as shown in Fig.2(b). Initially, the switch is
opened. To stage a fault, this switch should be closed at the specialized time(when the fault occurs). Once the
switch is closed, it resembles a short circuit between phase and the ground. Thus a time controlled switch is
used to simulate a fault. For the test system, it was assumed that the fault location was on phase c in the middle
of the second cable which connects the switchboard and load center. The SLG fault is staged at 60 milliseconds
after the start of simulation.
Accordingly, the following cases were studied:

- **Case 1**: No fault for test system
- **Case 2**: SLG fault for test system

For the test system, the following parameters were monitored as shown on Fig. 2(c) and 2(d).

- \( V_{ab}, V_{bc}, V_{ca} \) → line voltages of the transformer primary side shown in Fig. 2(d)
- \( V_{a'b'}, V_{b'c'}, V_{c'a'} \) → line voltages of the transformer secondary side shown in Fig. 2(d)
- \( I_{ab}, I_{bc}, I_{ca} \) → 450V load currents shown in Fig. 2(c)
- \( I_{a'b'}, I_{b'c'}, I_{c'a'} \) → 115V load currents shown in Fig. 2(d)
- \( I_a, I_b, I_c \) → Transformer primary line currents shown in Fig. 2(d)
- \( I_{a'}, I_{b'}, I_{c'} \) → Transformer secondary line currents shown in Fig. 2(d)
V RESULTS AND DISCUSSION

A. Simulation Results

Discussions of the results of the simulations follow below. Each case is illustrated with waveforms. PSpice adjusts the time step for simulation automatically, but one can define the maximum time step, called the step ceiling. For all the simulations in current studies, the step ceiling is chosen as 50µs.

Case 1

The waveforms for transformer primary and secondary voltages are shown in Fig. 3 and Fig. 4 respectively and those for 450V and 115V load currents are shown in Fig. 5 and Fig. 6. The waveforms for transformer primary and secondary currents are shown in Fig. 7 and Fig. 8 respectively. For transformer primary current, a long duration waveform is illustrated to demonstrate the transient and steady state behavior of the PSpice tool and a short duration to show the detailed behavior of the waveform. From Fig. 7, it can be seen that there is a transient in current on the primary side of the transformer. However, the transient is not present in the secondary side current waveform shown in Fig. 8. This behavior is due to the isolating nature of the transformer.

From the simulation results, it was found that the steady state value for each monitored parameter is as expected. From the waveforms of transformer primary and secondary voltages shown on Fig. 3 and Fig. 4, it can be seen that the primary steady state voltage (peak value) is 632V and secondary steady state voltage (peak value) is 161V, which implies that the voltage ratio is 632/161 = 3.925. From the ratings, the voltage ratio is 450/115 = 3.913. Therefore, percentage error between the expected value and the actual simulation result is about 0.3%. For other parameters, the percentage errors are similar.

Case 2

The waveforms for line-to-line voltage Vca and line-to-ground voltage Vcg are shown in Fig. 9. It can be seen that Vca is unaffected by the fault, while Vcg drops to zero after some transient.

In Case 2, for each of the monitored parameters, the simulation results are similar to those of Case 1, with an exception that in some waveforms, there is a small spike at the fault time. For example, consider the waveform of primary voltage is as shown in Fig. 10. It can be seen that there is a small spike in Vbc and Vca waveforms at the fault time.

Case 1 and Case 2 simulation results illustrate that an ungrounded delta configuration ensures normal voltage supply even in the presence of a single phase ground fault. This behavior also explains why navy ship designers adopted the delta configuration.
Fig. 4 Case 1--transformer secondary voltage

Fig. 5 Case 1--450V load current

Fig. 6 Case 1--115V load currents

Fig. 7 (a) short period
Fig. 7 (b) long period

Fig. 7 Case 1--transformer primary current

Fig. 8 Case 1--transformer secondary currents

Fig. 9 Case 2-- Line-to-line voltage Vca and line-to-ground voltage Vcg

Fig. 10 Case 2 -- transformer primary voltage for case 2
V. CONCLUSION

SPSs components are modeled in PSpice. A test system containing typical components of Shipboard Power Systems (SPSs) was simulated using PSpice. Different case studies were discussed. PSpice has an excellent graphical interface for building complex SPSs and flexibility in establishing monitoring points and processing output data. However, PSpice must be given basic circuit parameters for each component model, which can be gotten from other power system simulation tools or can be directly calculated. In conclusion, PSpice is a promising tool for simulating SPSs.

VI. ACKNOWLEDGMENTS

The authors acknowledge the Office of Naval Research for the support of this project through grant N00014-96-1-0523.

VII. REFERENCES


VIII BIOGRAPHY

Haibo Zhang received a B.S.E.E from Tsinghua University, China, in 1993 and a M.S.E.E from North China University of Electric Power, China, in 1996. She is currently working towards a Ph.D. degree in the Electrical Engineering Department at Texas A&M University.

Karen L. Butler is an assistant professor in the department of electrical engineering and assistant director of the power systems automation laboratory at Texas A&M University. She received a B.S. degree from Southern University -- Baton Rouge in 1985, a M.S. degree from the University of Texas at Austin in 1987, and a Ph.D. degree from Howard University in 1994, all in electrical engineering. In 1988-1989, Dr. Butler was a Member of Technical Staff at Hughes Aircraft Co. in Culver City, California. Her research focuses on the areas of computer and intelligent systems applications in power, power distribution automation, and modeling and simulation of vehicles and power systems. Dr. Butler is a member of the Louisiana Engineering Society. She is a registered professional engineer in the State of Louisiana.

N.D.R. Sarma obtained B.Tech (Elect.) and M.Tech (Power Systems) degrees from Regional Engg. College, Warangal, India in 1983 and 1986 respectively and a Ph.D. from Indian Institute of Technology, Delhi, India in 1995. From 1992 to September 1997 he was with the R&D Division of CMC Limited, Hyderabad, India. Since October, 1997 he is on sabbatical leave from CMC and is presently working as a Post Doctoral Research Associate in the Power System Automation Lab in the EE Dept. at the Texas A&M University.