



Fault and Load Settings Variation Using Master-Slave Feature

Written for PSCAD v4.5 and v4.6.

1. General description of the example

This example shows how to apply different type of faults on different locations of a transmission line automatically while the time and resistance of the fault is varied. The parameters changing in this example are listed as follows:

- Fault Time: This can be between a minimum and maximum fault time.
- Fault Type: Fault type i.e. three-phase to ground (7), two-phase to ground (4) and single-phase to ground (1).
- Fault Location: The location of fault in the power system.
- Load power: P and Q are active reactive powers of the load.

Also, this illustrates the PSCAD capabilities and is helpful to do a new simulation setup. In this example, a simulation set can be created using one simulation case as a Master and the other as a Slave.

Load the workspace “MasterSlave.pswx” into PSCAD as shown in Figure 1.

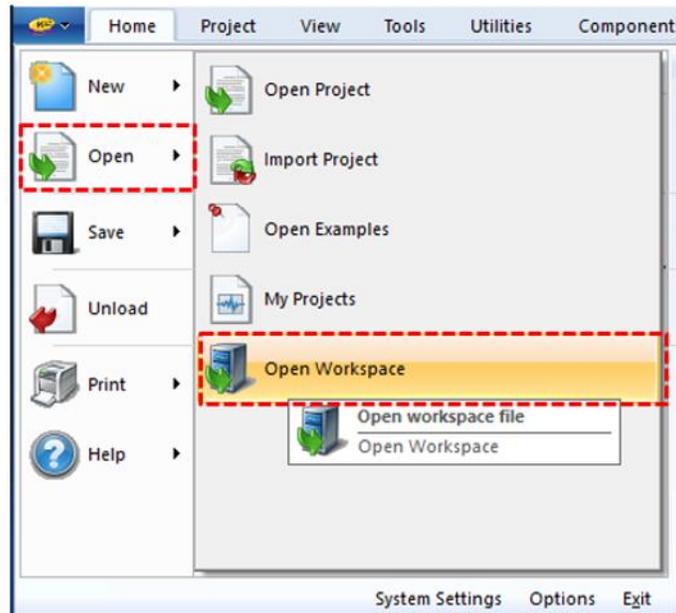


Figure 1: Load the workspace “MasterSlave.pswx”

As shown in Figure 2, by loading the workspace, the two PSCAD cases (MasterCase.pscx and SlaveCase.pscx), will load into PSCAD.

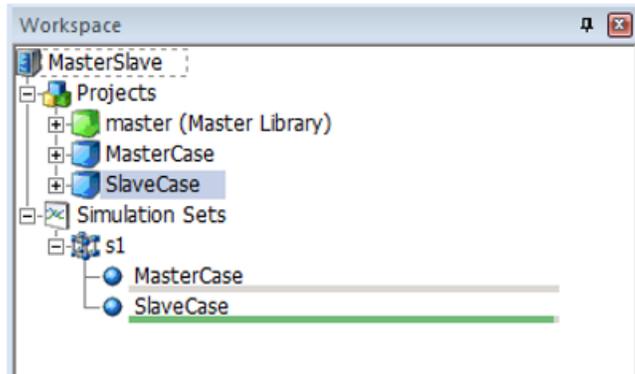


Figure 2: Master case and slave case are loaded into the workspace automatically

As shown in Figure 3 and Figure 4, a right-clicking on the project case and selecting project settings displays the following windows show that the Run Configuration which is set to Master and Slave for the cases.

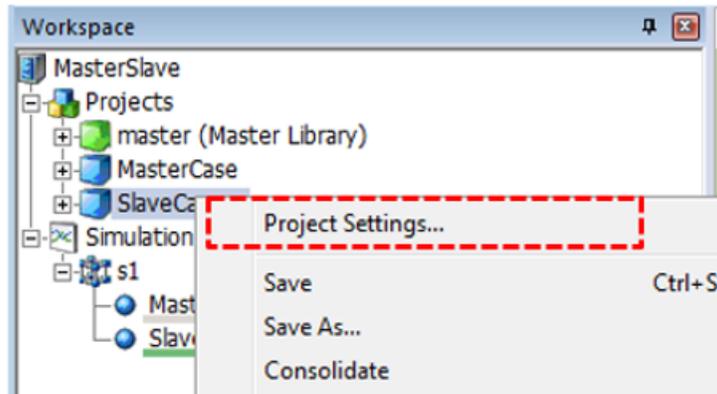


Figure 3: Find the project settings

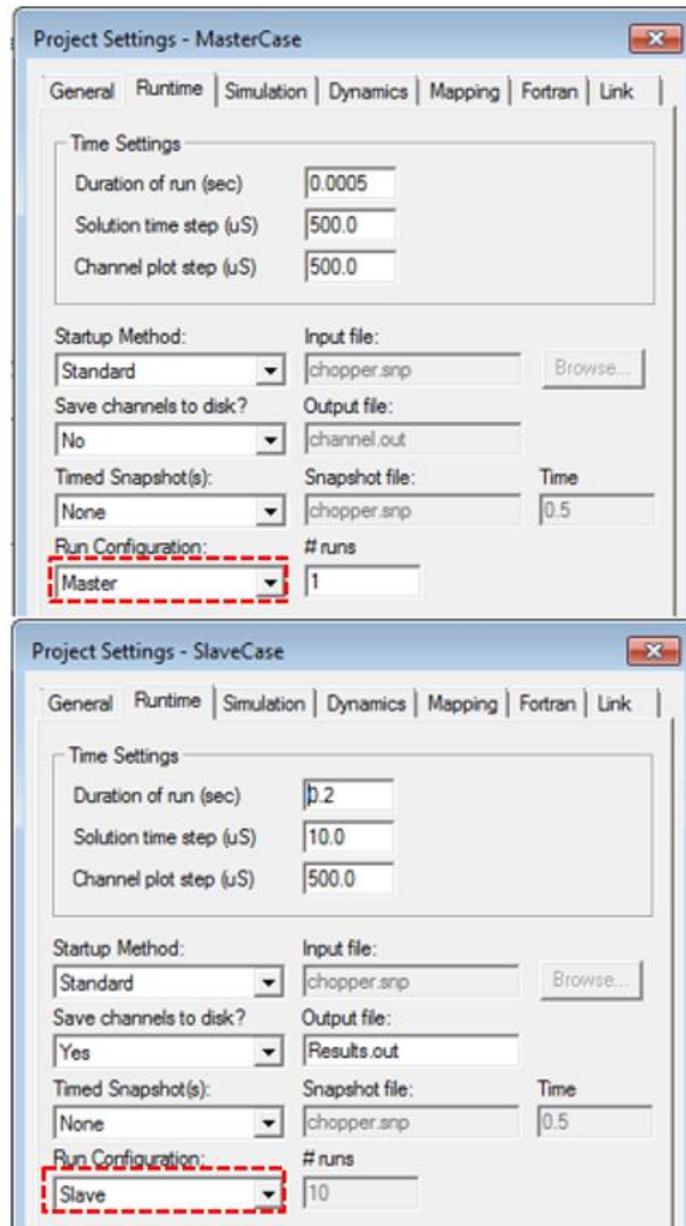


Figure 4: Define the project settings for the Master and Slave case

In the Master case, there is a multiple-run component that runs the simulation several times. During each run, the values of the parameters are read from text files in the Master case and sent to the Slave case. A simple circuit with a transmission line is implemented in the Slave case, and the parameter values received from the Master case are applied to the power system and run the simulation for those values. The simulation results can be saved into files in the Slave case.

2. Procedure

Right-click on the “Simulation Sets” to create a simulation set “S1” as shown in .Figure 5

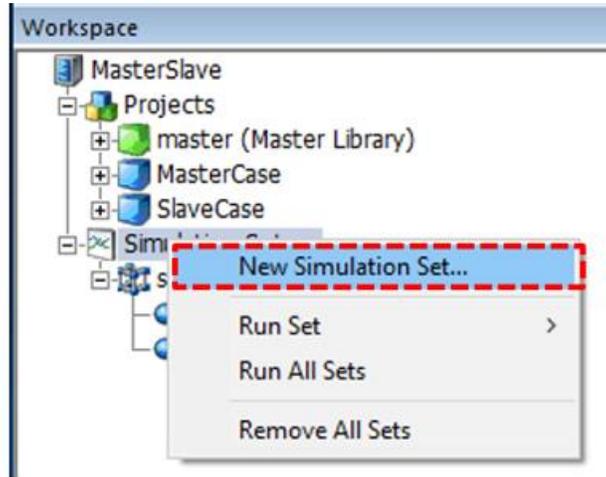


Figure 5: Create a simulation set

Then include the “MasterCase” and “SlaveCase” into the simulation Set (i.e. S1) as shown in Figure 6.

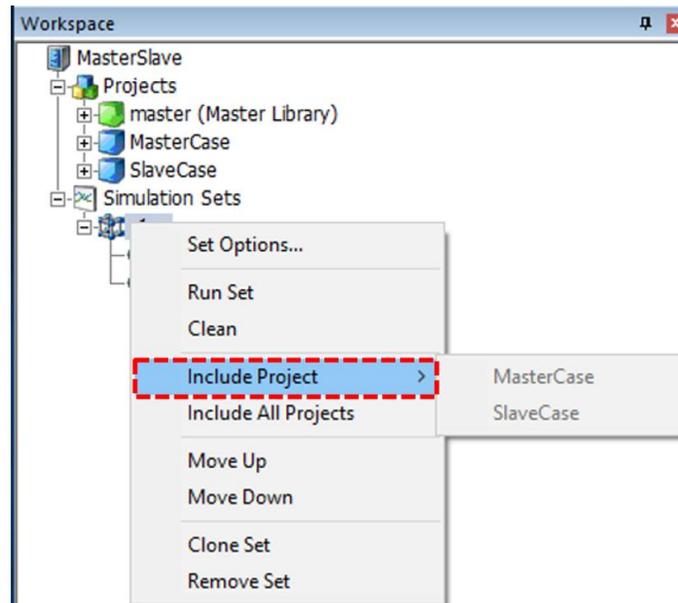


Figure 6: Include the master and slave projects to the simulation set

To run the set, as shown in Figure 7, right-click on “S1” and select “Run Set”.

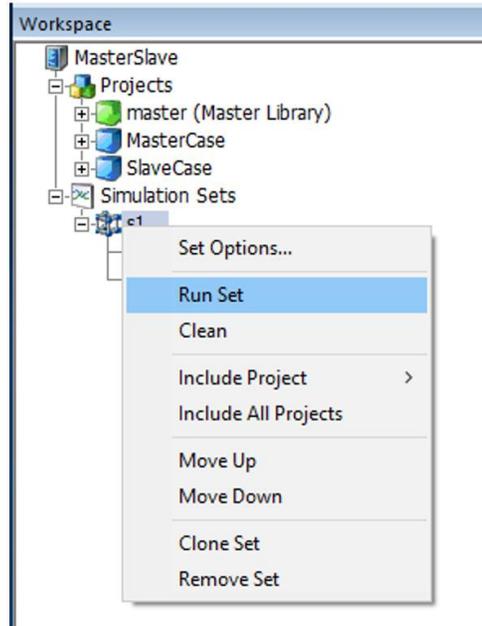


Figure 7: Run the simulation set

It can be seen how the simulation will run and how the values and location of fault can be changed in the Slave Case. Also the active and reactive powers of the load at the end of the transmission line varied for each run.

2.1 Simulation Settings

In the MasterCase, double-left-click on the multiple run, and the following settings can be seen on the first page, as shown in Figure 8.

The number of signals (variable) to control is “one” for this multiple run and the variable data type is “Integer”.

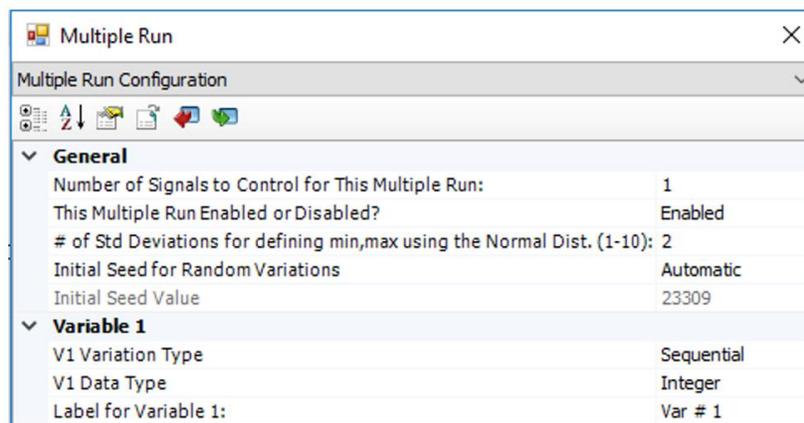


Figure 8: Multiple run setting

As shown in Figure 9, on the other page the number of runs is also adjusted to 162 and the Start, End and Increment for each run is defined. This means that PSCAD runs the case for 162 times (in this example) and in each run the value of the Variable increases by one.

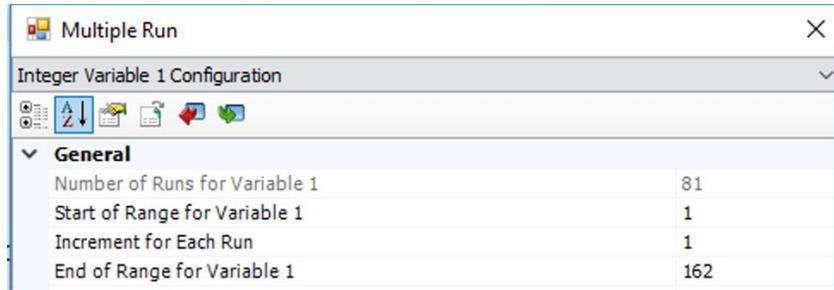


Figure 9: Load the workspace "MasterSlave.pswx"

As shown in Figure 10, in the excel file (faultData.xlsx) the load data including active and reactive powers (P and Q) and fault data (fault location, Type and time) are presented. In order to load the data into PSCAD they are saved in different text with a certain format.

Run #	P	Q	Location	Type	Fault time
1	25	50	1	1	0.1
2	25	50	1	1	0.101
3	25	50	1	1	0.102
4	25	50	1	1	0.103
5	25	50	1	1	0.104
6	25	50	1	1	0.105
7	25	50	1	1	0.106
8	25	50	1	1	0.107
9	25	50	1	1	0.108
10	25	50	1	4	0.1
11	25	50	1	4	0.101
12	25	50	1	4	0.102
13	25	50	1	4	0.103
14	25	50	1	4	0.104
15	25	50	1	4	0.105

Figure 10: Load (p and Q) and fault data (location, type and time)

The first column shows the Run # and the parameters are:

- Fault Time: Fault time varies between 0.1 to 0.108sec (signal name FTime).
- Fault Type: Fault type i.e. three-phase to ground (7), two-phase to ground (4) and single-phase to ground (1) (signal name Location).
- Fault Location: The location of fault in the power system (signal name Type).
- Load power: P and Q are active reactive powers of the load (signal names P and Q).

For the first 27 runs, the simulation applies fault to the first location at different times (varies between 0.1 and 0.108sec) and different type of faults (A-to-G (1), AB-to-G (4) and ABC-to-G (7)).

For the second set of 27 runs, the simulation apply fault to the second location, and so on.

After run # 81 the load power is changed. Please see the excel sheet to observe the variations in load flow.

The values of the above parameters are saved in text files.

For example, the variable “Location” is saved in “faultLocation.txt” as shown in Figure 11.

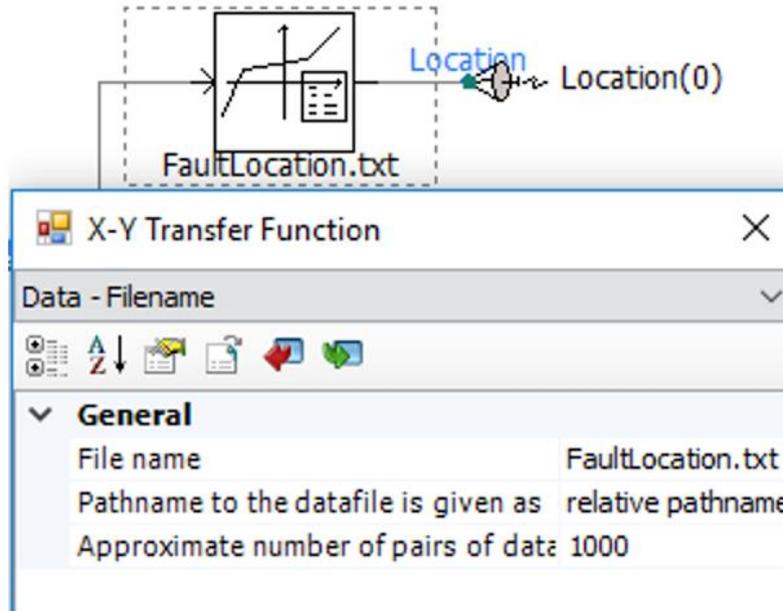


Figure 11: Fault location data which is loaded into PSCAD from a text file named FaultLocation.txt

The first column shows the Run # and the second column shows the fault location.

In this text file the location of faults are 1, 2 and 3 indicating three different locations.

Note:

The format of text file must be as it is used in this example.

2.2 Master Case settings

As shown in Figure 12, at each run the fault and load data are selected from the tables () and sent to the Slave Case.

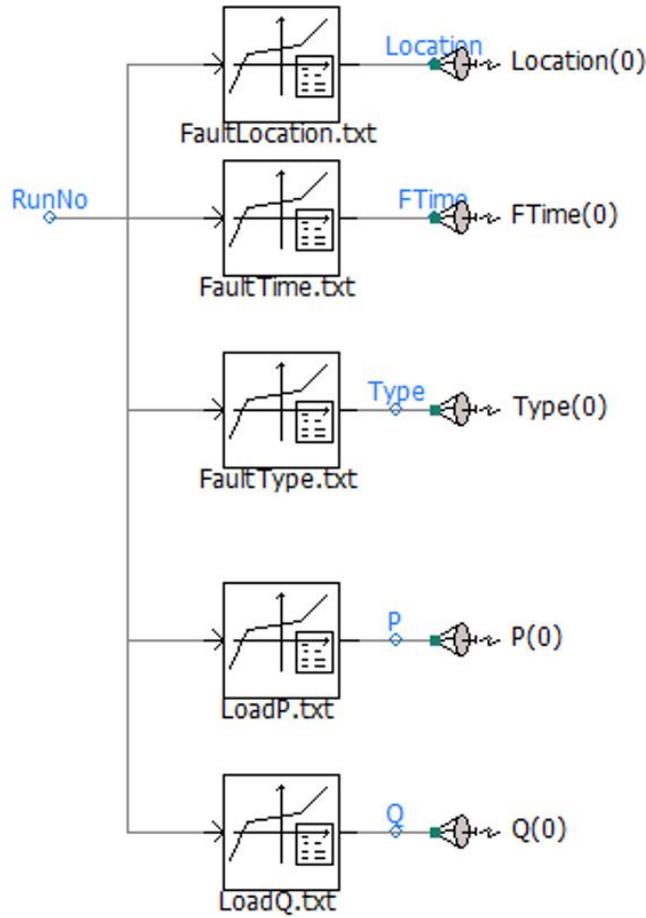


Figure 12: Based on the run number load and fault data are loaded to Master case and via transmitters are sent to the slave case

2.3 Slave Case settings:

In the Slave case, the data is received from Master case via transmitters as shown in Figure 13.

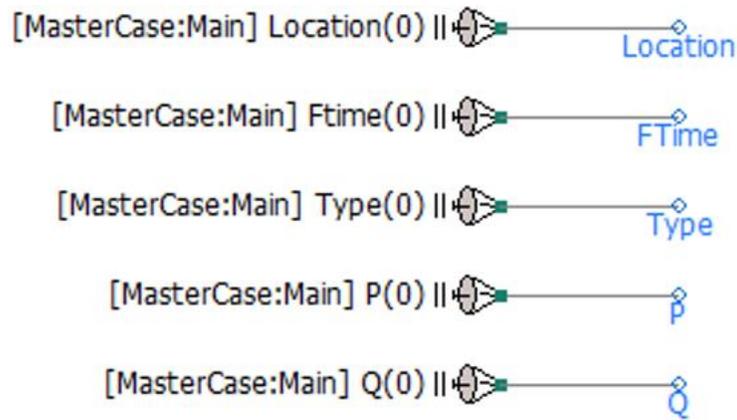


Figure 13: Transmitters that carry data from Master case to the Slave case

The location of the fault can be identified as shown as in Figure 14.

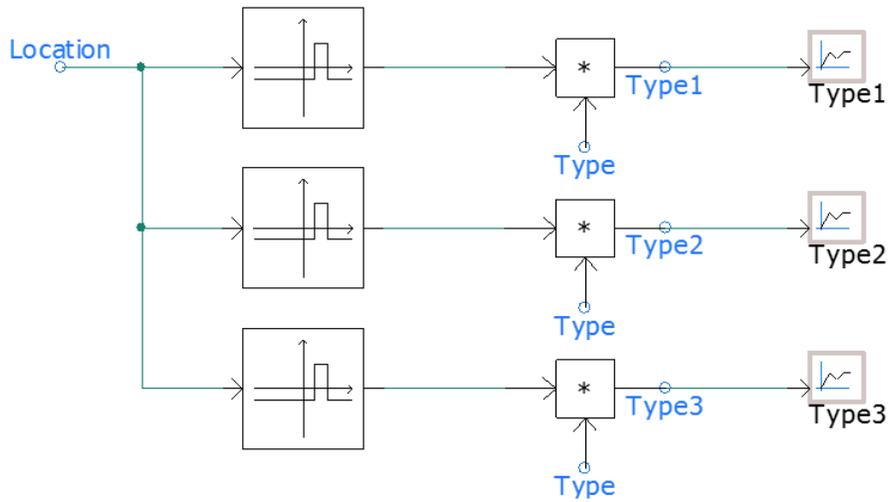


Figure 14: Define the location and type of a fault

The voltage signals; V1 and V2 are shown in Figure 15, can be saved into a file for each simulation run and then the results should be plotted using other software such as Excel. If one double-left-clicks on the channel, it can be seen that “Multiple Run Save” is set to “All runs”. This means the channel will save into a new file in each run.

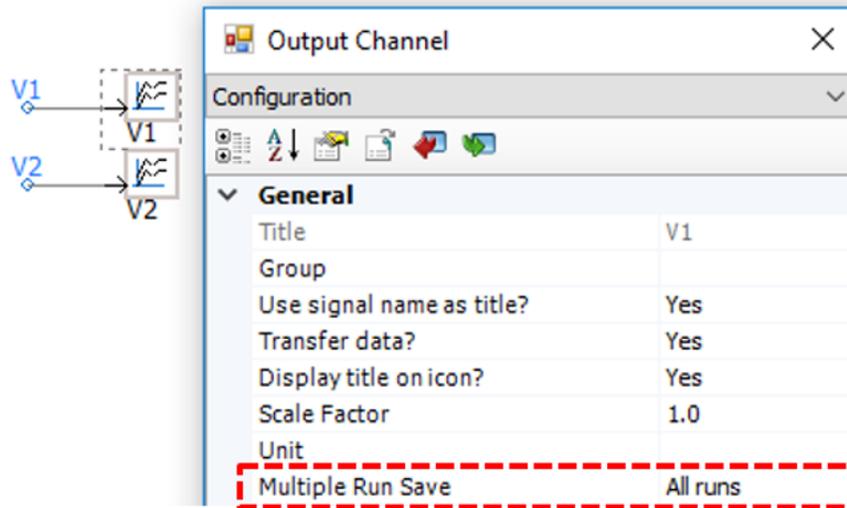


Figure 15: Voltage signals V1 and V2 that are connectd to channels be saved into file during simulation

As shown inFigure 16, in the Project Settings it can be seen that all the channels can be saved into an output file named “Results.out”:

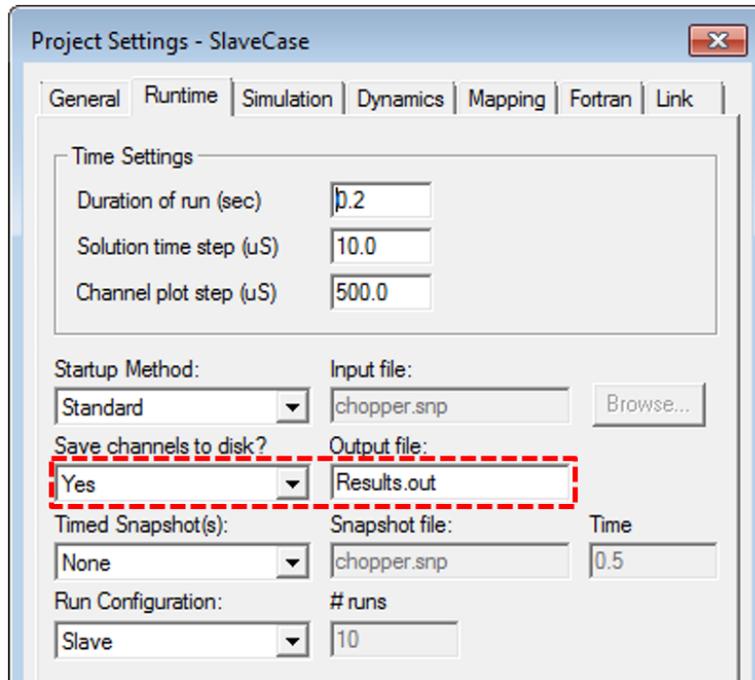


Figure 16: Project settings to save the channels (voltage signals) to disk

Run this simulation to see that PSCAD will run the case for 162 times and at each time the channels are saved in the output file (Results.out).

When the run is completed, right click on the simulation name and from the opening window as shown in Figure 17 and then select the “Show in the folder” to see the PSCAD simulation case in the folder on your computer.

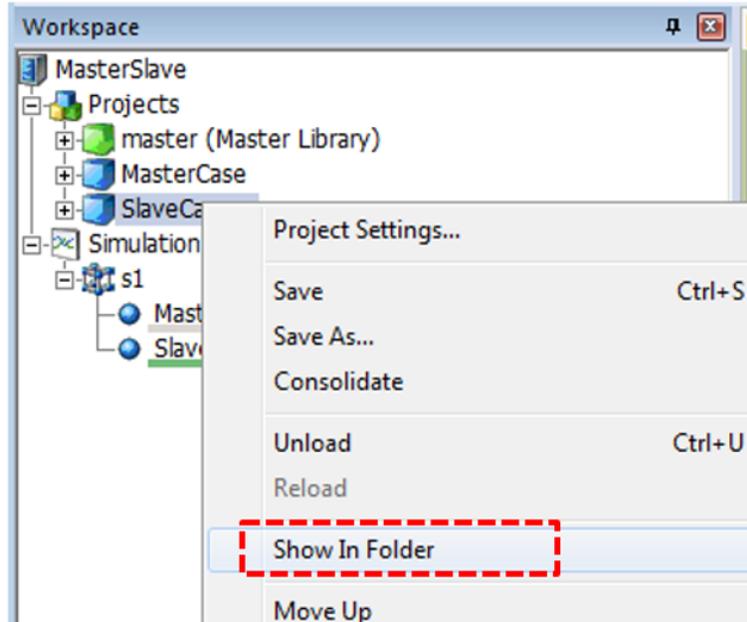


Figure 17: Find the simulation results (Results.out)

As shown in Figure 18, next to the PSCAD case there is a folder in which the simulation results can be found.

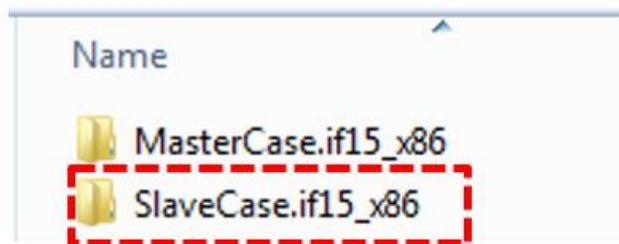
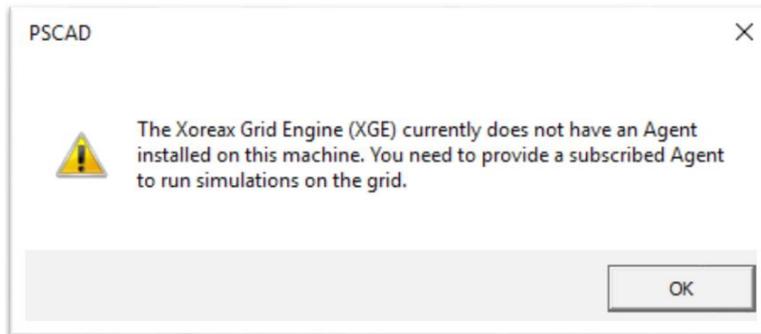


Figure 18: PSCAD foldere where the simulation results (Results.out) are saved

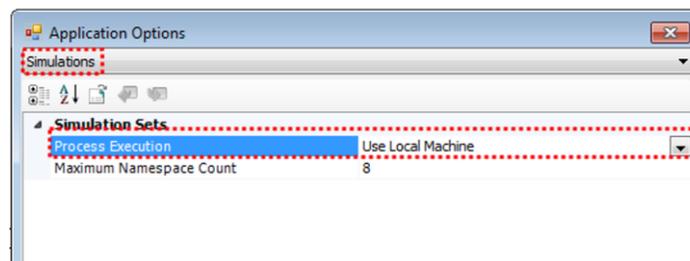
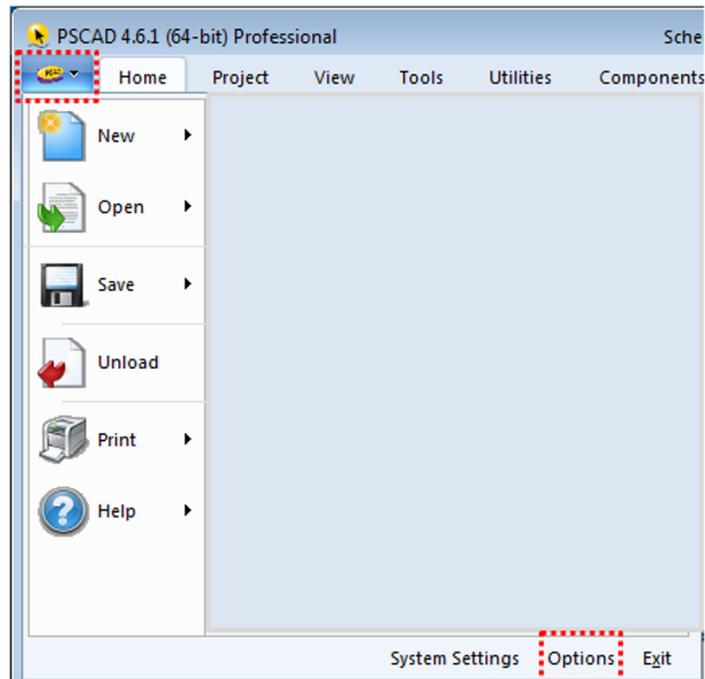
Double-left-click on the folder and find the output files generated by PSCAD at each run. Each file has simulation results for each run that can be loaded into Microsoft Excel in order to plot the voltage signals if they are required.

Appendix A: Resolving Xoreax Grid Engine Error

If the following error appears when trying to run the simulation set, please follow the solution as follows:



- Set the Process Execution as shown:



Copyright © 2018 Manitoba Hydro International Ltd. All Rights Reserved.