PRSIM User Manual v1.1.0

January 4, 2023
Initial
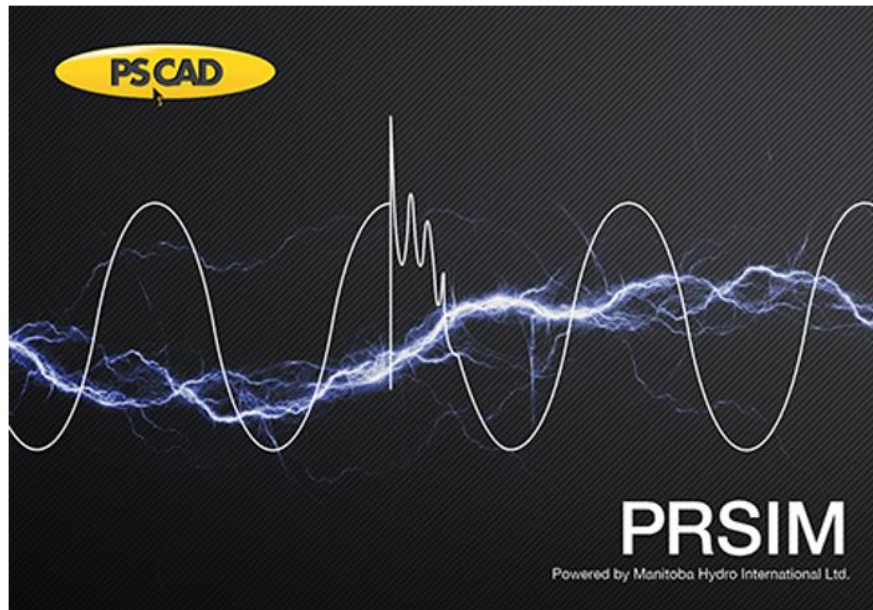
CONTENTS

1. OVERVIEW

The **PRSIM** (**P**owe**R** **S**ystem **IM**porter) import tool can convert **PowerFactory** DIgSILENT (PFD) and **PSSE** network data to PSCAD format which may be used to conduct electromagnetic transient simulation studies. PRSIM bridges the gap in the user transition from the phasor to the EMT domain platforms.

2. DATA EXPORT

PowerFactory versions (2019, 2020 and 2021) must first be converted with DGS version 6.00 in the XML file format which can then be imported by PRSIM. On the PSSE side, RAW file version 33 is preferred however versions 29, 30, 31, 32, 34 and 35 are also supported. To export PFD or PSSE network information, activate the case and execute the loadflow calculations, then save the loadflow results to one of our supported versions. These results will be used later in PRSIM to initialize the PSCAD components to the same steady state operating point.

PowerFactory users exporting component parameters and loadflow results must also use the definition file provided in the PRSIM support folder named **DGS Export Definitions v110**. This will ensure the correct parameters are identified and exported to the XML file. Users must import the definition file into their PFD database and then they will see a list of the PowerFactory parameters chosen for each element to be exported. Once the user performs the export process with the definition file, an XML file will be generated that can be imported into PRSIM.

RAW format 33 is preferred for exporting PSSE loadflow results. Bus locations, dynamic information for synchronous machines, and sequence data for various elements of the circuit may be exported using LOC, DYR, and SEQ files respectively. The generated RAW, DYR, SEQ and LOC files may be used to translate PSSE information into PSCAD.
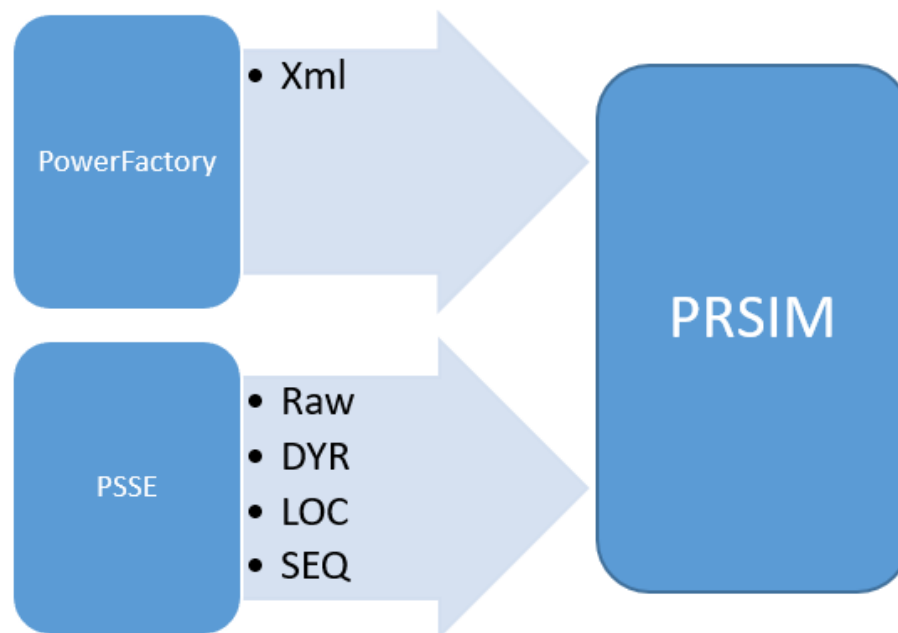
3. DATA IMPORT

You can launch PRSIM using the icon or menu item if you have it installed on your computer or by using the **MyUpdater** software. Once you have PRSIM running you will be able to import your previously generated xml or raw file into PRSIM. Begin by adding a **New** PRSIM case in the project folder under **File tab** of PRSIM.
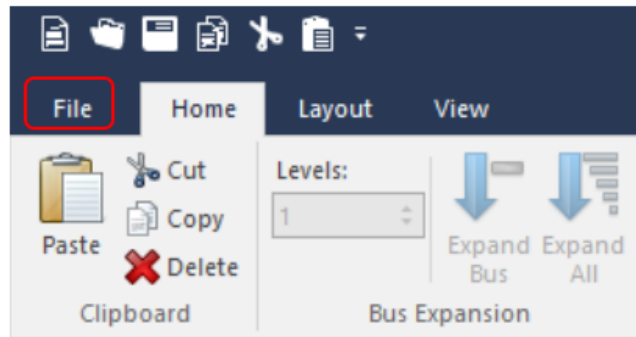

In the **New tab** you can:

1. Set the folder location here.

2. Set the name of the file here.

3. Make sure **New Case** is selected then Press the **Create** button to create a new PRSIM file which will be added to the workspace tree.
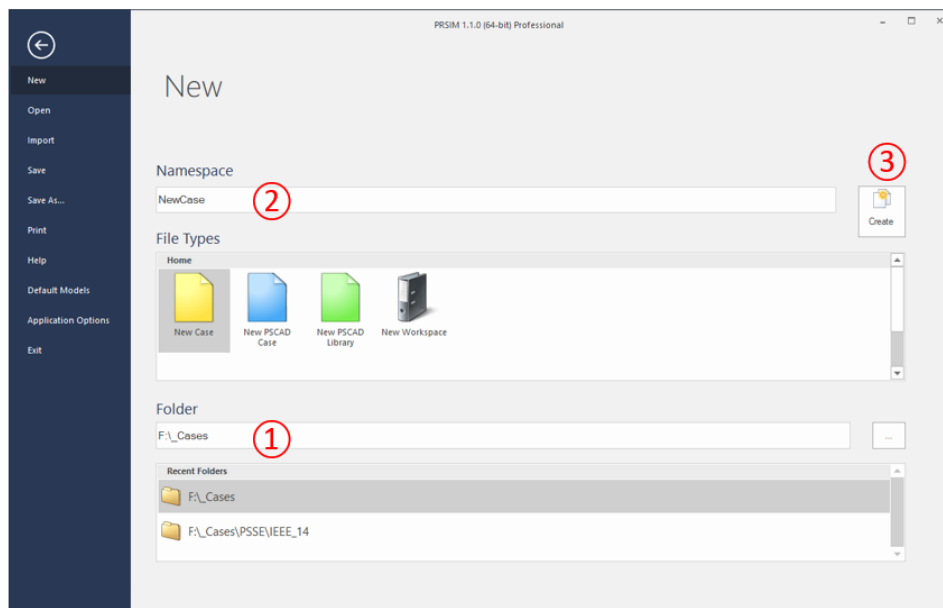
In the Workspace window right click on the new case in the project tree and select **Import RAW Data** or **Import DGS Data** to launch a browse dialog that you can use to select your xml or raw data file.

Click on the **Open** button to start the import process. If the data file is valid a set of data rows will be generated in the **Network Table** pane.
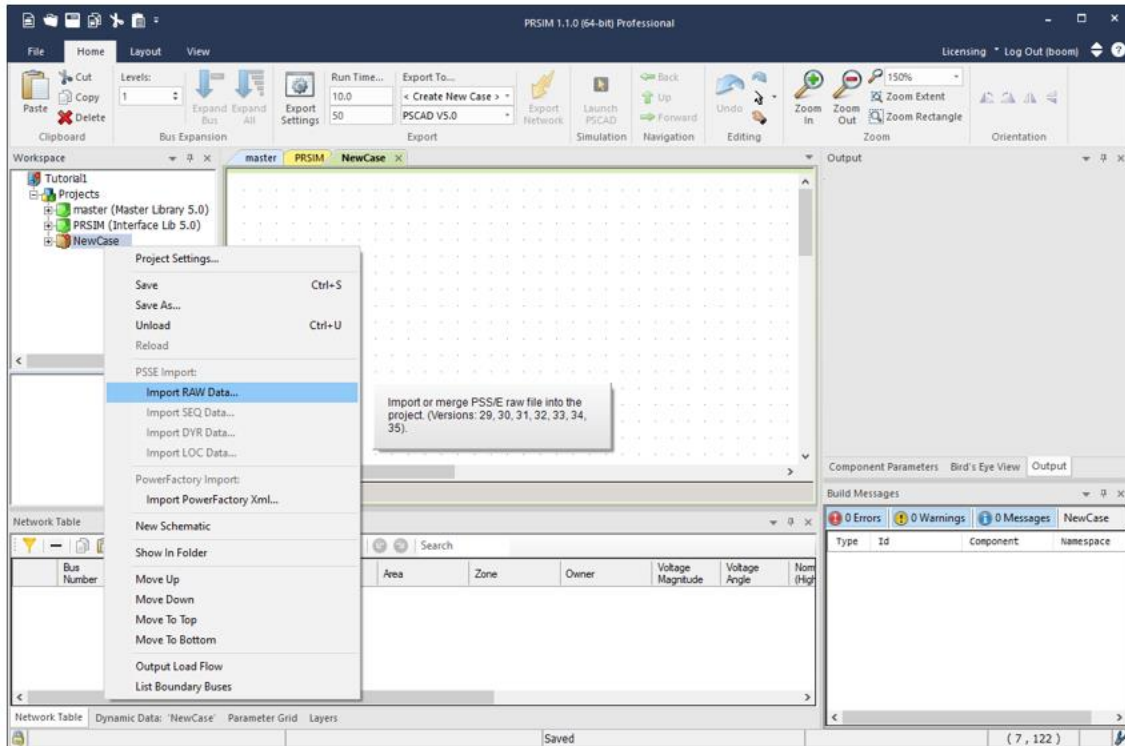
If you skipped or failed to import data in the previous step, open one of PRSIMs example cases by navigating to the **File** menu and click on the **Examples** button.
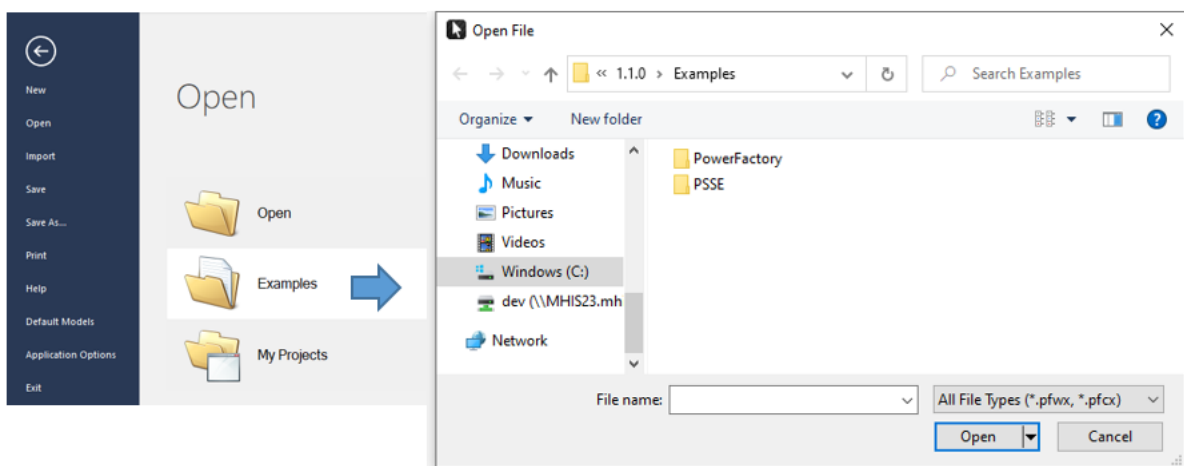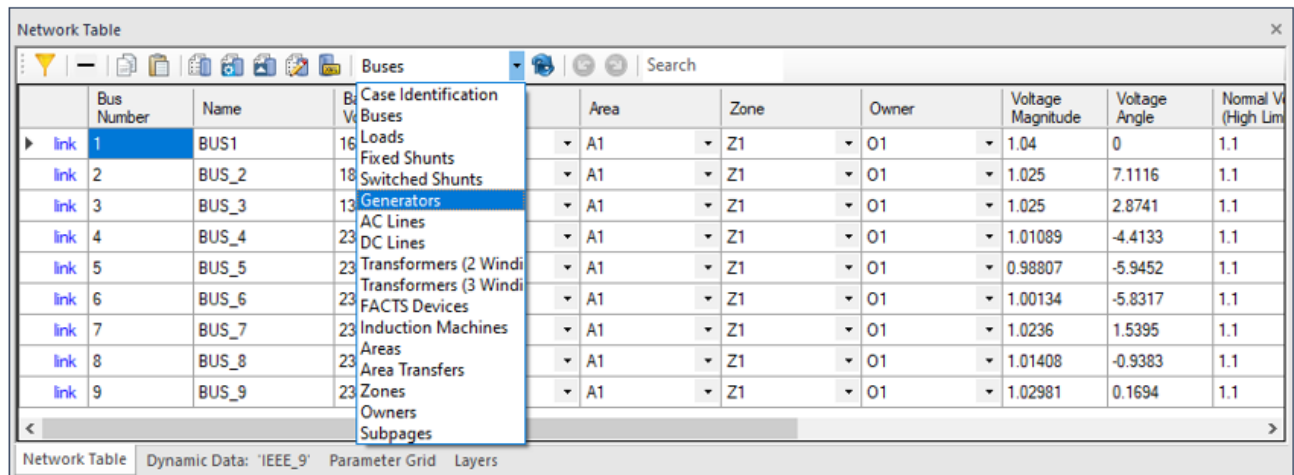
Your PRSIM will have a **PowerFactory** or a **PSSE** folder depending on your purchase. Navigate to the **IEEE_9** folder and open the example case you find there.
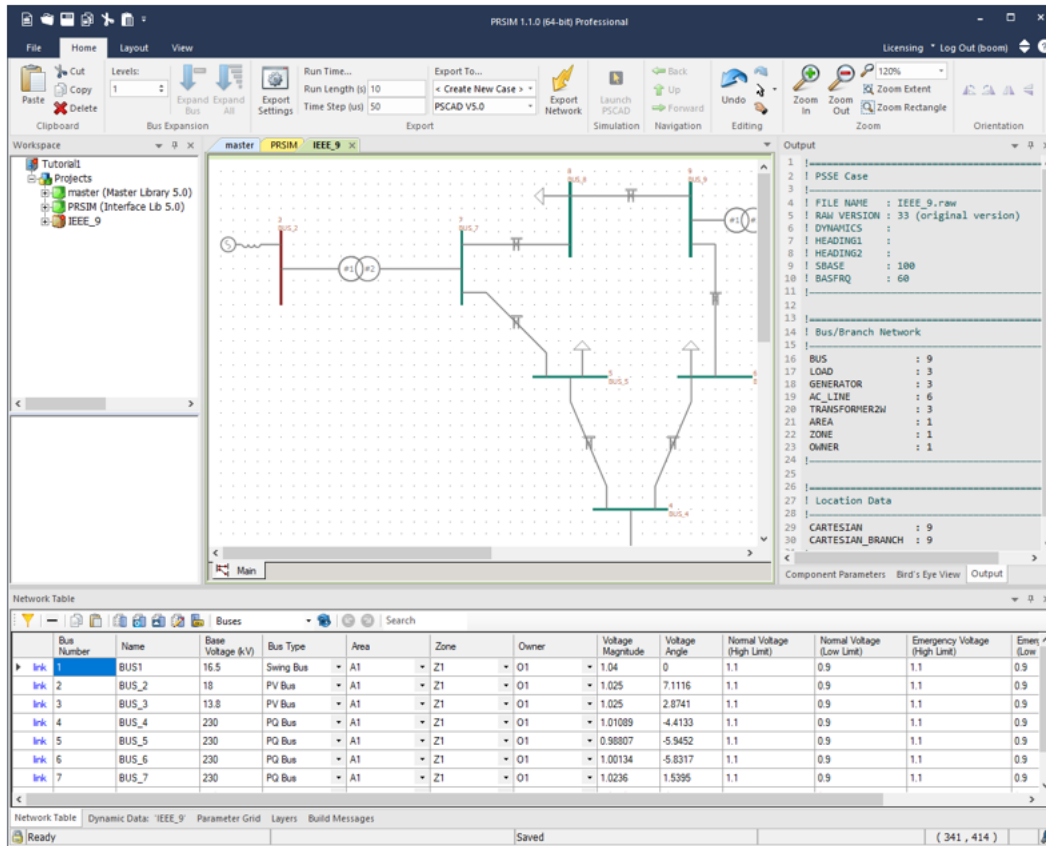
3.1.   Network Table

Once you have a case with a network loaded you will be able to view the data in the **Network Table** pane. You can use the dropdown list to view and visually verify the data in each table.

The Network tables represent information extracted from the PSSE or PowerFactory files. You can now draw a single line diagram of the buses you want to see in PSCAD. And if your case is large, you can create a multiport equivalent of the remainder of the network.

3.2.   Dynamic Data Table

If your case has dynamic information, you can view it in the **Dynamic Data** pane. The table below shows dynamic data from the **PowerFactory/Kundur** example case.

4. SCHEMATIC DEVELOPMENT

Once you have imported your data into a PRSIM case you can develop the single line diagram (SLD) view. In the **Home** tab click the **Expand Bus** button or **Expand All** if your network is very small.
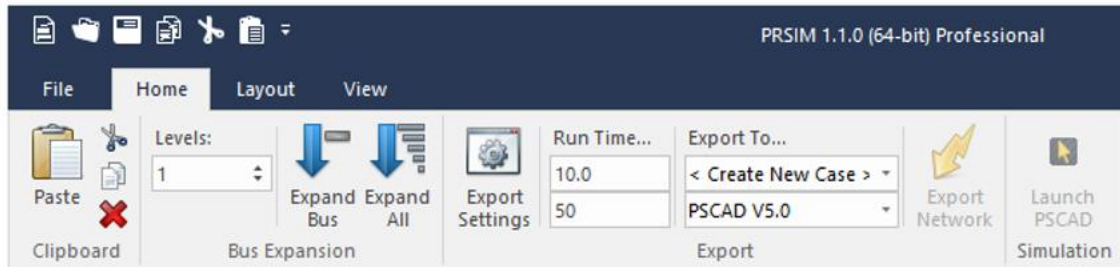
- **Expand Bus** will launch a pop-up window named **BusSelectionForm**, where you can select a start bus to begin your expansion. Additional filtering is available if you need to search for a bus in a large system.

- **Expand All** will attempt to expand the entire network into the active canvas view. Once a bus has been selected in the **BusSelectionForm**, click **Ok** to transfer it to the canvas and the network will be expanded until the canvas limits are reached or the network has been completely expanded. Don't worry, you can go ahead and if it fails or is too messy you can **Undo** or clear the canvas of the components you don't want. Removing components on the canvas will not remove them from the network.

- The **Levels** droplist can be used to expand any given bus outward by the radius amount.

Additional expansion options are also available in **Layout** tab.
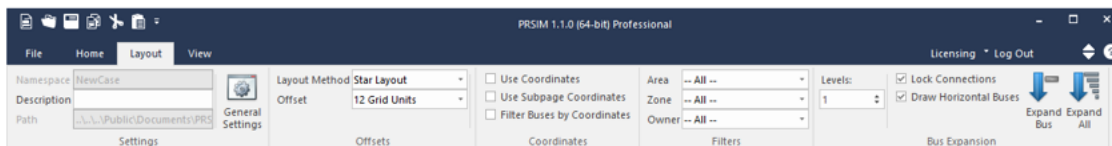
- **Layout Method**:
  - **Star Layout** expands bus up, up and left, up and right, down and left and down and right (in a star pattern).
  - **DropDown Layout** expands buses down and then to the right.

- **Use Coordinates**: If your case has a coordinate file or was exported from PowerFactory this option can accelerate the bus expansion as there is existing coordinate data for the bus locations.

- **Use Subpage Coordinates**: Enable this option if your case has subpage coordinates.

- **Filter Buses by Coordinates**: Enable this if your PowerFactory case has multiple diagrams.

- **Area, Zone, Owner**: You can filter the buses that get expanded by AREA, ZONE and OWNER.

- **Lock Connections** will ensure that bus and branch connections remain intact as you edit the schematic.

- Disable **Draw Horizontal Buses** if you want them oriented vertically. The default expansion direction will change to the right and then down.

Use the **Expand Bus** button to launch **the BusSelectionForm**, then you can select a start bus to begin the expansion process. Once you have a bus on the canvas you can continue the expansion by clicking on the ⊞ (plus icon) on the bus.
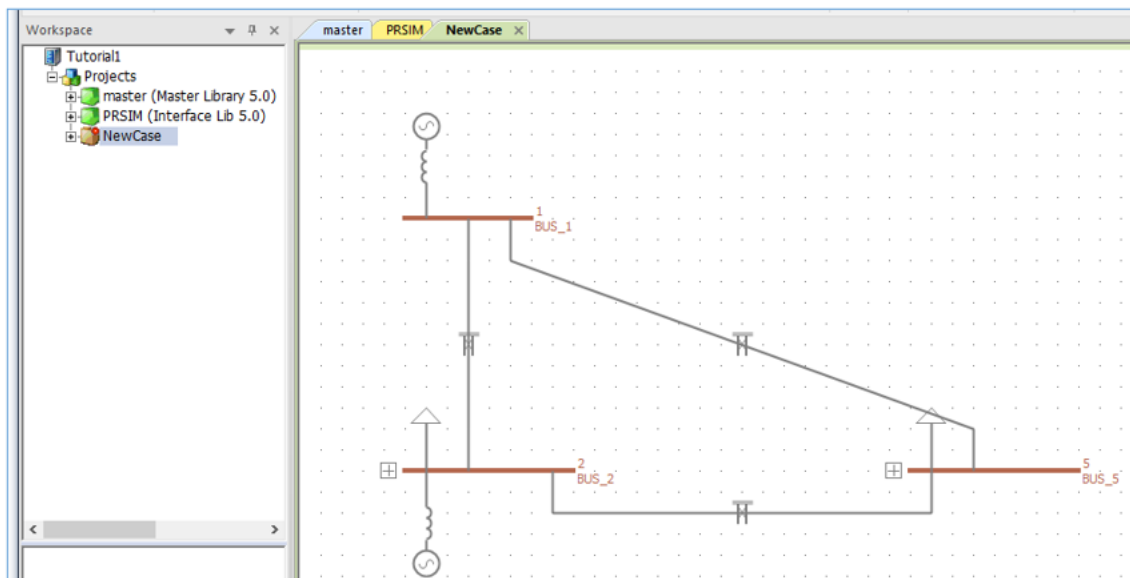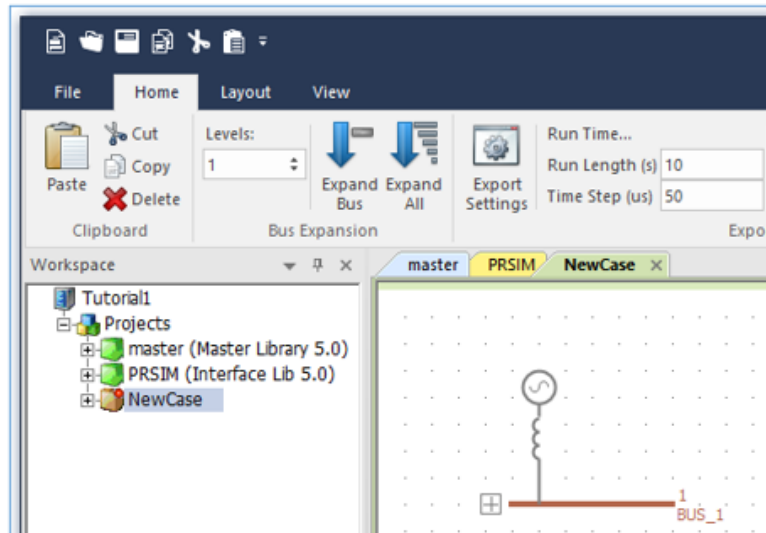
If your case has coordinate data and **Use Coordinates** is enabled, the buses will be automatically be positioned to their proper location on the PRSIM canvas. Otherwise, you can move and resize the buses to your liking. Buses with a lot of branches can be automatically sorted by clicking on them to gain focus and then use the S key to sort the branch end points.

5. PSCAD CONVERSION

Generating a PSCAD case may be done by selecting **Export Network** which will translate the schematic and PFD/PSSE information into a PSCAD EMT case. In this case, we'll use the default export settings. A detailed explanation for all available options under **Export Settings** will be provided in the following sections.

Clicking on **Export Network** button will launch the process to create a new PSCAD case and populate it with an equivalent PSCAD component for each network component in PRSIM. A script is run for each component to reads data from the network table and perform property computations which are then applied to the corresponding EMT components. Any remaining parts of the network that were excluded from PRSIM canvas will be represented with a network equivalent circuit on PSCAD side.
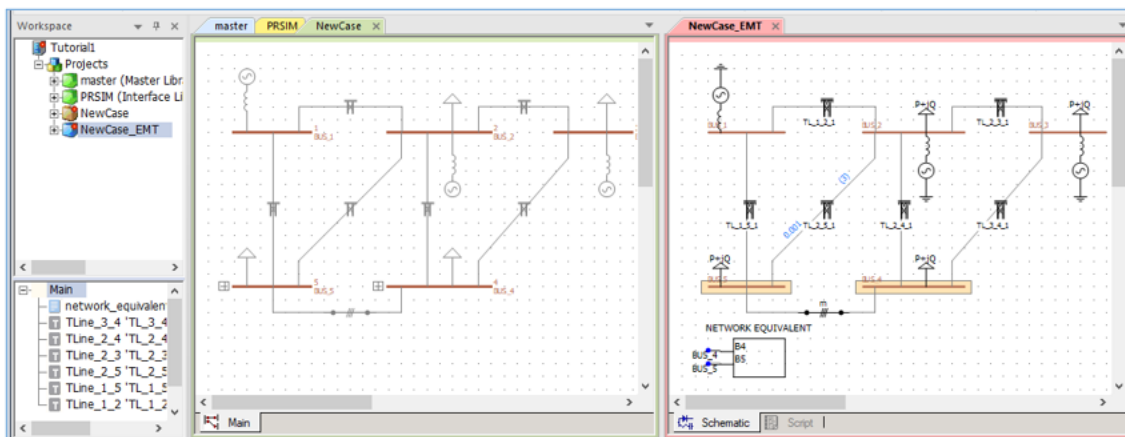
If you right-click on the PSCAD project and select **Show Script Computations**, you can review script computations that were performed for each component in the **Output** window.

```
================================================================================
Component: Line_ManualYZ
Identifier: 1078958325
Model type: AC_LINE
--------------------------------------------------------------------------------
Name               Old value            New value            RHS / Expression
--------------------------------------------------------------------------------
PU                 3                    3                    = 3
Estim              0                    1                    = 1
RPUP2              6.76e-8 [pu/m]       0.01938              = $R
XLPUP2             9.6e-7 [pu/m]        0.05917              = $X
BPUP2              1.73e-6 [pu/m]       0.0528               = $B
VR2                230 [kV]             138.0                = $BASKV_I
MVA2               100 [MVA]            100.0                = $SBASE
```
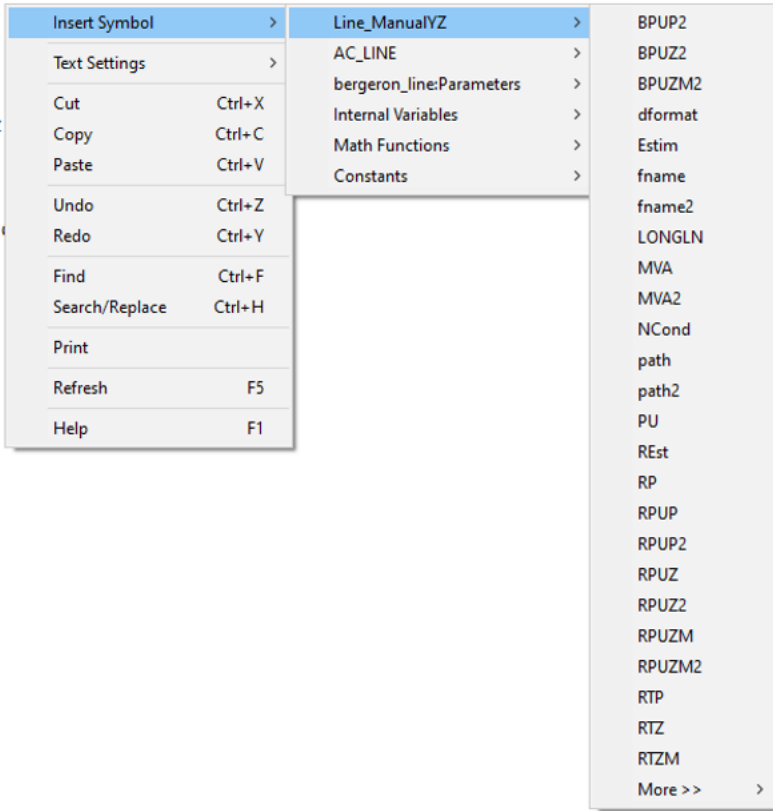
You can access and review the conversion scripts for each component by right-clicking on the model and selecting **Edit Scripts…** and then select the model displayed in the sub menu. You will see the **Convert-PSSE** or a **Convert-DGS** script in the script editor. The script for converting a PSSE **AC_LINE** is shown below. Hover the mouse over a variable to display tooltip information about it. If you right-click on the window the **Insert Symbol** menu contains component parameters as well as column names in the **AC_Line** table that can be used in the script equations.

The conversion scripts are computations sections intended for converting PRSIM component parameters to PSCAD component parameters. In this case, the target is the **Line_ManualYZ** component inside a PSCAD tline module.

When multiple definitions are needed for a component, they are selected using logic in the **Select-Model** script. The script shown to the right is used to select between a **fixed_load** in the master library and a **capacitive** or **inductive** source in the PRSIM library.

```
1  #IF ($FIX_LOADP == 1 .AND. $PL .LT. 0.0)
2      #IF($QL .LT. 0.0)
3          DEFINITION = PRSIM:source_3_C
4      #ELSE
5          DEFINITION = PRSIM:source_3_L
6      #ENDIF
7  #ELSE
8      DEFINITION = master:fixed_load
9  #ENDIF
```

6. MULTIPORT NETWORK EQUIVALENT

When a network is large or the area of interest is smaller than the total network you can create and use a schematic of the area you want to export and include a multiport network equivalent representing the rest of the network.

To do this, select **Export Settings** (or **Export Network**) under the **Home** tab as shown below.
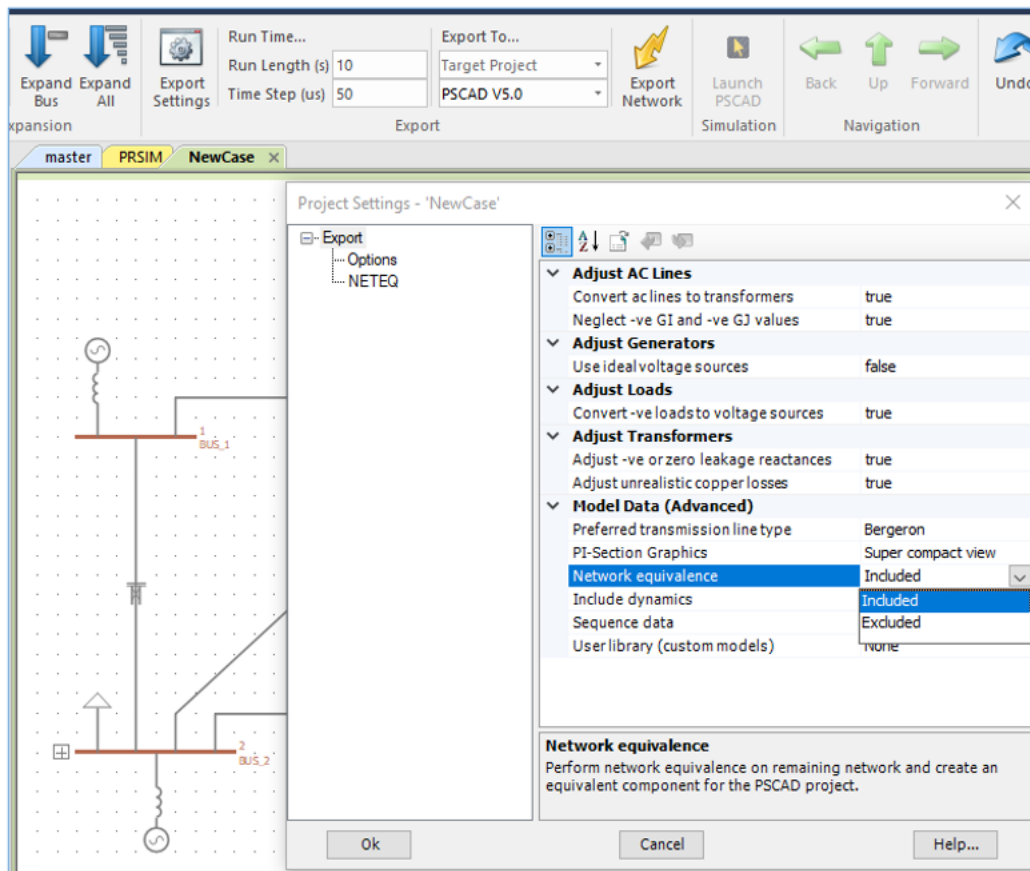

In the dialog you can verify that **Included** is selected in the **Export/Network Equivalence** dropdown list. If you haven't already done so, you can now generate a new PSCAD case by clicking on **Export Network**.

A network equivalent will appear as a module one or more ports representing the connection points from the network equivalence to the rest of the system at the boundary buses.
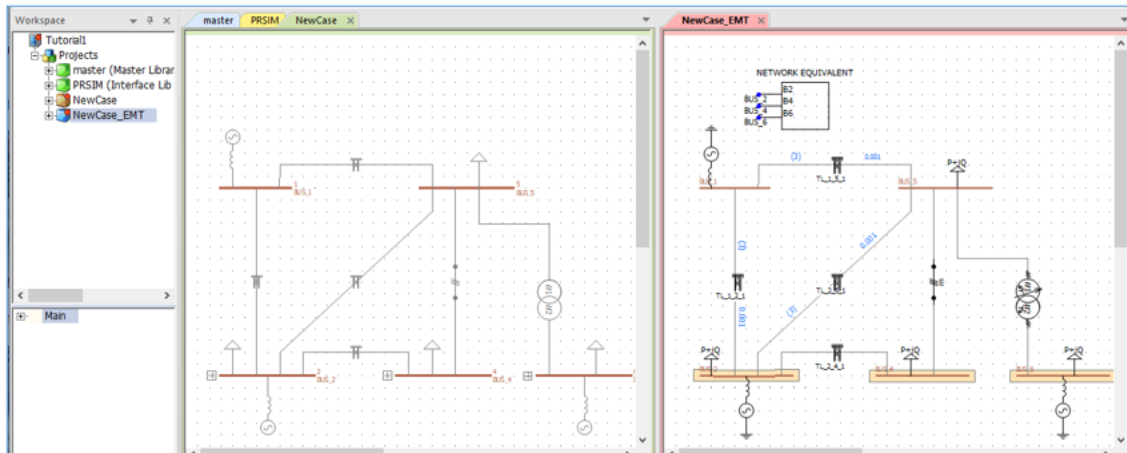
If the accuracy of generated Network Equivalent module is required, users may generate a test case during export. The option **Project Settings/Create NETEQ test module**, will generate a test case that you can use to verify the accuracy of Network Equivalent Module at each boundary bus.
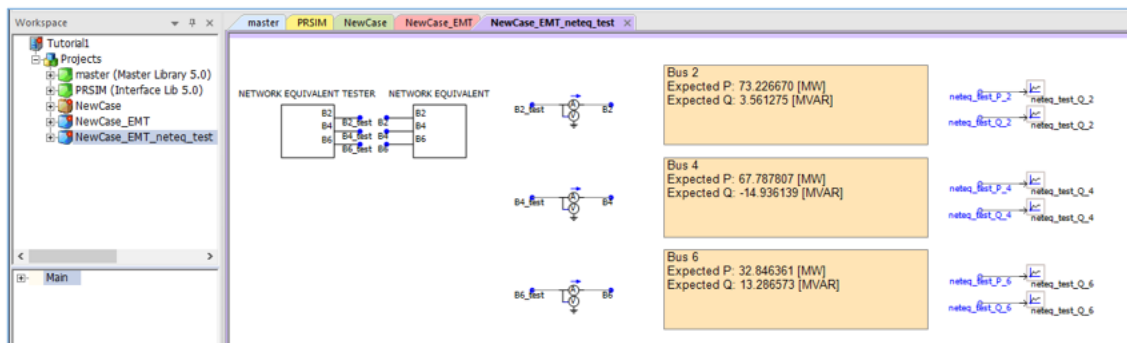
7. DYNAMIC DEVICES

PRSIM supports system dynamics provided your PowerFactory DGS or PSSE DYR network file has it. PowerFactory dynamics are automatically processed during the import process so all you need to do is verify it is there by examining the dynamics table. If you have a PSSE network, you will have to import the DYR file (see menus under the project tree) after you have imported the RAW file.
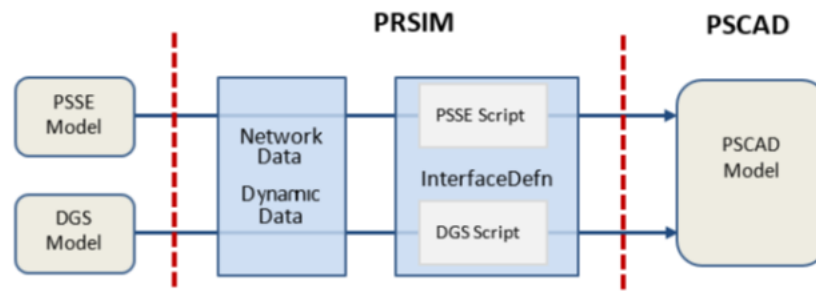
Applying system dynamics will replace network source models with detailed equivalent dynamic models which may be a combination of a synchronous machine, exciter, turbine, and governor as shown below.
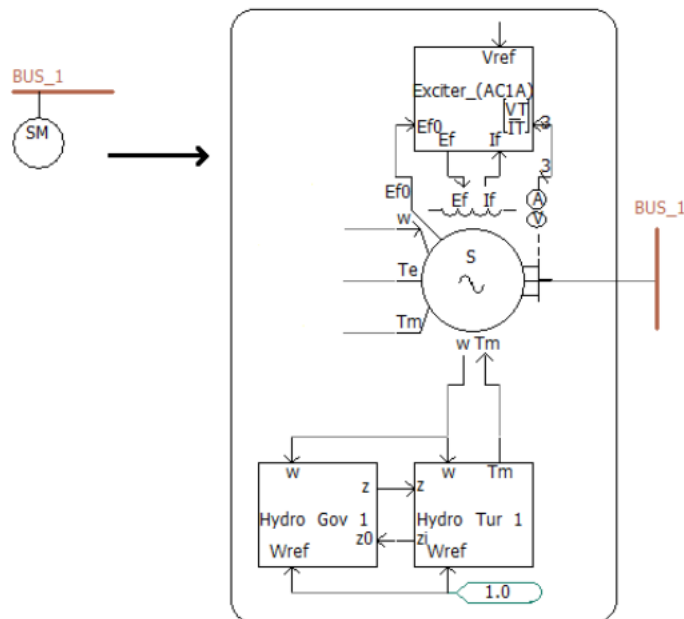
8. TRANSMISSION LINES

Two primary types of line models are included in PRSIM that are used in the generation of PSCAD transmission lines, **Pi-section** and **Bergeron line**. However, the **Selection** script shown below will override these if specific conditions are met.

```
#IF ($SHUNT == $I .OR. $SHUNT == $J)
    DEFINITION = master:fixed_load
#ELSEIF ($TLINE_IF == $I .OR. $TLINE_IF == $J)
    DEFINITION = master:tline_interface
#ELSEIF ($FIX_LINEKV == 1 .AND. ABS($BASKV_I - $BASKV_J) .GT. 1)
    DEFINITION = master:db_xfmr_3p2w
#ELSEIF ($dgs_elem .EQ. 9)
    DEFINITION = PRSIM:bergeron_line
#ELSEIF ($dgs_elem .EQ. 8 .OR. $dgs_elem .EQ. 7 .OR. $dgs_elem .EQ. 6)
    #IF ($DGS_TLINES .EQ. 1)
        DEFINITION = PRSIM:TowGeo_Line
    #ELSE
        DEFINITION = PRSIM:bergeron_line
    #ENDIF
#ELSEIF ($dgs_elem .EQ. 5)
    #IF (ABS($BASKV_I - $BASKV_J) .GT. 0.1)
        DEFINITION = master:db_xfmr_3p2w
    #ELSEIF (ABS($X) <= $TOL_ZEROIMP && $R < 1e-10)
        #IF ($B == 0)
            DEFINITION = PRSIM:Zero_Imp
        #ELSE
            DEFINITION = master:newpi
        #ENDIF
    #ELSE
        DEFINITION = master:newpi
    #ENDIF
#ELSEIF($dgs_elem .NE. 1 && $dgs_elem .NE. 6 && $dgs_elem .NE. 13)
    DEFINITION = master:rtcbranch
#ELSEIF ($BREAKER == 1)
    DEFINITION = master:switch
#ELSEIF (ABS($X) <= $TOL_ZEROIMP && $R < 1e-10)
    DEFINITION = PRSIM:Zero_Imp
#ELSEIF ($B == 0 && $X < 0.0)
    DEFINITION = master:rtcbranch
#ELSEIF ($B == 0)
    DEFINITION = master:coupled_3lines
#ELSEIF ($BERGERON == 0 || $R <= 0 || $X <= 0 || $B <= 0)
    DEFINITION = master:newpi
#ELSEIF (sqrt($X * $B) / (2 * PI * $BASFRQ) <= $TIMESTEP * 1e-6)
    DEFINITION = master:newpi
#ELSEIF ($r0_pu == 0 && $x0_pu == 0 && $bi0_pu == 0)
    DEFINITION = PRSIM:bergeron_line
#ELSEIF ($EXPORT_SEQ == 1 && ($r0_pu <= 0 || $x0_pu <= 0 || $bi0_pu <=
    DEFINITION = master:newpi
#ELSE
    DEFINITION = PRSIM:bergeron_line
#ENDIF
```

Note that you are not limited to the predefined models in the PRSIM library. The next section will show you how to specify on any other type of transmission line models and include them in the conversion process.

The **PI-section** is the first pre-defined transmission line model in the **Export Settings**. By selecting this type from the **AC-Line Model** section and then converting the network as shown below, all transmission lines will be replaced with **PI-section** equivalents regardless of their actual length.
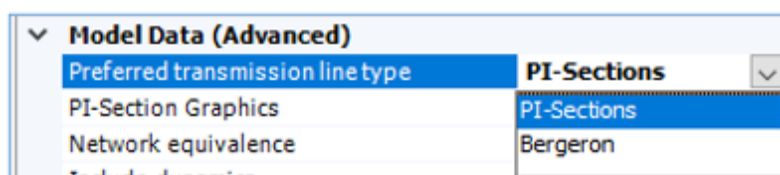
For the **IEEE14Bus** system, selecting **PI-Sections** will produce in the following PSCAD case where all the transmission lines are represented by their PI-section equivalent.
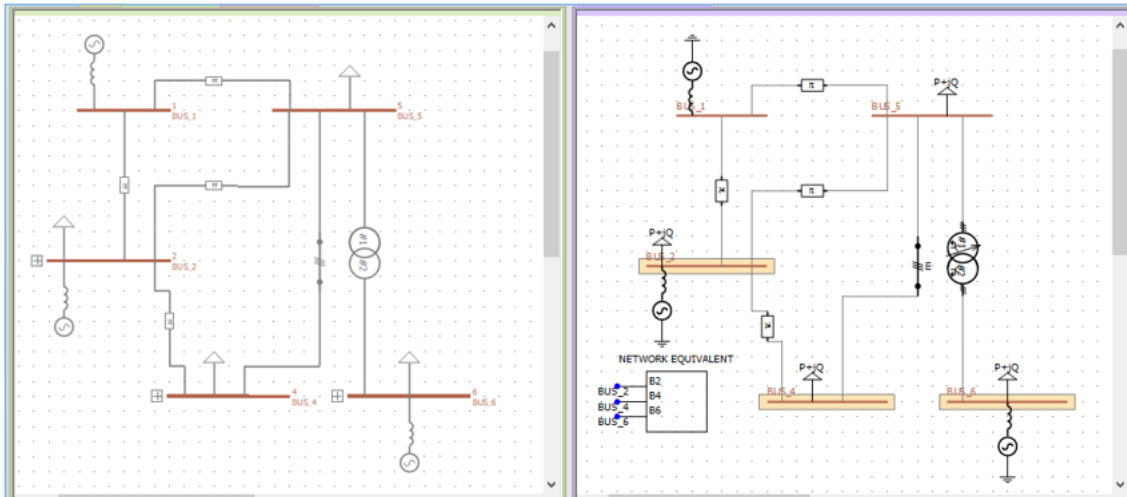
Bergeron T-Lines is the second option. If this option is selected, PRSIM will replace all long transmission lines with their Bergeron equivalents whenever the travel time of the line is greater than the selected simulation time step. Time step is accessible under **Runtime Settings** category in **Export Settings**. Otherwise, transmission lines will be replaced with **PI-sections** automatically.

Selecting **Bergeron T-Lines** for mapping the **IEEE14Bus** example to PSCAD will produce the following simulation case.
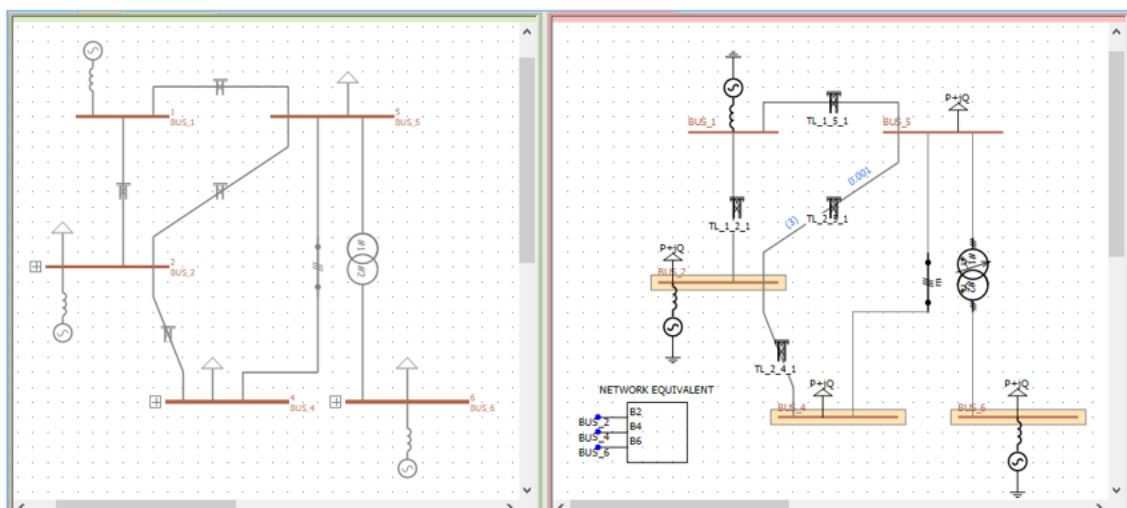
9. PSCAD USER LIBRARY

PRSIM provides a **User library** feature that will allow you to replace selected standard library components with your own custom models. Users can fully customize the models and scripts that calculate the target component parameters. The custom models may include detailed page module components as well as black box components. One application of the **User library** feature is to replace generators with dynamic models including exciters, governors, and power system stabilizer components. Another application is considered here to replace transmission lines with detailed models that include tower geometry and conductor information.

The **User library** can be used in three modes:

- **Global substitution**
- **Single component substitution**
- **Local name substitution**

9.1. Global Substitution

This feature substitutes all of the components from a user-defined PSCAD library. The substitution library that you developed must first be loaded into PRSIM workspace tree. Following that you may optionally set it as a **start up** library in the **Default Models** dialog under **File** tab. Once there, PRSIM will always load your library along with the master and PRSIM libraries.
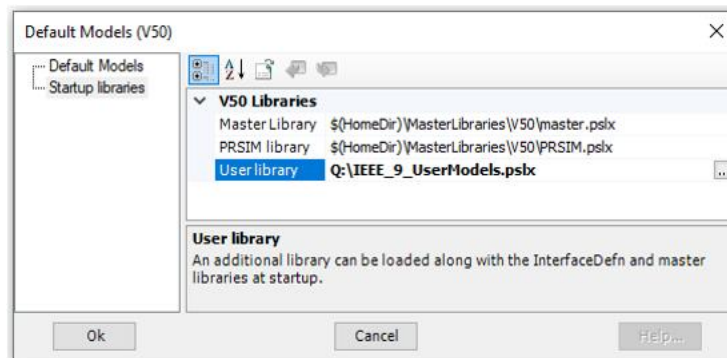

You can enable your new custom library in your project by selecting it in the **Project Settings** dialog.
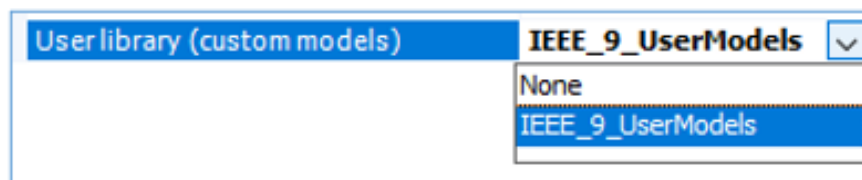
During the export process for the specified project, PRSIM will first search the user library for any matching Interface definitions. If found, your custom definition will be used otherwise the process will revert to the default definition for that branch type located in the PRSIM library.

9.2. Single Component Substitution

Single component substitution is used when a custom model is needed as a replacement for a specific component in the schematic. Select the properties menu on a specific branch component on the schematic then use the droplist to select an alternate model.
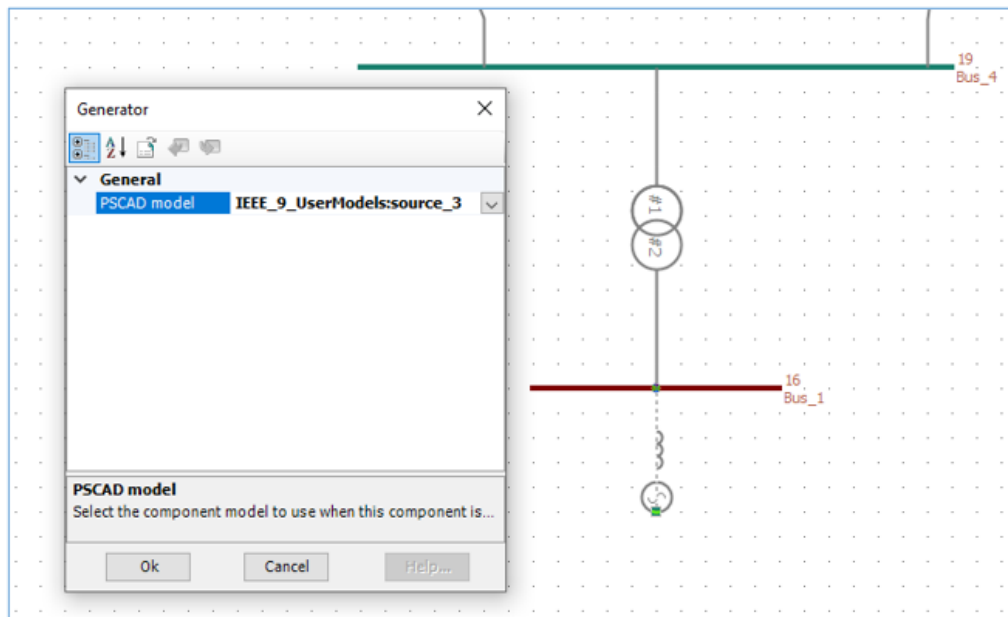
## 9.3.   Local Name Identification and Substitution

Local name substitution is an alternative method for specifying custom components. This approach requires the user to name their definitions according to **PowerFactory** element names (**loc_name**) or using our standard naming convention for **PSSE** networks. PRSIM will automatically select the components that conform to these local naming conventions.
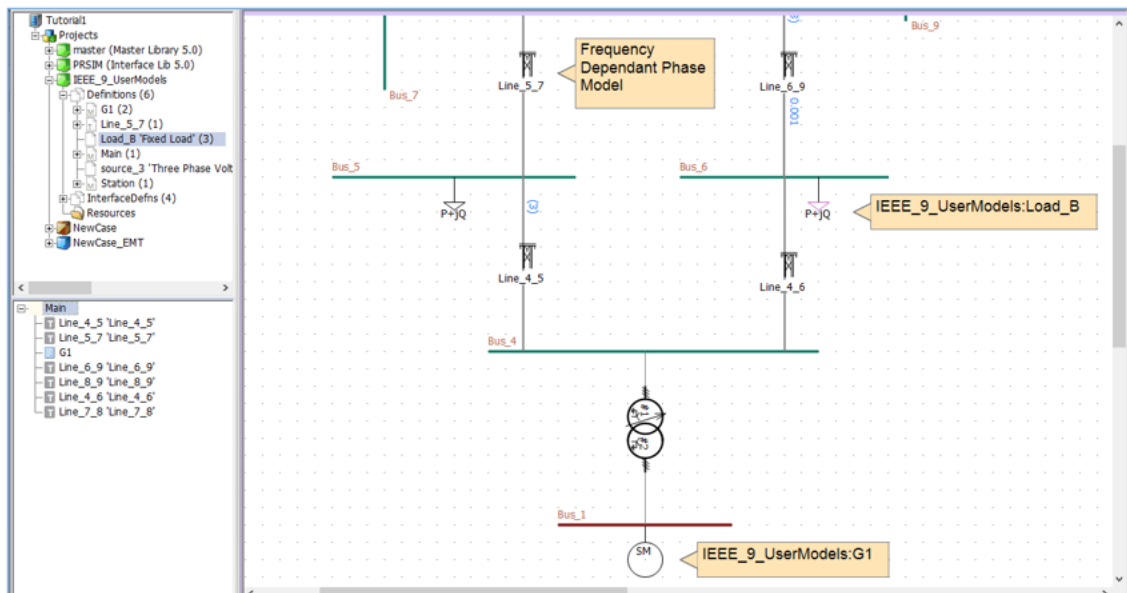
10. RUNNING PSCAD CASE

To run the generated PSCAD file simply click the **Launch PSCAD** on the ribbon. PRSIM will launch the PSCAD version that is currently active in PRSIM.

- If you have more than one V46 and V50 PSCAD version, you can select your preferred version in the Application Options/PSCAD Executables.

- PRSIM will attach any dependant libraries to your case when it launches PSCAD
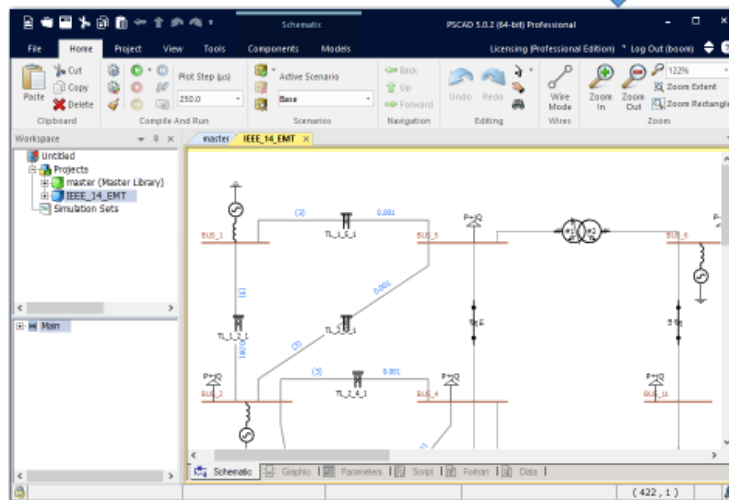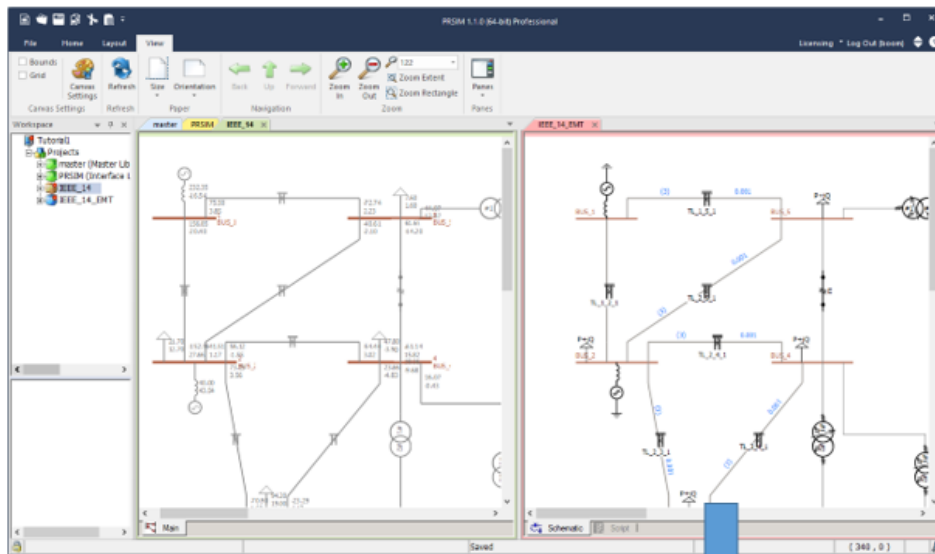
DOCUMENT TRACKING

Rev.	Description	Date
0	Initial	04/01/2023