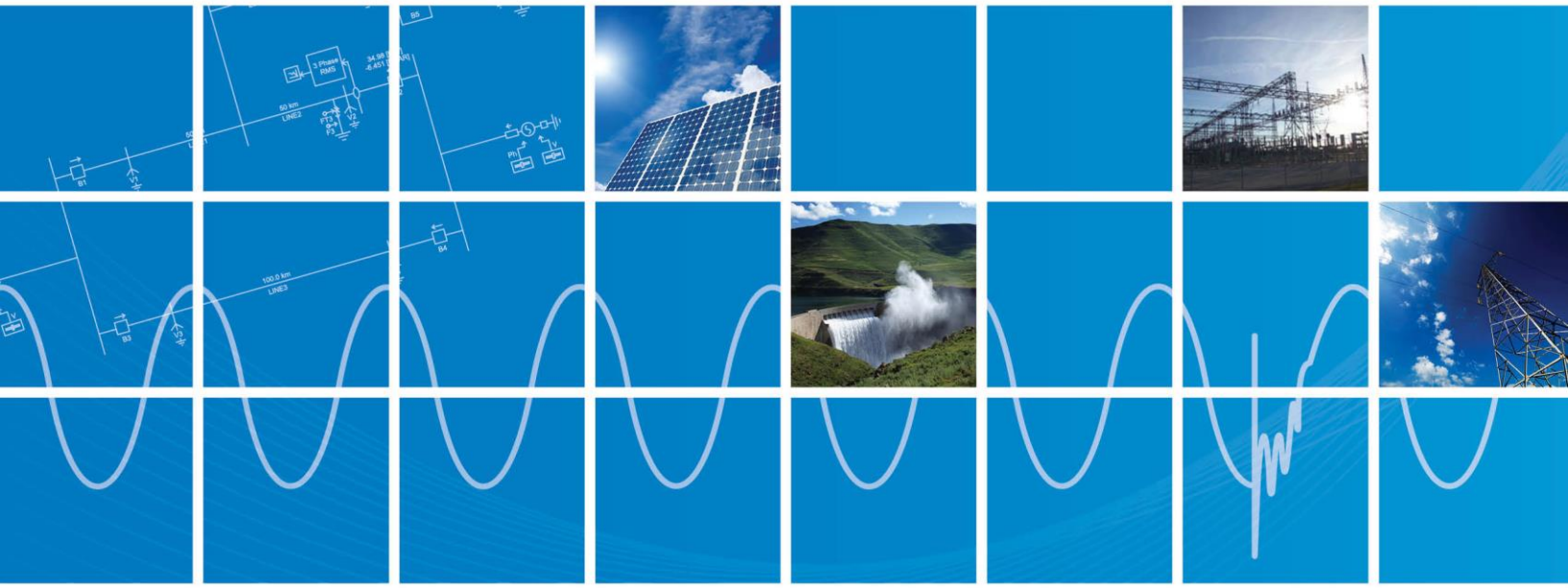
PSCAD - Resolving Launching, Compiling, and Running Issues

Written for PSCAD X4

November 30, 2020
Revision 73



Powered by Manitoba Hydro International Ltd.
211 Commerce Drive
Winnipeg, Manitoba
R3P 1A3 Canada
mhi.ca

Contents

1. Introduction

1.1 Overview

This manual presents known issues and solutions related to PSCAD and associated software. It is intended to be a continuously developing diagnostic tool as the software evolves.

The information in this manual is applicable to PSCAD X4 and V5, which includes version 4.3.0 and newer, and covers the following topics:

- Section 2: Launching PSCAD (Section 2)
- *Section 3: Content (lock-based licensing issues) moved to new manual "Resolving PSCAD Lock-Based Licensing Issues" (see Section 1.2)*
- *Section 4: Content (certificate licensing issues) moved to new manual "Resolving Certificate Licensing Issues" (see Section 1.2)*
- *Section 5: Content (Issues when Licensing PSCAD – Legacy Lockless) moved to new manual "Resolving Lock-based Licensing Issues" (see Section 1.2)*
- Section 6: Using PSCAD
- Section 7: Compiling Cases in PSCAD
- Section 8: Running Cases in PSCAD
- *Section 9: Content (MyCentre issues) moved to new manual "Resolving MyCentre Issues" (see Section 1.2)*
- *Section 10: Content (diagnostic tools issues) moved to new manual "Resolving PSCAD Lock-Based Licensing Issues" (see Section 1.2)*
- Section 11: Resolving FORTRAN coding issues
- Appendix A: Using the Fortran Medic utility
- *Appendix B: Content (using the Get Info utility) moved to new manual "Resolving PSCAD Lock-Based Licensing Issues."*
- *Appendix C: Content (requirements for the License Manager - legacy lock-based licensing) moved to new manual "Resolving PSCAD Lock-Based Licensing Issues."*
- *Appendix D: Content (requirements for connecting to the license server) moved to new manual "Certificate Licensing Issues" (see Section 1.2)*
- Appendix E: Instructions for displaying hidden folders
- Appendix F: Listing of files created when a PSCAD project is run
- *Appendix G: Content (testing certificate licensing connectivity) moved to new manual "Certificate Licensing Issues" (see Section 1.2)*

1.2 Related Support Documents

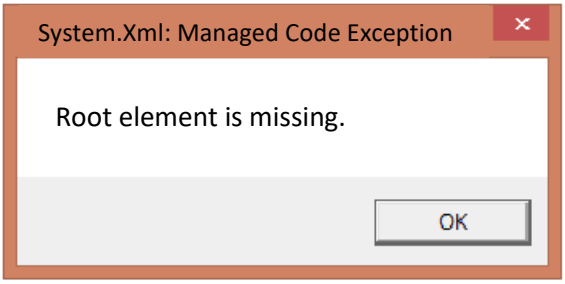
Refer to this article for other, related, support documents.

2. Issues when Launching PSCAD

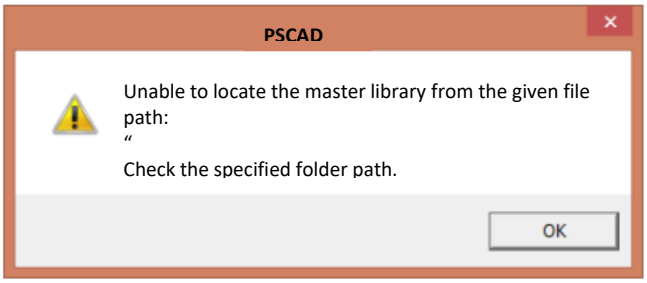
2.1 Receiving a "Root element is missing" Error Message

Problem

When launching PSCAD, the following error message displays:



Clicking the "OK" button displays the following error message:



Clicking the "OK" button closes PSCAD.

System

- PSCAD v4.3
- PSCAD v4.5.1
- Windows XP (32-bit) Service Pack 3

Cause

A problem with the PSCAD user_profile.xml file.

Solution 1

Update to PSCAD v4.5.3 or later. Ensure to log out then log back in on your machine to apply the changes.

Solution 2

Fix this using the Fortran Medic utility:

- Run the utility as per Appendix A.
- Once the utility has retrieved the information, scroll down to the following message:

  Installed PSCAD versions
  PSCAD X4 Release (4.5.1 (x86)) [Release date: 2012.12.13 16.36.04]

  Workspace file: C:\Documents and Settings\Username\Local Settings\Application Data\Manitoba HVDC Research Centre\PSCAD\user_profile.xml
  File exists but appears to be empty

- Right-click on the "Workspace file" message and select the option to delete this file.

2.2 The PSCAD Launch is Prolonged

Problem

When PSCAD is launched, it may take up to several minutes before the application is ready for user input. Specifically, when PSCAD is launched, the PSCAD splash window remains displayed for an extended time, during which the application is neither licensed nor usable.

System

- PSCAD v4.5.1 to v4.5.3
- All Windows platforms
- Using either the “elevated” or “non-elevated” launch links in the Windows Start menu.

Cause

The PSCAD Start Page cannot load, and is delaying the start of the application.

Solution 1

Update to PSCAD v4.5.4 or later.

Solution 2

Fix this using the Fortran Medic utility:

- Run the utility as per Appendix A.
- Once the utility has retrieved the information, scroll down to the following error message:

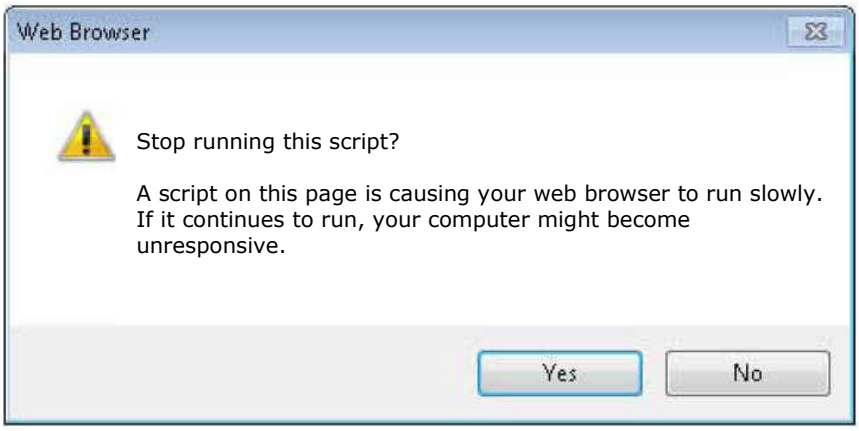
  Conflicts
  PSCAD will not display the latest Start Page.

- Right-click on the error message and select the option to repair it.

2.3 Receiving a "Stop running this script?" Error Message

Problem

When PSCAD is launched, the following error message displays:



System

- PSCAD v4.5.1 to v4.5.3
- All Windows platforms

Cause

The PSCAD Start Page cannot load, and is delaying the start of the application.

Solution 1

Update to PSCAD v4.5.4 or later.

Solution 2

Fix this using the Fortran Medic utility:

- Run the utility as per Appendix A.
- Once the utility has retrieved the information, scroll down to the following error message:

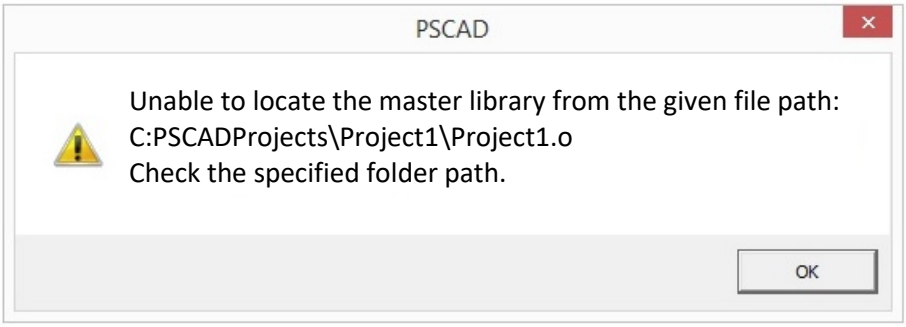
  Conflicts
  PSCAD will not display the latest Start Page.

- Right-click on the error message and select the option to repair it.

2.4 Receiving an "Unable to locate the master library from the given file path" Error Message

Problem

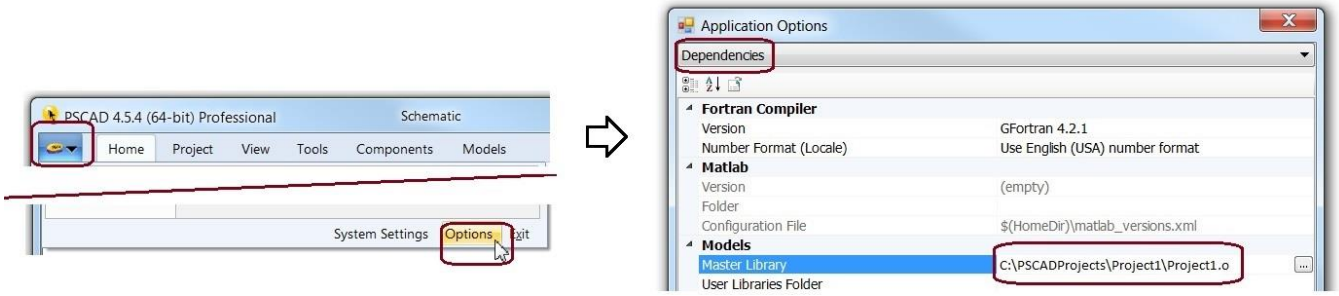
When launching PSCAD, a message similar to the following is displayed:



PSCAD might crash when "OK" is selected.

Cause

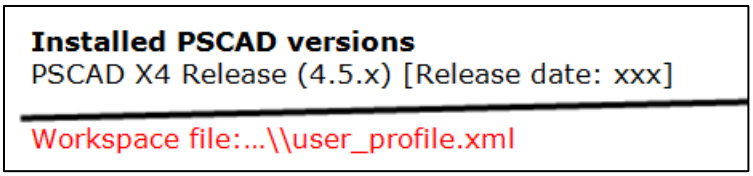
An incorrect file type (for example, .o) was entered into a field reserved only for library files (.pslx). See location of this library file field, below.



Solution

(1) **If PSCAD crashed:** Delete your PSCAD user profile to restore the master library, as follows:

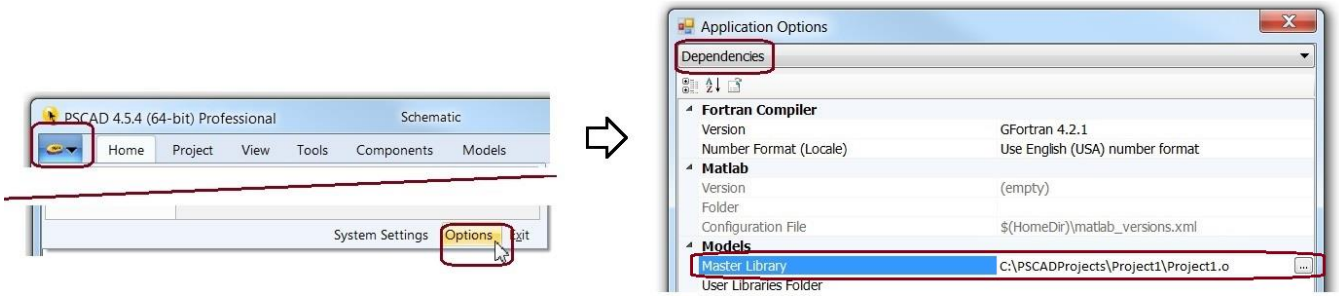
- Run the Fortran Medic utility (per Appendix A1).
- Once the utility has finished retrieving information, scroll down to the PSCAD version number, and note the path for the "Workspace file".



- Open a browser to this file path, and delete the user_profile.xml file.
- Proceed to Step (3) below.

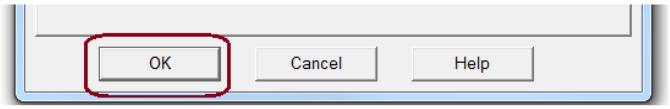
(2) **If PSCAD did not crash**: Restore the master library in the PSCAD application as follows:

- Display the Master Library field:



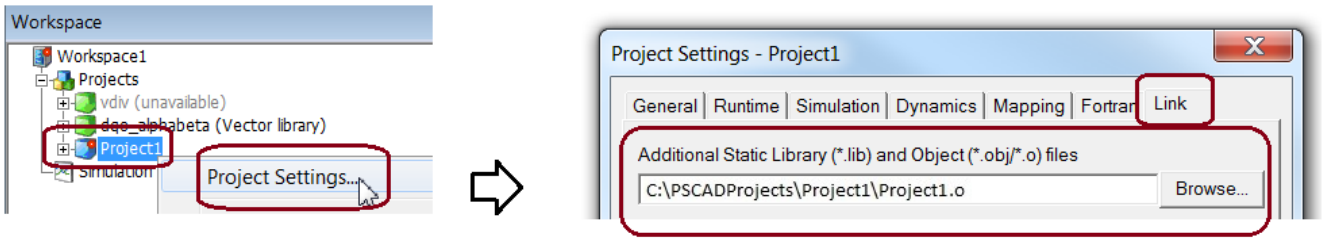
- Delete the text text in this field, and enter the following: $(HomeDir)\master.pslx

- Select "OK" to apply the change:



- Proceed to Step (3).

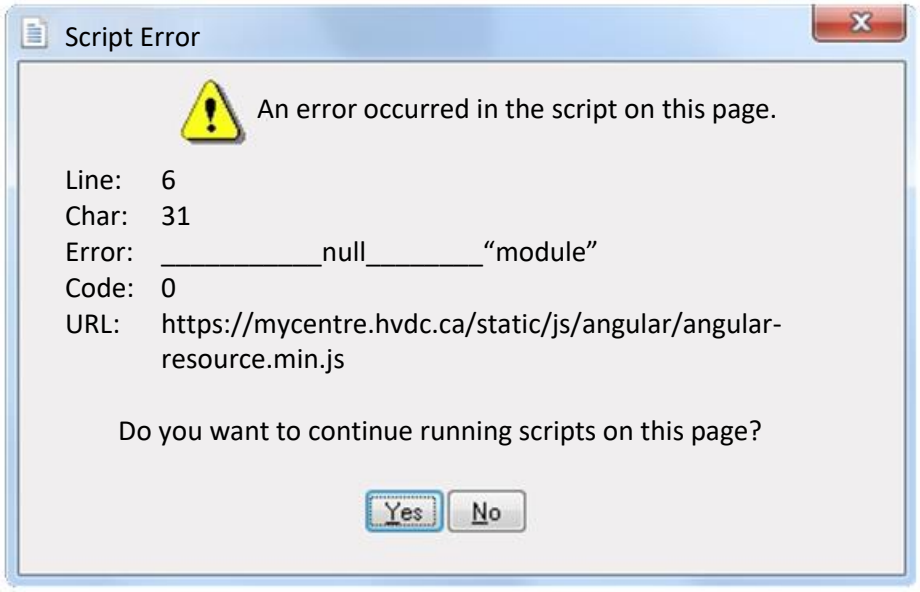
(3) Link your file in PSCAD through the Project Settings dialog box, as shown:



2.5 Receiving a "Script Error" Error Message

Problem

When launching PSCAD, a message similar to the following is displayed:



Cause

The PSCAD "Start" page, which has links to YouTube, is being blocked. Possible causes are firewalls and anti-virus. These blocks might be applied locally on the user's computer, or more broadly across the network of an organization or even a country.

> **Note**
> The Start page is a tab in PSCAD containing the same information and help videos that are available in MyCentre (https://mycentre.hvdc.ca/).

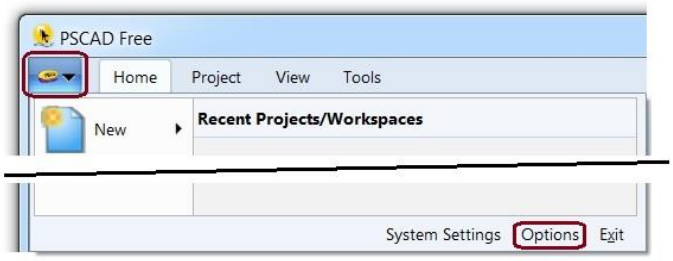
Solution 1

Turn off the software that is blocking the start page (firewall or anti-virus).

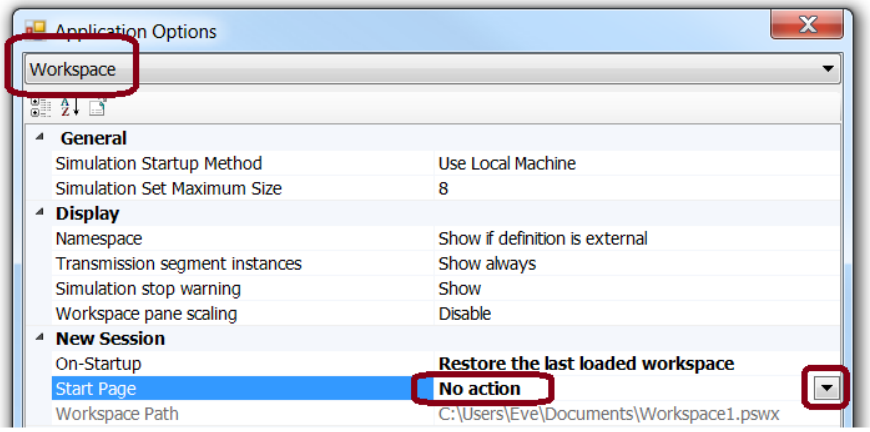
Solution 2

Disable the Start Page so that it does not attempt to load upon launching PSCAD, as follows:

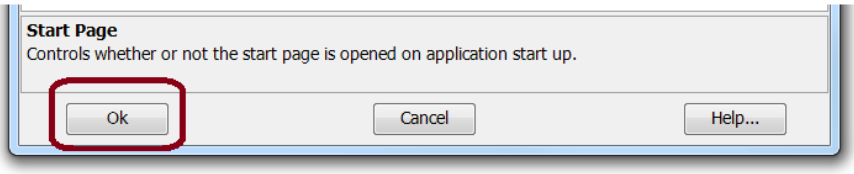
- Launch PSCAD.
- Display the Application Options dialog box:



- In the Workspace page, select “No action” from the “Start Page” drop-down button:



- Select “Ok” to apply the change:

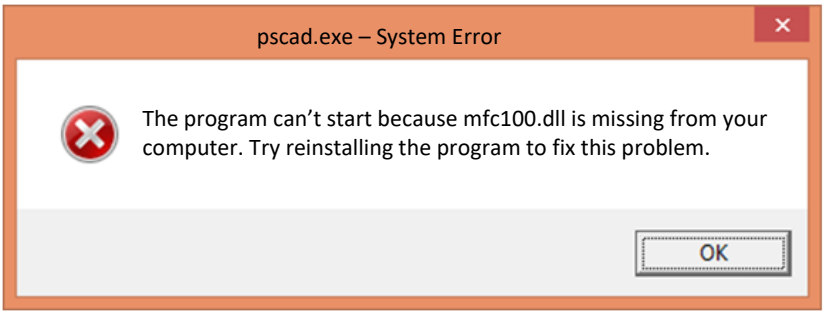


- The Start Page will not attempt to display upon PSCAD startup.

2.6 Receiving an "mfc100.dll is missing" Error Message

Problem

When PSCAD is launched, the following error message is displayed:



Cause

This message displays because the Visual C++ 2010 SP1 Redistributables is missing.

Solution

Use the Fortran Medic utility to install the Visual C++ 2010 SP1 Redistributables, as follows:

- Run the latest Fortran Medic utility as per Appendix A.2.
- **For a Windows 32-bit machine:** Select Help | Install Visual C++ 2010 SP1 Redistributables (x86).
- **For a Windows 64-bit machine:** Select Help | Install Visual C++ 2010 SP1 Redistributables (x64).

2.7 Receiving a "The procedure entry point … dynamic link library SHELL32.dl" Error Message

Problem

When PSCAD v4.6.0 or later is launched, an error message similar to the following is displayed:

Cause

This message displays because the user is trying to run PSCAD on an unsupported operating system (Windows XP).

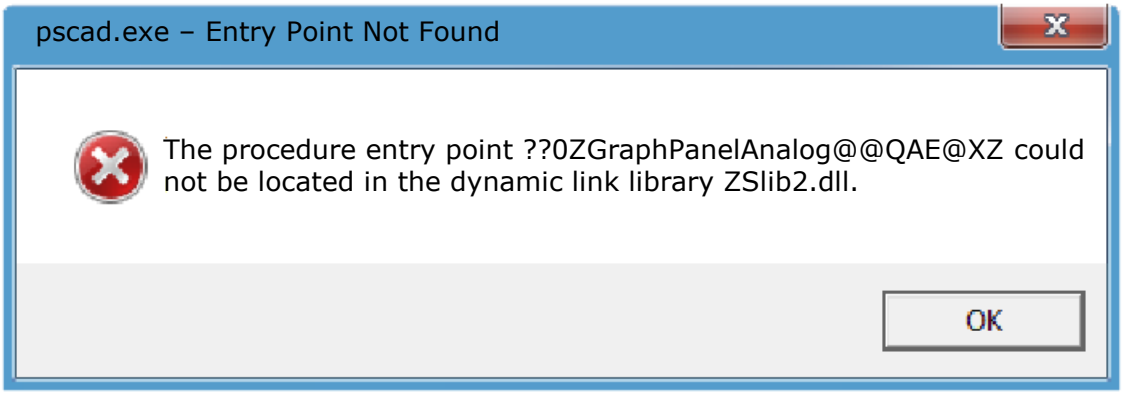
Solution

PSCAD v4.6.0 and later is not supported on Windows XP. This software is officially supported on Windows Vista and Windows 7, but has also been shown to work on Windows 8.

2.8 Receiving a "The procedure entry point …could not be located in the dynamic link library ZSlib2.dll" Error Message

Problem

When PSCAD is launched, an error message similar to the following is displayed:



Cause

A problem has developed with the PSCAD program files.

Solution

a. Delete the PSCAD program folder and all subfolders and files. These will typically be located in a path similar to the following:
   - C:\Program Files (x86)\PSCADx

b. Re-install or repair PSCAD. This will re-install all the required files.

For Additional Information

See Appendix A.5, Item 10.

2.9 PSCAD crashes upon startup

Problem

PSCAD crashes upon startup.

System

Most or all versions of PSCAD.

Cause

The corporate security package, Digital Guardian, can cause PSCAD to crash after .NET 4.6.1 is installed.

Solution

Whitelist PSCAD in the corporate security package.

2.10 The PSCAD BETA Edition crashes upon startup

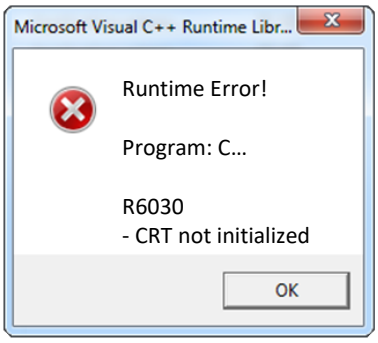
Problem

When launching the PSCAD Beta Edition from MyUpdater, the launch fails:

- The following error message displays in the MyUpdater messages tab:

  Process terminated with error: E0434352

- The following dialog message displays:



System

PSCAD Beta Edition on Windows 7 SP1.

Cause 1

This problem is most often caused by certain security software programs. It is also possible to get this error when another program uses code injection techniques to trap certain DLL library calls. Some intrusive security programs use this technique. In versions of Visual C++ before Visual Studio 2015, it is possible to use a statically-linked CRT library to address the issue, but this is not recommended for reasons of security and application updates.

Solutions 1

1. Disable your security software and see if this resolves the issue.
2. Your security software may have specific instructions for mitigating this issue. Check your security software vendor's website for details.
3. Check for updated versions of your security software, or try different security software.
4. Re-install the Visual C++ Redistributables, then reboot your machine, and then try launching the Beta Edition again.
5. Try this installation on a different machine.

Cause 2

You may have corrupt Windows files.

Solutions 2

Have your IT staff perform an **sfc** scan according to the following instructions:

http://support.microsoft.com/kb/929833

If the above step fails to fix the problem, try repairing Windows.

2.11 PSCAD v4.6.0 or v4.6.1 64-bit does not launch

Problem

When launching PSCAD v4.6.0 or v4.6.1, the launch fails.

> **Note**
> PSCAD 32-bit does run.

Cause

There is a problem with the .NET framework installation.

Solution

Resolve the .NET Framework software installation (e.g. re-install fresh, or install the patches/updates).

Or, run PSCAD on a different machine.

2.12 Receiving a "The computer must be trusted for delegation…" error message

Problem

When launching PSCAD, the launch fails, and the following messages are displayed:

> Application.ThreadException
> Type: System.Exception
> Invoking Type: CredentialManagerLib.UCredentialManager
> Method: Void initialize
> Cause: The requested operation cannot be completed. The computer must be trusted for delegation and the current user account must be configured to allow delegation.

Cause

"Mandatory profiles" are being used on the machine.

Mandatory profiles, which are essentially read-only profiles, do not support certain cryptographic functions which are essential to the applications.

Specifically, these functions are essential for the following tasks:

- To create a new user key;
- To use functions which are used by PSCAD and the MyUpdater to cache and store the user's MyCentre credentials, which are required by the following:
  - By the MyUpdater utility to log into the user's MyCentre account to retrieve the list of available products, and
  - By PSCAD to log into the user's MyCentre account to obtain a new PSCAD license certificate.
- Currently, this also appears to be an issue even if the client would like to use the legacy lock-based License Manager with PSCAD.

The latest Fortran Medic tool indicates if a user's profile is mandatory. Refer to Appendix A.5, Item 29.

Applicable to

Will likely impact the following:

- PSCAD v4.5.4 to v4.6.2 and the Free Edition
- Enerplot, FACE, Initializer, Update Client (MyUpdater)

Solutions

- Allow users to use non-mandatory profiles.

  Note
  PSCAD 4.6.2 is designed to not require local admin rights, and should be useable by normal users, however, this is also dependent on certain Windows 10 Group Policy settings.

- Allow the PSCAD users to log in to local (non-domain) accounts on the machines hosting PSCAD.

2.13 How to launch PSCAD without Windows Administrator Privileges

Objective

For some facilities, users are not provided Windows Administrator privileges, and therefore must be able to launch PSCAD with Windows Users privileges.

Solution (1)

Pin a non-elevated PSCAD shortcut to the Windows Taskbar, Windows Start menu, or to the desktop. For example, for PSCAD v4.6.2:

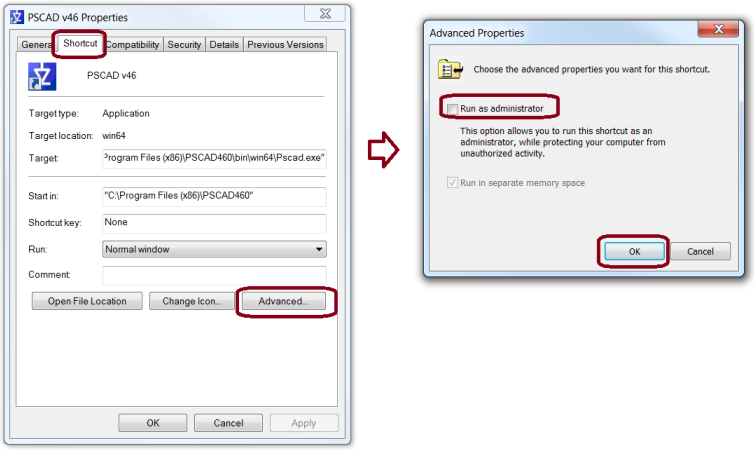
- Open a Windows file browser to here:

  C:\Program Files (x86)\PSCAD46\bin\win64

- Right-click on “pscad.exe” file and select “Properties”. Review the properties, and ensure that this link is non-elevated.
- Right-click on “pscad.exe”, and select the required action:
  - “Pin to Taskbar”, or
  - “Pin to Start Menu”, or
  - “Send to”…”Desktop”
- Always launch PSCAD using the new shortcut.

Solution (2)

De-elevate the existing PSCAD launching shortcut on the Windows Taskbar. Right-click on the shortcut, select “Properties”, and perform as shown below:



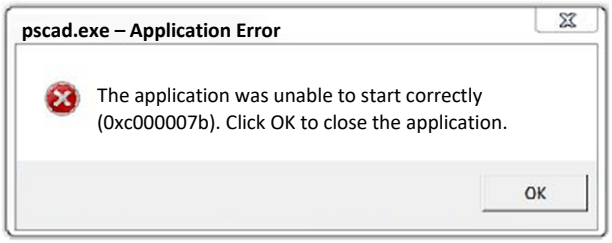
Solution (3)

If the Solutions (1) and (2) do not work, one suggestion is to use the Microsoft Application Compatibility Toolkit to create a custom security DB, and apply the DB to the PSCAD application. This should then force the application to run in the context of the person trying to launch the application.

2.14 Receiving a "The application was unable to start correctly (0xc00007b)" Error Message

Problem

When launching PSCAD, the following error displays:



System

This was detected when trying to launch PSCAD v4.6.2 with the following setup:

- Windows 7 SP1 Operating System
- Visual C++ 2015 Redistributable (x64) (14.0.24215.1)

Cause

Microsoft Visual C++ 2015 Redistributables is corrupted.

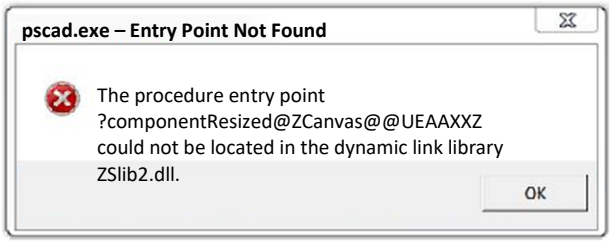
Solution

Re-install Microsoft Visual C++ 2015 Redistributables.

2.15 Receiving an "Entry Point Not Found" Error Message

Problem

When launching PSCAD, the following error displays:



Cause

The PSCAD installation became corrupted because PSCAD v4.6.1 was installed first, and then PSCAD v4.6.0 was allowed to be installed.

Solution

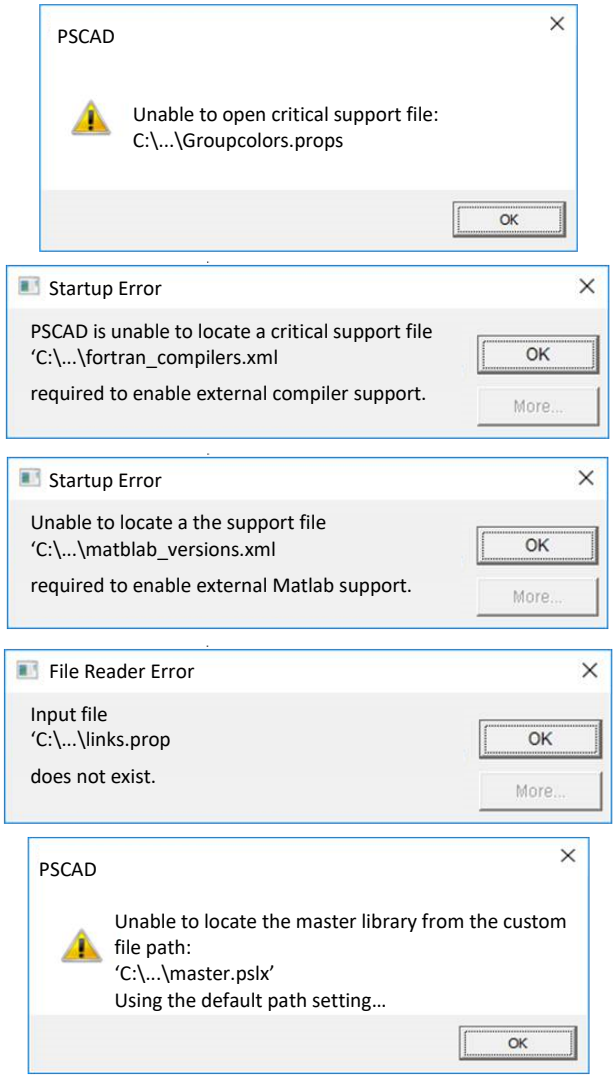
Uninstall all versions of PSCAD v4.6.*, then install v4.6.1.

> **Note**
> It is possible to install multiple versions within the same branch on a machine, for example, v4.6.0 and v4.6.1.
>
> Normally, a version within a branch (i.e. "patch") will overwrite the previous version.
>
> To circumvent this, install the latest version first (e.g. v4.6.1), ensuring to save the files to a new program folder when prompted (e.g. "PSCAD v461") rather than to the default folder ("PSCAD46"). Next, install the earlier version (e.g. v4.6.0), ensuring to save the files to another new program folder when prompted (e.g. "PSCAD v460").

2.16 Receiving Missing Critical File Messages when trying to launch PSCAD

Problem

When launching PSCAD, errors similar to the following display:



Cause (1)

The PSCAD software was installed on a machine protected by Beyond Trust, which causes the software to not fully install, especially the required PSCAD registry keys.

Solution (1)

Contact the Beyond Trust support staff for assistance.

Continued…

Cause (2)

Possibly, the PSCAD installation could have been corrupt, or anti-virus software may have corrupted the program files.

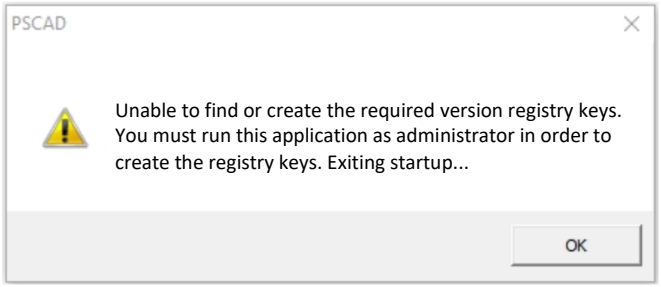
Solution (2)

- If it is suspected that anti-virus software corrupted your program files, turn this off temporarily.
- If it suspected that the PSCAD installation files were corrupted, obtain a new, trusted download of PSCAD (sales@pscad.com).
- Uninstall PSCAD.
- Delete any desktop shortcuts to PSCAD.
- Re-install PSCAD.
- PSCAD should be able to run.

2.17 Receiving an "Unable to find or create the required version registry keys" Error Message

Problem

When trying to launch PSCAD, the launch fails, and the following error displays:



Cause

The Windows Registry keys for PSCAD are not permitted to be modified. Therefore, when PSCAD is launched, and attempts to modify the Registry Keys, this fails, and PSCAD cannot be run.

These Windows Registry keys for PSCAD are listed in the requirements document, which may be viewed from this webpage.

Solution (1) Preferred – Provide Permissions for Modifying the Windows Registry Keys for PSCAD

Configure full permissions for the Windows Registry Keys for PSCAD. The corresponding keys are listed in the above webpage. PSCAD may then be launched, PSCAD will modify the Registry Keys, and then can be run.

Solution (2) Run PSCAD Initially with Windows Administrator Privileges

Launch PSCAD with Windows Administrator privileges (from the Windows Start menu, browse to the PSCAD XX link, right-click and select Run as administrator, then select Yes when prompted by the Windows UAC). PSCAD should be able to create the those registry points, and thereafter, PSCAD should be able to be launched as a"normal" Windows User.

If the Matter Persists

Please download and run our latest Fortran Medic utility, and send in the generated log file to the MHI support desk as per these instructions.

3. Issues when Licensing PSCAD – Legacy Lock-Based

Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues"

See Section 1.2

4. Issues when Licensing PSCAD – Certificate-Based

Content moved to new manual "Resolving Certificate Licensing Issues"
See Section 1.2

5. Issues when Licensing PSCAD – Legacy Lockless

Content moved to new manual "Resolving Lock-Based Licensing Issues"
See Section 1.2

6. Issues when Using PSCAD

6.1 PSCAD Crashes when the "Associations" tab is selected

Problem

PSCAD crashes when the "Associations" tab is selected in the "System Settings" dialog.

System

PSCAD v4.5.1

Cause

PSCAD v4.5.1 crashes due to a missing file: "external_tools.xml".

Solution 1

Use the Fortran Medic utility to repair the problem:

a. Download the latest "FortranMedic" from our website:

http://updater.pscad.com/utilities/FortranMedic.zip

b. In the downloaded .zip file, run the FortranMedic.exe file.

c. Click on the "Actions" menu and select "Start".

d. After the utility is done retrieving information:

- Scroll down to: Installed PSCAD versions | PSCAD 4.5.1.
- Locate the error in red text related to the missing file: "external_tools.xml".
- Right-click on the error, and from the displayed menu, select the option to restore this file.

Solution 2

Obtain the file "external_tools.xml" from our Support Desk, and save it to the following location on Windows Vista and Windows 7:

*C:\Users**YourUserID**\AppData\Local\Manitoba HVDC Research Centre\PSCAD*

(if the appdata folder is hidden, it may be displayed as per Appendix E)

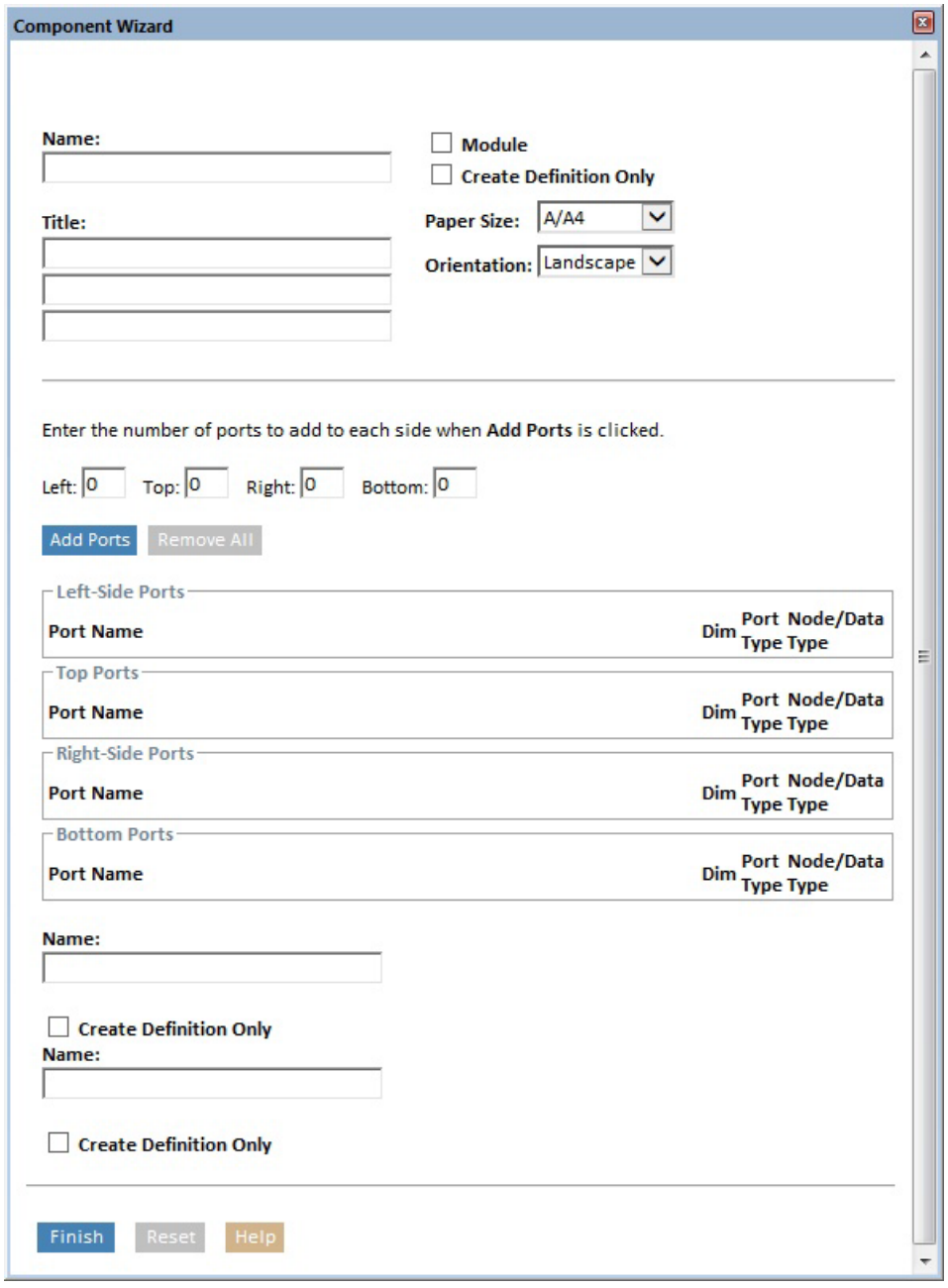
Solution 3

Update your software to v4.5.2 or later (contact our Sales Desk at sales@pscad.com).

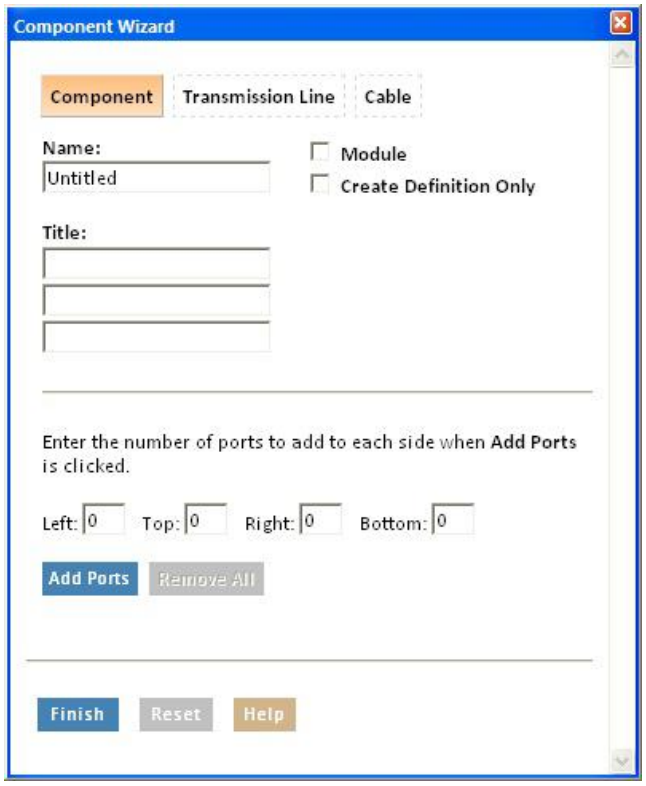
6.2 The Component Wizard Pane is not Displaying Correctly

Problem

The Component Wizard pane is not displaying correctly:



Note
The correct layout for the Component Wizard is as follows:



System

PSCAD v4.5.2 and v4.5.3 x64, on platform with Windows 7
(6.01.7601 Service Pack 1)

Note
This problem does not exist using Windows 7 x86 platform.

Cause

An issue with Internet Explorer 11.

Solution

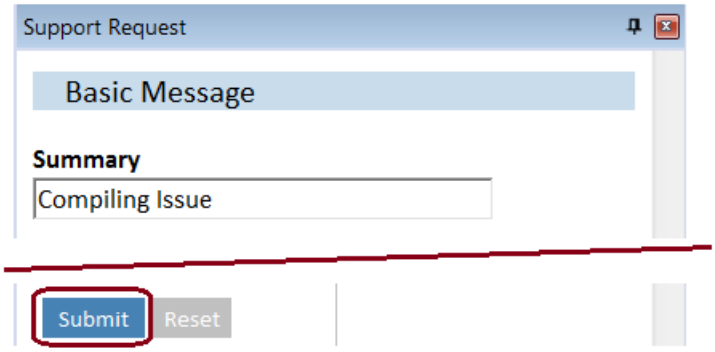
Update to PSCAD v4.5.5 or later, or

Uninstall all Internet Explorer versions, then install and use Internet Explorer 9.

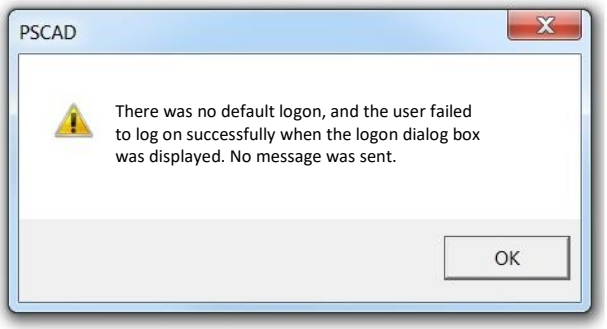
6.3 Receiving a "The PSCAD automated email system has failed to send your request" Message

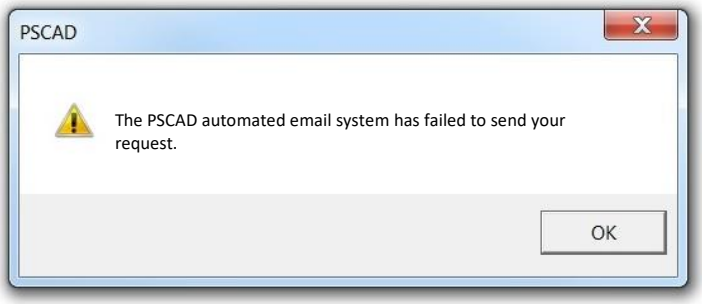
Problem

When the "Submit" button is selected in the PSCAD application…



…the following four messages are displayed, and the support request is not e-mailed:





System

This occurs in PSCAD v4.5 on a select number of machines.

Cause

The cause is unknown, but is likely related to machine settings.

Solution

Consult with your IT personnel to determine what machine settings are preventing PSCAD from sending an e-mail using your e-mail tool and your e-mail server.

6.4 Receiving an "Unable to display start page" Error Message

Problem

When PSCAD is launched, the following error message is displayed in the canvas:

> Unable to display start page. Detected version of Internet Explorer is not supported.
> Required: Version 7 or better
> Detected: Version 7

Applicable Software

This is applicable to licensed versions of PSCAD (Educational, Professional, Trial), for versions 4.5.1, 4.5.2, and 4.5.3.

Solution 1

This has been fixed in later versions; update your software to v4.5.4 or later.

Solution 2

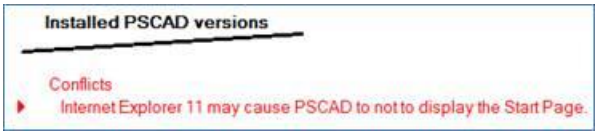
Use the Fortran Medic utility to resolve this issue as follows:

- Download the latest "FortranMedic" from our website:

  http://updater.pscad.com/utilities/FortranMedic.zip

- In the downloaded .zip file, run the FortranMedic.exe file.
- Click on the "Actions" menu and select "Start".
- After the utility has finished retrieving information:

◦ Scroll down to "Installed PSCAD versions", and right-click on the "Conflicts" message as shown:



◦ When prompted, select "OK".

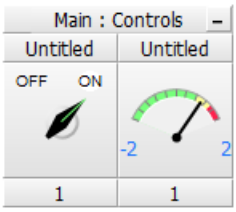
- The PSCAD Start Page in the PSCAD application should be available for display.

6.5 Copying a Control will Switch its Value to the Default Setting

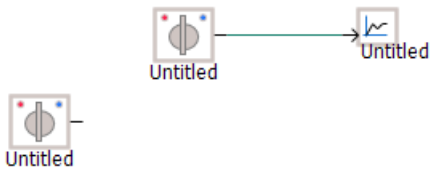
Problem

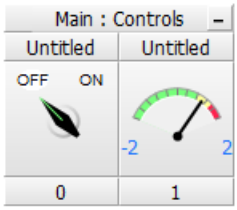
When a control is copied, the value of the original control switches to the default setting.

For example, if the following controls are copied…



…the value of original On/Off control is switched from ON to OFF:





Solution

Update to v4.5.4 or later; this is a bug that was present in earlier versions of PSCAD.

6.6 Receiving an "Unable to connect to MyCentre server" Error Message

Problem

When PSCAD is opened, the following error message displays in the canvas:

Applicable Software

This is applicable for versions 4.5.1, 4.5.2, and 4.5.3. It is less likely to appear when running v4.5.4

Cause

This message is displayed in the following situations:

- If you do not have internet connection, or
- If your internet is too slow, or
- If MyCentre is down

Solution (1)

Update your software to v4.5.4.

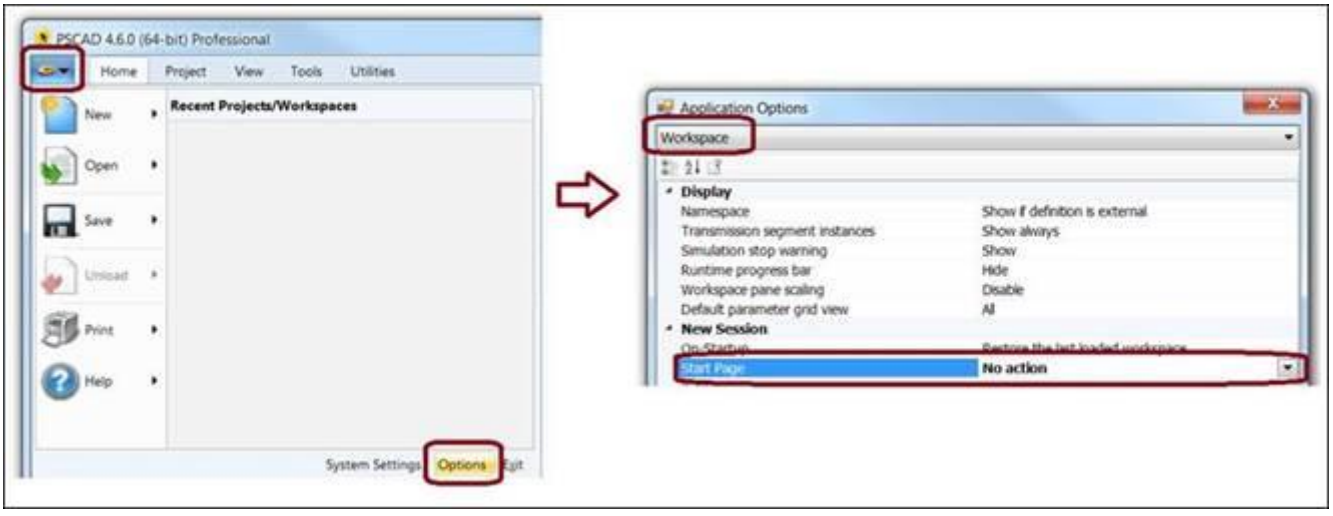
Version 4.5.4 has slightly better handling for this. Version 4.5.3 tries to connect once, whereas version 4.5.4 will try to connect several times before giving up and displaying this message.

Solution (2)

Press the F5 key to prompt the start page to try loading again.

Solution (3)

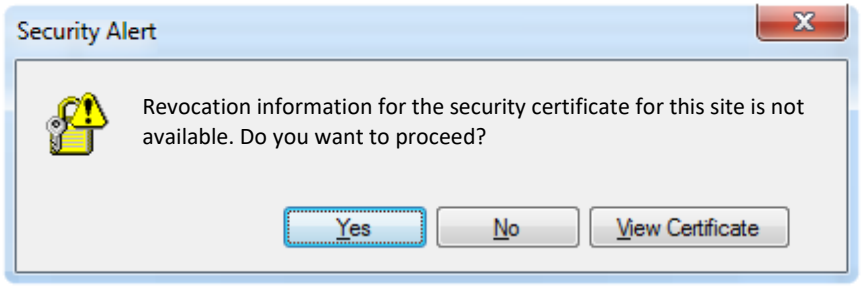
Disable the Start Page, PSCAD will not attempt to contact the server, and the error will not display:



6.7 Receiving a "Revocation information for the security certificate for this site is not available" Error Message

Problem

When PSCAD is run, the following error message is displayed:

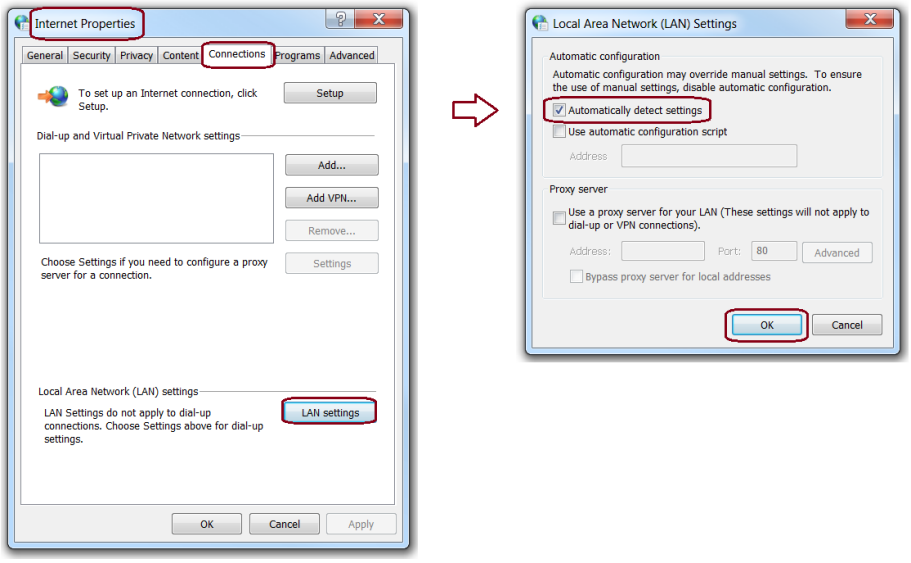


Cause

This message displays when PSCAD attempts to obtain information from our server to display in the Start Page, and may be caused by an incorrect setting in your Windows Internet Properties.

Solution

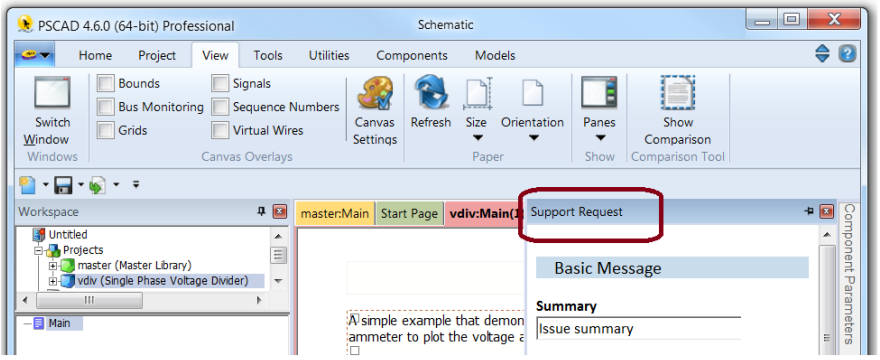
Change your LAN settings to "Automatically detect settings", as shown below:



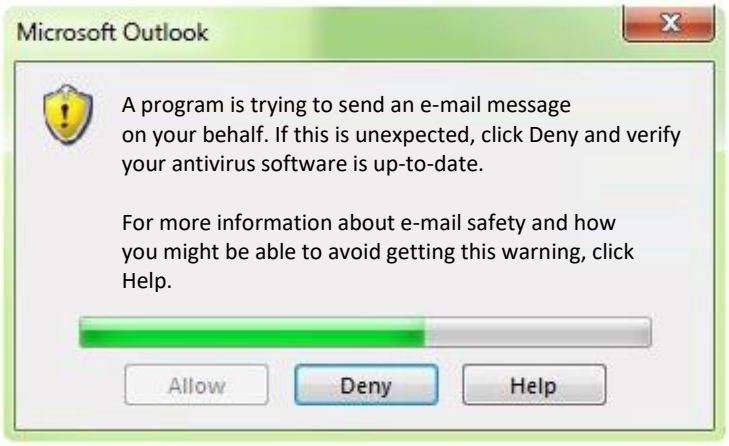
6.8 Receiving a "A program is trying to send an e-mail message on your behalf" Error Message

Problem

When a support question is submitted through the PSCAD application…



…the following error message is displayed:



Cause

The Microsoft Outlook application displays the above message when it is invoked by another application, and if it detects that a virus scanner is not installed or is not current.

Solution 1

Select the "Allow" button once the progress bar is 100% complete. The e-mail will be sent.

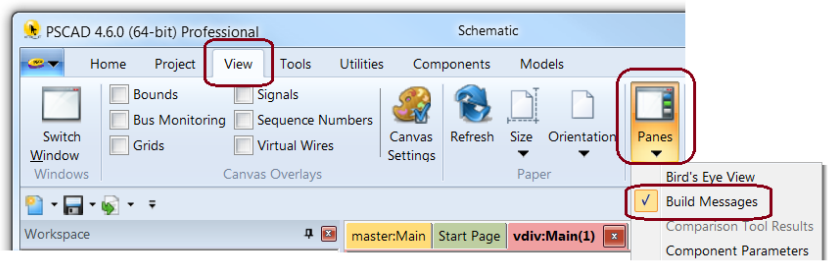
Solution 2

Install or update your virus scanner in order to prevent this message from displaying again.

6.9 A Pane Appears to be Missing

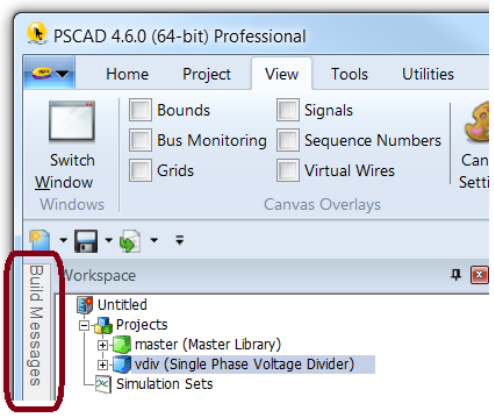
Problem

A PSCAD pane is not displayed in the PSCAD application, even though it is selected for display in the View | Panes drop-down menu:

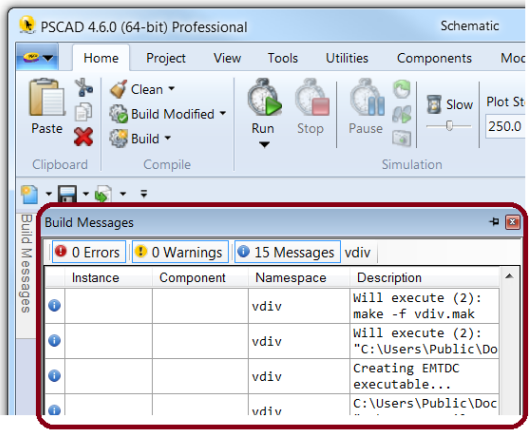


Solution 1

The pane is docked, or minimized as a tab along the outer edge of the application.

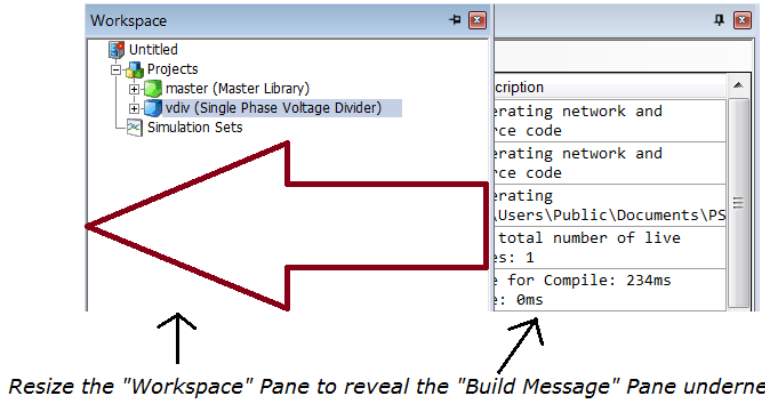


Display the pane by left-clicking on the tab. Left-clicking outside the application will cause this pane to revert back to the docked position.



Solution 2

The pane is hidden behind another pane. Resize the top pane to reveal the pane underneath:



Resize the "Workspace" Pane to reveal the "Build Message" Pane underneath

Solution 3

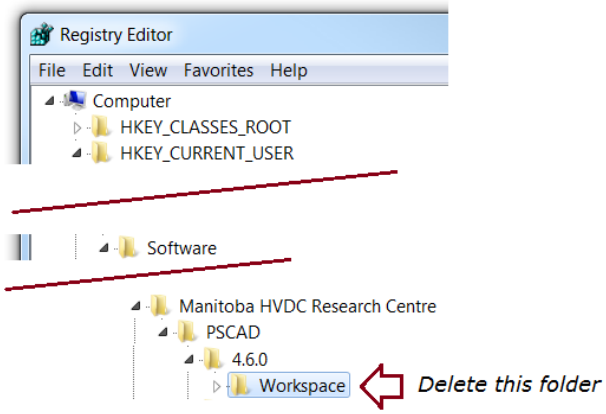
If the above solutions do not resolve the issue, the PSCAD docked windows may be reset to the default state. This is performed by deleting the Workspace key in the registry for the particular version of PSCAD that you are using.

> **Note**
> Windows Administrator Privileges are required for this.

- Launch the Windows Registry Editor with Windows Administrator privileges (from the Windows Start menu, browse to and right-click on regedit.exe, and select the option to "Run as administrator" when prompted).
- In the Registry Editor, browse as follows:

  HKEY_CURRENT_USER | Software | Manitoba HVDC Research Centre | PSCAD | [your version of PSCAD] | Workspace

- Delete the entire "Workspace" folder as listed in the above path. For example, this is the path for deleting the Workspace folder in v4.6.0:



6.10 Graphs do not Display

Problem

When PSCAD is run, no results are displayed in the graphs.

Solution

There is an issue with one or more of the components in your network.

6.11 Issues with Debugging PSCAD

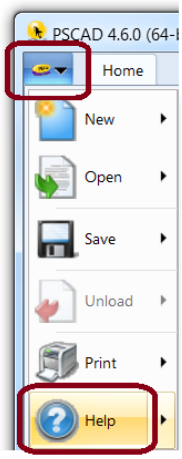
Problem

User is unsure how to debug his Fortran files.

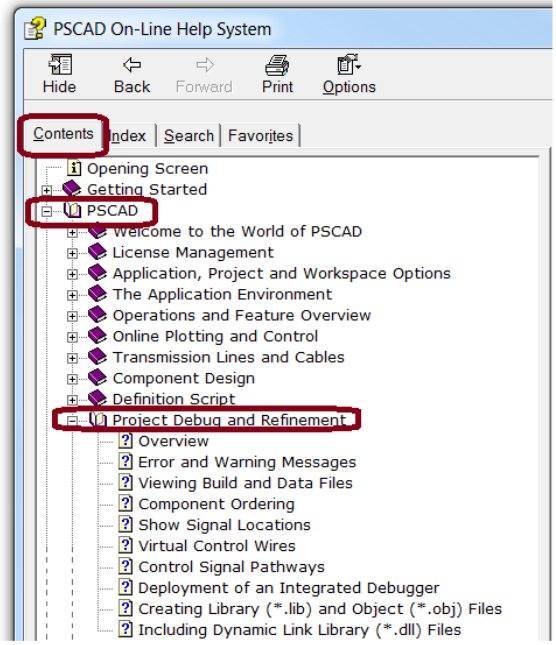
Solution

Refer to the PSCAD Help system, available as follows:

- Launch PSCAD.
- Display the Help system from the PSCAD Start menu:



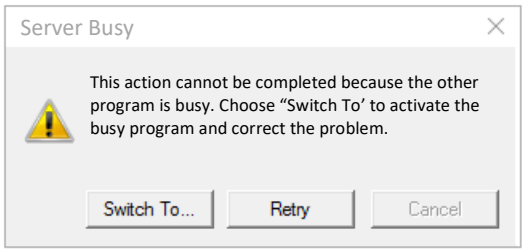
- Browse to the “Project Debug and Refinement” section:



6.12 Receiving a “Server Busy” Error Message

Problem

PSCAD is launched and licensed, but the following error message displays:



Cause

The Server Busy message appeared when PSCAD was running on Windows in a non-fully-trusted mode. A code change in PSCAD 4.6.1 resolved the issue.Solution (1)

Update to PSCAD v4.6.1 latest build (Build 146), released February 10, 2017.

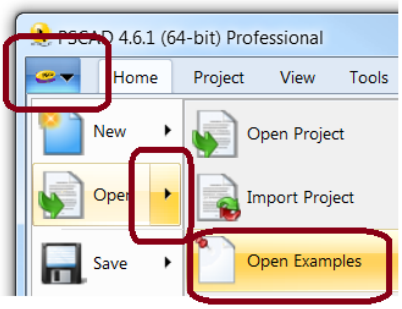
Solution (2)

Retain your current version of PSCAD, but select “Retry” to continue.

6.13 The "Open Examples" Menu Option does not work

Problem

The "Open Examples" menu option does not automatically display the folder containing the examples. Instead, it opens a previously-used folder.

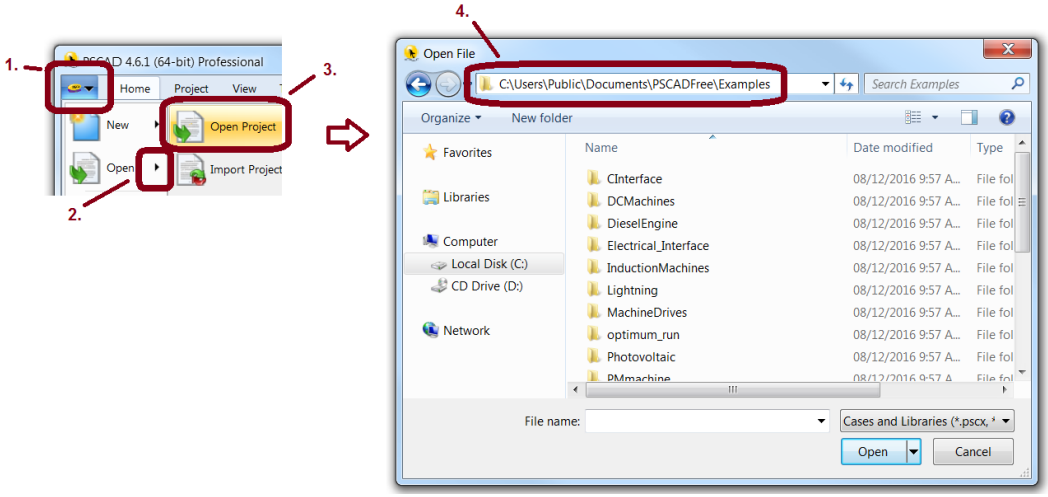


Cause

To be determined.

Solution

Display the available examples as shown below. The examples may then be selected and loaded into PSCAD.



6.14 Cannot load a PSCAD Project

Problem

When trying to load a project file from a file path containing Unicode, for example Kanji characters, the project cannot open, and returns an error similar to the following:

C:\...<SOME KANJI CHARS>\…

Cause

Unicode is not supported in PSCAD.

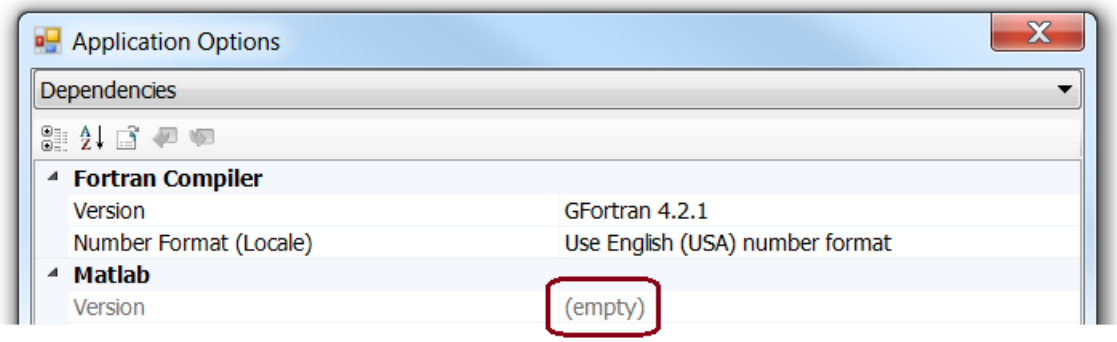
Solution

Ensure that PSCAD project folders and case names do not contain Unicode.

6.15 Matlab is not detected by PSCAD

Problem

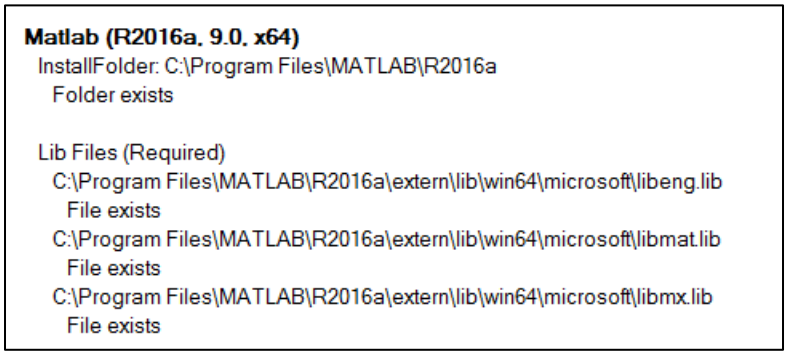
In the PSCAD Appliation Options dialog, Matlab is not detected, and may therefore not be selected:



Cause (1)

Matlab is not fully installed.

The Fortran Medic utility may be used to check whether this software is fully installed. Launch the Medic utility as per Appendix A.2, and view the Matlab installation section. If Matlab is installed, the installed version will be listed, along with the required files, similar to the following:



Solution (1)

Fully install Matlab.

Cause (2)

An incompatible version of Matlab is installed. For example, PSCAD v4.5 and earlier are 32-bit applications, and will not detect a 64-bit version of Matlab.

Solution (2)

Install a compatible version of Matlab.

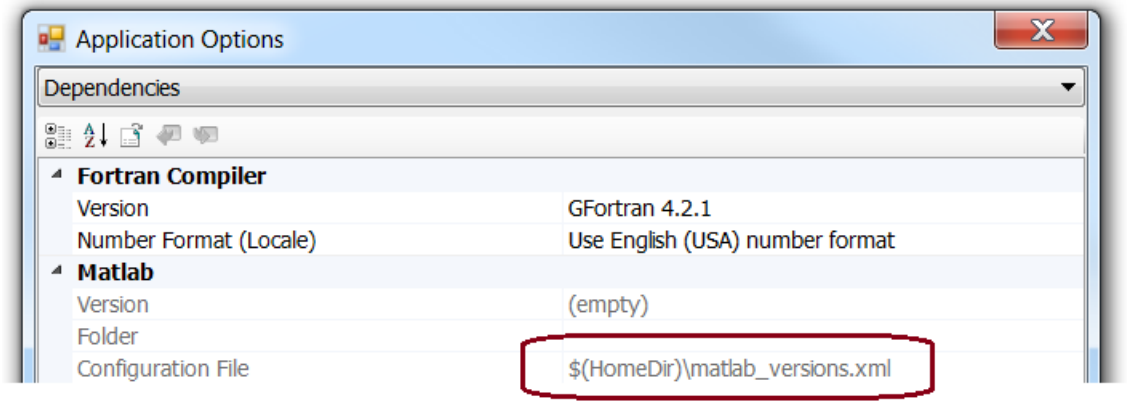
Cause (3)

The default configuration file has been changed, and is pointing to a wrong path.

Solution (3)

Reset the configuration file to the default setting:

$(HomeDir)\matlab_versions.xml)



6.16 Receiving a "Security Alert" Message

Problem

When PSCAD is launched, the following messages display:

This is simply a warning. Selecting "Yes" allows PSCAD to be used.

Or, if "View Certificate" is selected, a dialog box similar to the following displays:

Cause

This message is related to viewing the Start Page in the PSCAD application. It may not be viewed because the user's network is somehow not allowing the certificate issuer, GeoTrust SSL CA – G3.

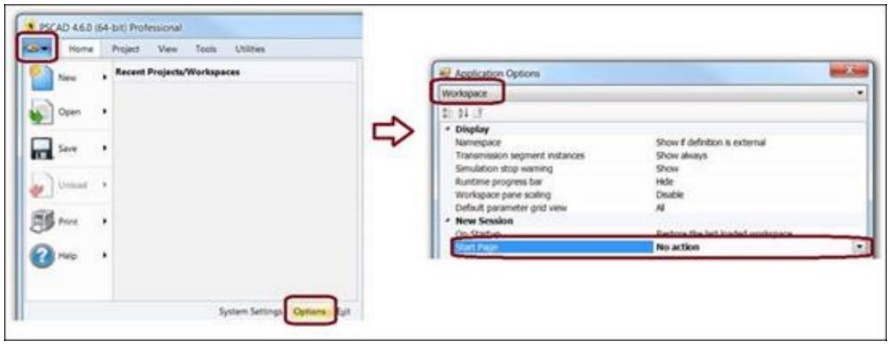
Solution (1)

Select “Yes” in the above view, and proceed to use PSCAD.

Solution (2)

If you do not want to display the Start Page at all, this may be de-activated as follows:

- Disable the Start Page from displaying:



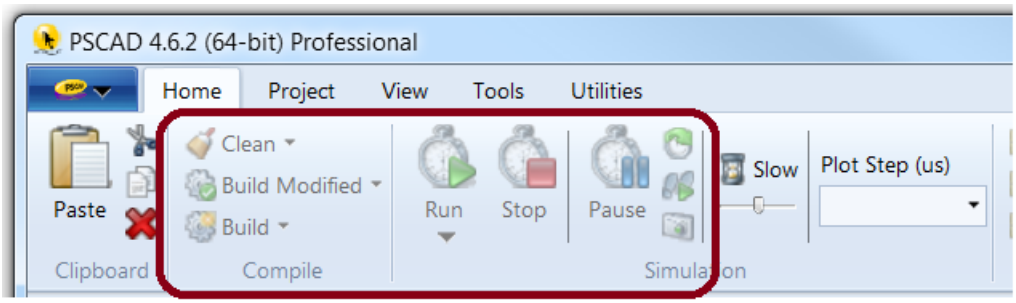
- Modify the name of the “index.html” file, which will help to disable the Start Page. The following is an example, showing that I added “_disable” to this filename:

C:\Program Files (x86)\PSCAD[version]\Forms\startpage\index_disable.html

6.17 Menu Tools are not Active

Problem

The tools in the menus are not active (they are greyed out). See the following image for an example:



Cause 1

PSCAD is not licensed.

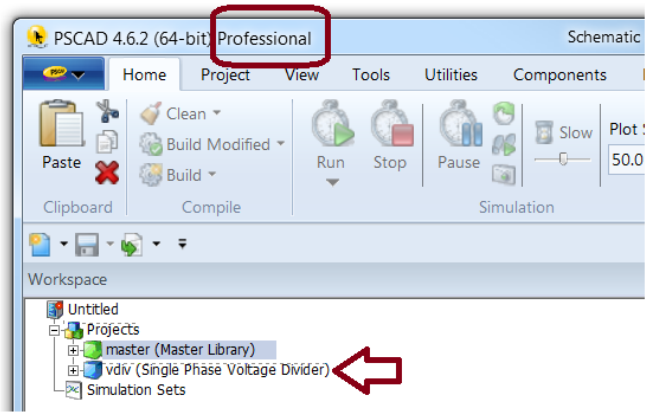
Solution 1

License PSCAD.

Cause 2

PSCAD is licensed, however, a project is not loaded and selected (highlighted) in the Workspace.

For example, in the following view, the software is licensed and a project is loaded into the Workspace ("vdiv (Single Phase Voltage Divider"). However, this project has not been selected, instead the Master Library is selected, and therefore, the menu tools are not active.



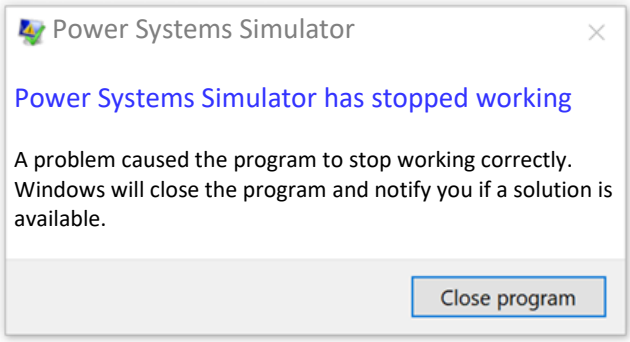
Solution 2

Left-click on the project to select it.

6.18 Receiving a "Power Systems Simulator has stopped working" Error Message

Problem

When using the tools in PSCAD v4.6.2 in a normal manner, the following error displays, and PSCAD crashes:



If MyUpdater is being used to launch PSCAD, a MyUpdater error displays when trying to launch PSCAD v4.6.2:

Process terminated with error: C000041D

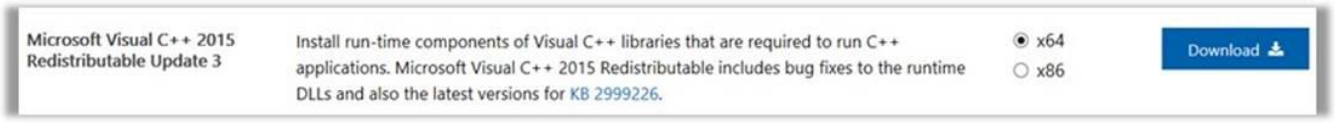
Cause

Solution

- Open a web browser to here:

  https://www.visualstudio.com/vs/older-downloads/

- Scroll down to the very bottom of the above webpage, select the Visual Studio C++ 2015 Redistributable Update 3 (**x64**), and download and install it:

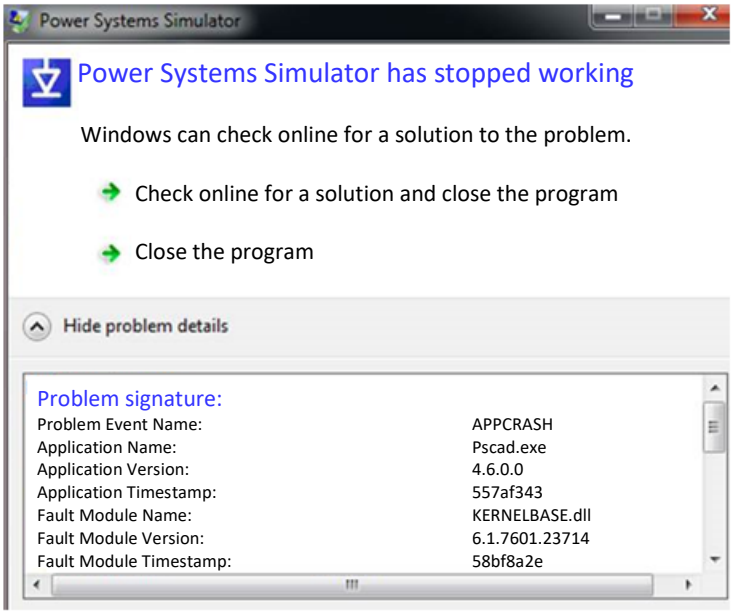


- Select the Visual Studio C++ 2015 Redistributable Update 3 (x86) edition, and download and install it:

6.19 Receiving a "Power Systems Simulator has stopped working" Error Message

Problem

When attempting to launch PSCAD, the following error displays, and PSCAD crashes:



Cause

The executable file for launching PSCAD is being blocked by protection software. For example, it may be blocked by a Data Loss Prevention (DLP) tool.

Solution

Ensure that the protection software is configured to allow PSCAD to be launched. For example, for PSCAD v4.6, the following files must not be blocked by the protection software:

- For PSCAD 64-bit:

  C:\Program Files (x86)\PSCAD46\bin\win64\pscad.exe

- For PSCAD 32-bit:

  C:\Program Files (x86)\PSCAD46\bin\win\pscad.exe

6.20 64-bit Editions of Intel Fortran are not detected by PSCAD

Problem

When attempting to select a compiler in the PSCAD application, only the 32-bit editions of the Intel Fortran compiler are detected. The 64-bit editions of Intel Fortran are not detected. Furthermore, the GFortran 4.6 compiler is not detected.

System

PSCAD v4.6

Intel Fortran 15+

Cause

The fortran compiler file is outdated in the PSCAD program files. For example, the file may be from PSCAD v4.5, which did not support 64-bit editions of the Intel Fortran compiler nor GFortran 4.6.

Solution

Replace the existing fortran compiler file with the most recent one as follows:

- Locate the existing fortran_compilers.xml file. It will be located in a path similar to the following:

  C:\Program Files (x86)\PSCAD46\fortran_compilers.xml

- Rename the file. For example:

  From: fortran_compilers.xml

  To: fortran_compilers**_1**.xml

- Download and launch the latest Medic tool as per Appendix A.2.

- When the tool has finished detecting your information, scroll down to the fortran_compilers.xml file error, which will be similar to the following, right-click on the error, and select the option to update the file:

  Conflicts
  The PSCAD 4.6.2 fortran_compilers.xml file is missing and will not detect any Fortran compilers
  File: C:\Program Files (x86)\PSCAD462 Testing\fortran_compilers.xml

6.21 User does not have permissions for LocalAppData folder

Problem

If a user does not have permissions for the LocalAppData folder, then PSCAD is unable to store its user profile settings, other configuration files, and log files.

This folder is normally set to: C:\Users\USERID\AppData\Local

Solution (1) – For any version

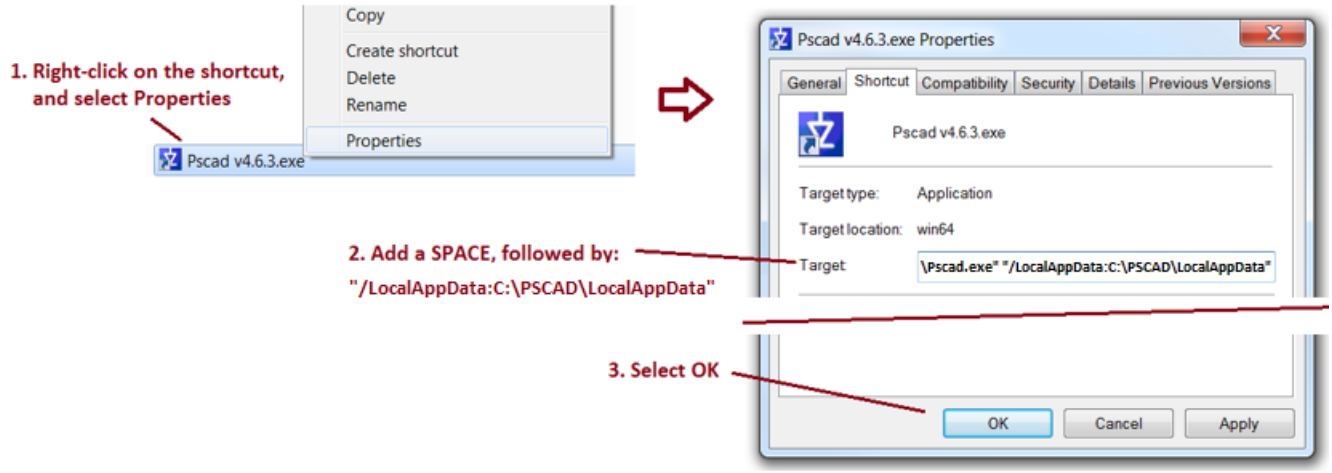
Obtain permission to create and write to the above location.

Solution (2) – For PSCAD v4.6.3+

If using PSCAD v4.6.3 and later, configure PSCAD to use a folder for which you have permissions, as follows:

In a location in which you have permissions, create a new folder. For example: C:\PSCAD\LocalAppData

- Add the following path to the folder: Manitoba HVDC Research Centre\PSCAD. In the above example, the path will become: C:\PSCAD\LocalAppData\Manitoba HVDC Research Centre\PSCAD
- Determine your preferred PSCAD launch shortcut (e.g. desktop, Windows Start menu, Task Bar).
- In the PSCAD launch shortcut, modify the command line argument to point PSCAD to the new folder. Using the above example, this would be added as shown:



- Whenever PSCAD is launched using this shortcut, the user files will be stored in the new folder.

Note about Solution (2):

The alternative folder must be specific to a machine. You cannot use a public folder accessible to all machines as then all users would share a common user profile settings, which could be modified by any user.

Example of a valid path:

\\SomePublicFolder\Machine1\LocalAppData [used only by Machine1]

\\SomePublicFolder\Machine2\LocalAppData [used only by Machine2]

Example of an invalid path:

\\SomePublicFolder\LocalAppData [used by all machines]

6.22 Information in the PSCAD Upper and Lower Frames is not Visible

Problem

Sometimes, the information in the PSCAD upper and lower frames is not visible.

This issue can occur when a PC is disconnected and reconnected to a dock, when remote desktop is enabled then disabled, or when a different monitor is plugged in to a PC.

Cause

The information in the upper and lower frames is actually being displayed, but it is not visible because the colour of the text and the fill in the frames are the same.

System

This issue is known to occur with Windows 10.

This issue is known to occur when using 4k monitors.

This issue is present with PSCAD 4.6.x, as this version is built on a Microsoft framework that has known problems with rendering application frames.

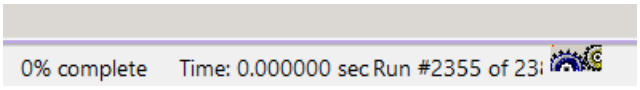
Solutions

1. Adjust your PC settings, for example, the Theme, and the Contrast.
2. Restart PSCAD
3. Update your software to PSCAD v5+ (unreleased at the time of publishing).

6.23 Some Information in the Status Bar is not Visible

Problem

When running a PSCAD project, the status bar is misaligned, and some data is not visible. For example:



Cause

This is a suspected bug in the Microsoft Windows 10 framework.

System

Occurs in PSCAD v4.6 on Windows 10.

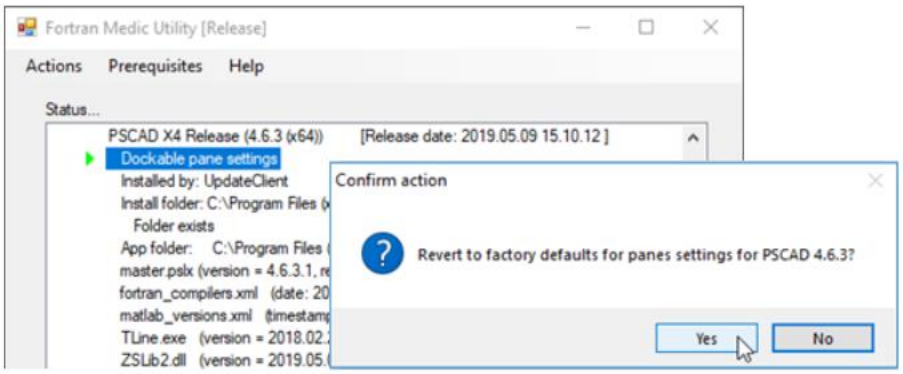
Solution (1) – Update your Software

Update your software. This issue has been was fixed in PSCAD v5.0 (not released at time of publishing).

Solution (2) - Reset Docking State

The Fortran Medic can be used to reset factory default pane settings as follows:

- Run the Fortran Medic utility as per Appendix A.2.
- Scroll down to the PSCAD installation, and right click on “Dockable pane settings”.
- Select “Yes” restore factory defaults.



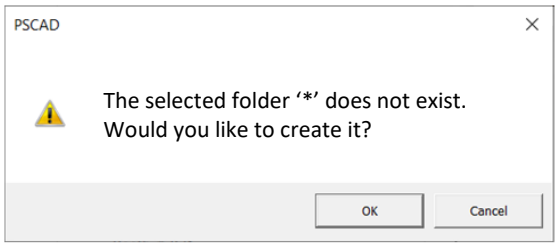
Solution (3) - Update the .Net Framework and C++ Redistributables

Updating the .NET Framework and/or Microsoft® Visual C++ Redistributables might help with this issue. These software may be updated using either the Fortran Medic utility or the MyUpdater utility. Instructions for proceeding are available in this article.

6.24 Receiving a "The selected folder '*' does not exist…" Error Message

Problem

When trying to save a project in the PSCAD application, the following error displays:



Cause

The file path contains non-English characters, which are not recognized by PSCAD.

System

Occurs in PSCAD v4.6.

Solution (1)

Update your software. This issue has been was fixed in PSCAD v5.0 (not released at the time of publish).

Solution (2)

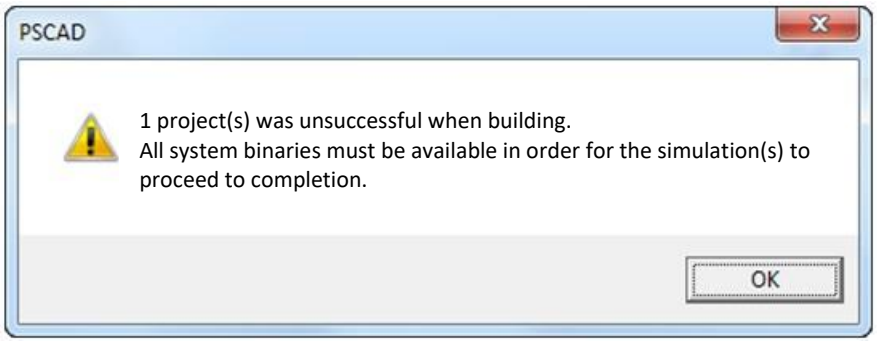
Rename the file path to include only English characters.

7. Issues when Compiling Cases in PSCAD

7.1 Receiving a "1 project(s) was unsuccessful when building" Error Message

Problem

When compiling a case, an error message similar to the following displays:



Cause 1

Simulations are crashed in the background.

Solution 1

Reboot your computer.

Cause 2

A program called "Cygwin" is installed on this machine, and the GFortran compiler is being used to run the case. Cygwin has a GNU compiler that interferes with GFortran, which comes bundled with PSCAD.

Solution 2

- Uninstall Cygwin from this machine, or.
- Retain Cygwin, but remove Cygwin variables from your Windows PATH or LIB default environment variables: Cygwin will still be able to be used with other applications, however, the user must always first invoke Cygwin, before use, or
- Run PSCAD on a machine on which Cygwin is not installed, or
- Switch from using GFortran to the Intel Fortran compiler, to compile PSCAD projects. Further information is available here.

Cause 3

If the Build messages are similar to the following…

> Fatal Error: Can't open module file 'xxx.xx' for
> reading at (1): No such file or directory
> gfortran.exe: Internal error: Aborted (program f951)
> make: ***[user_source_1.o] Error 1
> Unable to generate a simulation executable for
> namespace 'xxx"
> Make failed to generate a simulation executable for
> namespace 'xxx. Binary file was not found.

…then the error is due to a to a conflict in Fortran compilers. This project is compiling with GFortran, but it is calling an object or library that was precompiled with Intel Fortran.

Solution 3

The following are possible solutions:

- Install Intel Fortran Compiler, and compile the project using Intel instead of GFortran.
- Obtain the original source code for this item, then recompile the item using GFortran.
- Request that the developer of this item re-compile it using GFortran.

Cause 4

If the Build messages are similar to the following...

> Creating EMTDC executable...
> '\\<some network drive path>.gf42
> CMD.EXE was started with the above path as the current directory.
> UNC paths are not supported. Defaulting to Windows directory.
> C:\Windows>call
> C:\Program~2\GFortran…gf42vars.bat
> Make: <name.mak>: No such file or directory

…then the error is due to running a project on a network drive.

Solution 4

- Move your case files to a local drive (e.g. desktop) and run the project from there.

Cause 5

See Section 7.5, Problem 2.

7.2 Receiving an "unresolved external symbol" Error Message

Problem

When compiling a case, an error similar to the following is displayed in the Build Messages:

*.obj : error LNK20## : unresolved external symbol _xxx

Cause 1

The case is dependent on an object or library that has not yet been declared in PSCAD.

Solution 1

- Ensure that the object or library is saved to your machine.
- Link the object or library in PSCAD. Instructions may be found on the following webpage:

  https://hvdc.ca/knowledge-base/read,article/478/linking-objects-and-libraries-into-pscad/v:

Cause 2

PSCAD is not configured to run with the selected compiler; there is an issue with the environment variables.

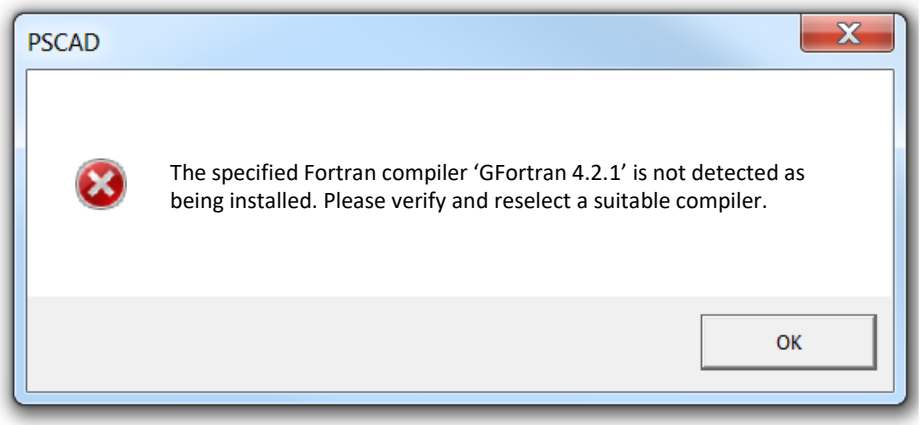
Solution 2

Run the Fortran Medic utility and send in the generated log file (see Appendix A.3).

7.3 Receiving a "Fortran compiler…is not detected" Error Message

Problem

When compiling a case, the Fortran compiler that was specified within PSCAD is not detected, and the following error message is displayed:



Cause 1

The Fortran compiler has not yet been installed.

Solution 1

Install the Fortran compiler, then log out and log back in on your computer to apply the changes.

Cause 2

There is an issue with the Windows operating system.

Solution 2

Defragment the Windows operating system.

Cause 3

PSCAD and GFortran compiler v4.2.1 were initially both installed, and were used to run a project. However, the GFortran compiler v4.2.1 was later uninstalled.

Solution 3

There are two options:

- Re-install the GFortran compiler v4.2.1, then log out and log back in on your computer to apply the changes. See the manual posted here:

  https://hvdc.ca/knowledge-base/read,article/357/installing-the-gfortran-compiler/v:

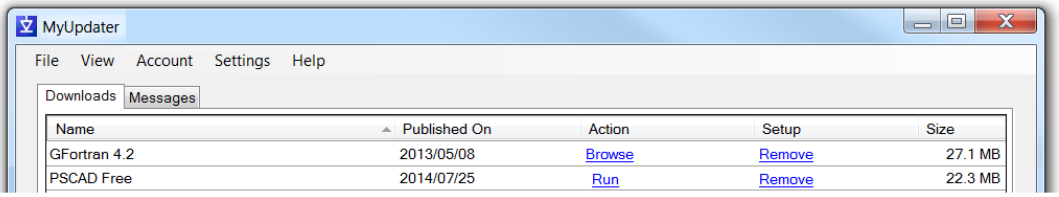
- Or, install GFortran compiler v4.6.2, log out and log back in on your computer, and select GFortran 4.6 in the Application Options (from the PSCAD Start button, select the Options link, select the Dependencies page, then select the compiler.

  Note
  This option is applicable if you are running PSCAD v4.6.0 or later, or the Free Edition.

If the Problem Persists

Please send an e-mail containing the following information to our Support Desk at support@mhi.ca.

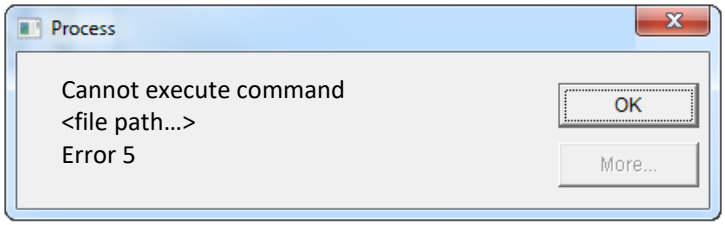
- Provide a synopsis of the problem, including a screenshot(s) of the message(s).
- Indicate which versions of PSCAD and Fortran compiler that you are trying to run.
- Provide your Fortran Medic log file (per Appendices A.2 and A.3).
- Provide screenshots showing your installed programs (from the Windows Start Menu, go to Control Panel | Programs and Features).
- If using MyUpdater for this installation, provide a screenshot of your MyUpdater utility when logged into this utility:



7.4 Receiving a " Cannot execute command…Error 5" Error

Problem 1

When compiling a PSCAD case, the following error message displays:



Note
See Section 8.9 for similar issue.

Cause 1-1

This error can occur if a previous simulation is still active in the background, either because it crashed or stopped responding for some reason.

Solution 1-1

To resolve this, reboot your computer to force it to shut down the previous simulation. Then retry the simulation.

In the future, you can check the processes running on your computer and manually end the process if required. For example if your project is called HVSIM and it crashes, you may be left with an orphaned simulation called HVSIM.exe that is still running.

Cause 1-2

PSCAD was able to compile the emtdc executable, but was not able to launch/start it due to: Error #5

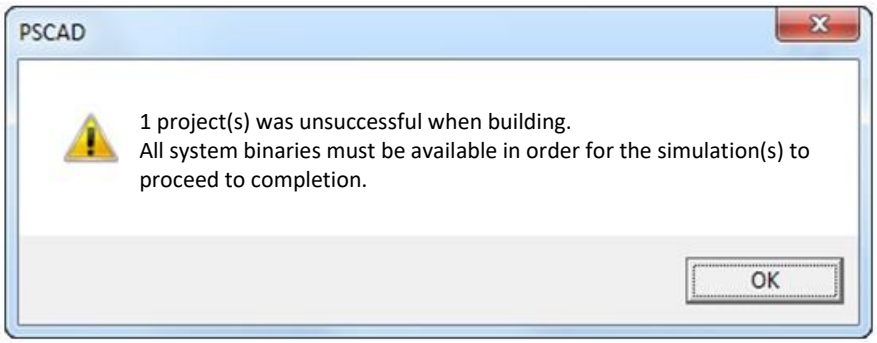
Solution 1-2

Error 5 = Access denied. The following are possible reasons for why access is denied:

- Firewall or anti-virus is preventing the launching of the EMTDC executable. Refer to Appendix F for information on the creation of executable files during a simulation.
- Machine is otherwise locked down by IT. Refer to Appendix F for information on the creation of executable files during a simulation.
- User did not launch PSCAD with elevated privileges on a Windows Vista/7 operating system.

Problem 2

When running a PSCAD case, the following messages display:



Cause 2-1

This issue may be due to the anti-virus or a firewall being set too aggressively. Refer to Appendix F for information on the creation of executable files during a simulation.

Solution 2-1

Turn off all anti-virus programs and firewalls and see if that fixes your problem. If it fixes the problem, then PSCAD should be put on the anti-virus and/or firewall exceptions list.

7.5 Receiving a "'gfortran.exe: Main.f: Invalid argument" Error

Problem

When compiling a case, compiling messages similar to the following display:

```
Will execute: call C:\Program Files\GFortran\4.2.1\bin\gf42vars.bat
Will execute: make -f vdiv.mak
Will execute: C:\Users\Public\DOCUME~1\PSCAD44\examples\tutorial\VDIV~1.GF4\vdiv.bat
Creating EMTDC executable...
C:\Users\Public\Documents\PSCAD44\examples\tutorial\vdiv.gf42>call
  C:\PROGRA~1\GFortran\428484~1.1\bin\gf42vars.bat
cygwin warning:
MS-DOS style path detected: C:\Users\ YOUR-USER-ID\AppData\Local\Temp\make44882.sh
Preferred POSIX equivalent is: /cygdrive/c/Users/ YOUR-USER-ID/AppData/Local/Temp/make44882.sh
CYGWIN environment variable option "nodosfilewarning" turns off this warning.

Consult the user's guide for more details about POSIX paths:
      http://cygwin.com/cygwin-ug-net/using.html#using-pathnames

Compiling Main.f

cygwin warning:
MS-DOS style path detected: C:\Users\ YOUR-USER-ID\AppData\Local\Temp\make44883.sh
Preferred POSIX equivalent is: /cygdrive/c/Users/ YOUR-USER-ID/AppData/Local/Temp/make44883.sh
CYGWIN environment variable option "nodosfilewarning" turns off this warning.
Consult the user's guide for more details about POSIX paths:
      http://cygwin.com/cygwin-ug-net/using.html#using-pathnames

gfortran.exe: Main.f: Invalid argument
gfortran.exe: no input files
make: *** [Main.o] Error 1
```

Cause

Another product (compiler) is interfering with the operation of GFortran. The following compilers have been seen to interfere with GFortran:

- Cygwin
- QNX

Solution (1)

See solutions in Section 7.1, or see below…

Solution (3)

If the offending compiler is required on this computer, locate and change the compiler program folder names temporarily, for using PSCAD, then switch the names back to use the offending compiler.

For example, the following changes were made on one customer's machine to retain the QNX software on the machine, but prevent it from interfering when running a PSCAD case:

- Open a Windows browser to here: C:\Program Files (x86)\QNX Software Systems\
- Change the name of this folder:
  - From: QNX Software Systems
  - To: QNX Software Systems_1
- Open a Windows browser to here: C:\QNX650
- Change the name of this folder:
  - From: QNX650
  - To: QNX650_1
- PSCAD cases may be run using GFortran. When done, change the folder names back to their original name in order to use the QNX software.

To automate the changes in the above example, the script below may be used. Save the text below in a Microsoft text file, change the file name extension from .txt to .bat, and run the .bat file. This script toggles both filenames, to include or to remove the _1:

```
IF EXIST "C:\Program Files (x86)\QNX Software Systems" (
ren "C:\Program Files (x86)\QNX Software Systems" "C:\Program Files (x86)\QNX Software Systems_1"
) ELSE (
IF EXIST "C:\Program Files (x86)\QNX Software Systems_1" (
ren "C:\Program Files (x86)\QNX Software Systems_1" "C:\Program Files (x86)\QNX Software Systems"))

IF EXIST "C:\QNX650" (
ren "C:\QNX650" "C:\QNX650_1"
) ELSE (
IF EXIST "C:\QNX650_1" (
ren "C:\QNX65_1" "C:\QNX650"))
```

Note

The above script must always be run with Windows administrator privileges, since this is required for modifying the name of one of the folders (C:\Program Files (x86)). To do so, ensure you have privileges on your machine, right-click on the .bat file, select "Run as administrator", and select "Yes" when prompted by Windows whether to allow it.

7.6 Receiving an "Error 1 – Multiple definition of '…' - first defined here" Error

Problem

When compiling a case, the following compiling error is displayed:

```
Linking object code and libraries into binary '*.exe'
Main.o:Main.f:(.text+0x0): multiple definition of 'dsdyn_'
DS.o:DS.f:(.text+0x0): first defined here
Main.o:Main.f:(.text+4ec): multiple definition of 'dsout_'
DS.o:DS.f:(.text+0x803): first defined here
Main.o:Main.f:(.text+0x743): multiple definition of 'dsdyn_begin_'
DS.o:DS.f:(.text+0xac5): first defined here
Main.o:Main.f:(.text+9bc): multiple definition of 'dsout_begin_'
DS.o:DS.f:(.text+0xc02): first defined here
collect2: Id returned exit status
make: ***[config1.exe] Error 1
```

Cause

The case had a definition titled 'DS', which is a reserved name that cannot be used in PSCAD.

Solution

Rename the definition DS and then recompile the case.

7.7 Receiving an "Error 1 – Syntax error in argument list at (…)" Error

Problem

When compiling a case, the following compiling error is displayed:

Error: Syntax error in argument list at (1)
Make: ***[Main.o] Error 1
Unable to execute make.

Cause

The PSCAD 4.3.0 master library had a bug in the synchronous machine model.

Solution

Upgrade to the latest release of PSCAD X4.

7.8 Receiving a "Make failed to generate a simulation executable for namespace '***'. Binary file was not found" Error

Problem 1

When compiling a case, a compiling error similar to the following is displayed:

> Creating EMTDC executable…
> C:\Users\Public\Documents\Pscad4.5\Examples\tutorial\vdiv.gf42> call
> C:\PROGRA~2\GFortran\420484~1.1\bin\gf42vars.bat
> 'make' is not recognized as an internal or external command,
> operable program or batch file
> Unable to generate a simulation executable for namespace '***'
> Make failed to generate a simulation executable for namespace '***'. Binary file was not found.

Cause 1

This error occurs if GFortran 4.2.1 (and possibly GFortran 4.6) is installed, but the user did not log out of Windows then log back in after installing GFortran.

Solution 1

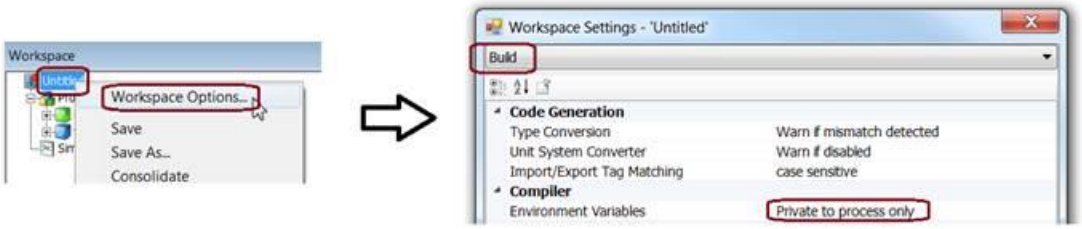
After installing GFortran 4.2.1 or GFortran 4.6, the user should log out of Windows, then log back in.

Cause 2

Some problems with your settings.

Solution 2

- Ensure that the "Compiler…Environment Variables" in the "Workspace Settings" are set to "Private to process only":



- Turn off your anti-virus or firewall, as it may be preventing the compiler from being launched.
- Test your setup by running a simple PSCAD example:

  C:\Users\Public\Documents\PSCAD\...\Examples\tutorial\vdiv.pscx

- If the example does not run, send your Build Messages and your Fortran Medic log file (Appendix A.3) to our Support Desk (support@mhi.ca).

Problem 2

The following compiling error is displayed:

> make: [<file.exe>] Error 1
> Unable to generate a simulation executable for namespace '<file>'
> Make failed to generate a simulation executable for namespace '<file>'. Binary file was not found.

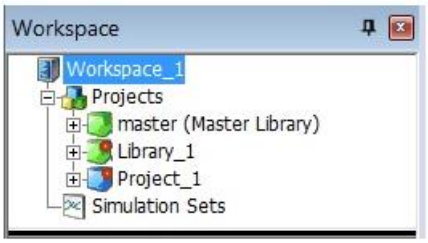
Cause 1

The case calls an object and library; these are not properly linked.

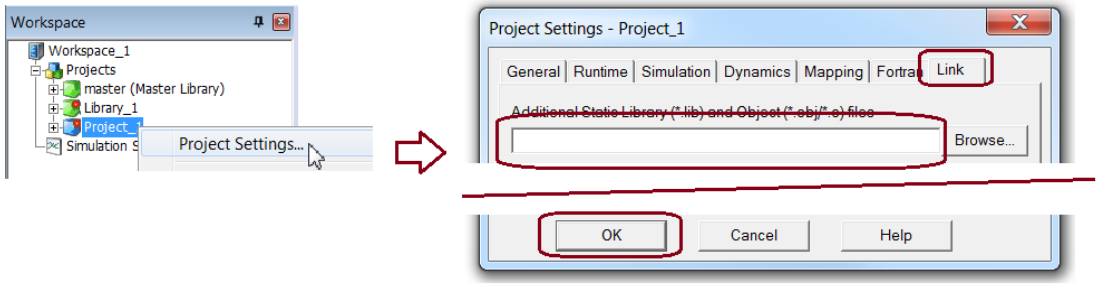
Solution 1

Link the object and library properly in PSCAD as follows:

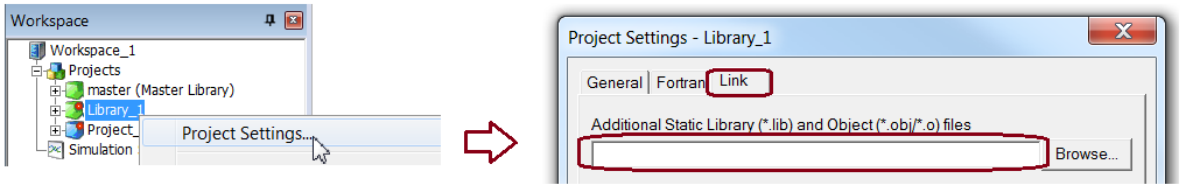
- Ensure the case files are all saved in the same folder (not required at v4.6.0 and later).
- Ensure the library is loaded ahead of the case. For example, "Library_1" is loaded ahead of (or, above) "Project_1":



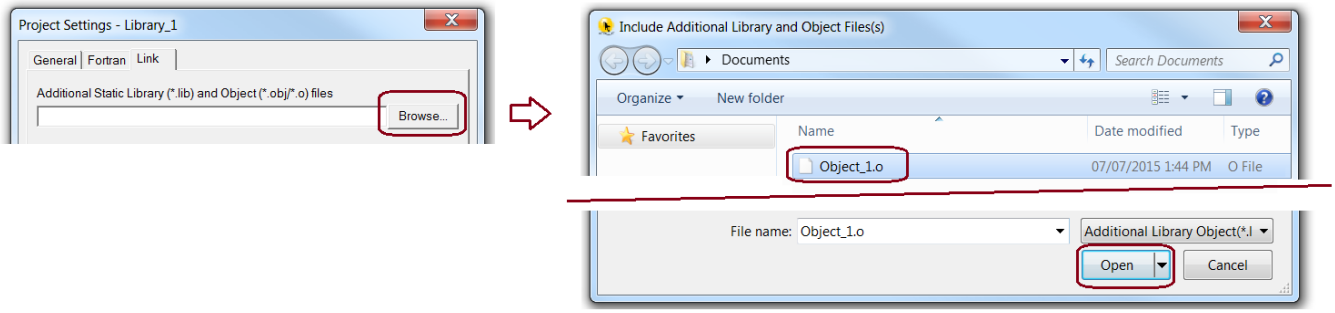
- Open the PROJECT'S Project Settings, delete any links in the field, then select "OK":



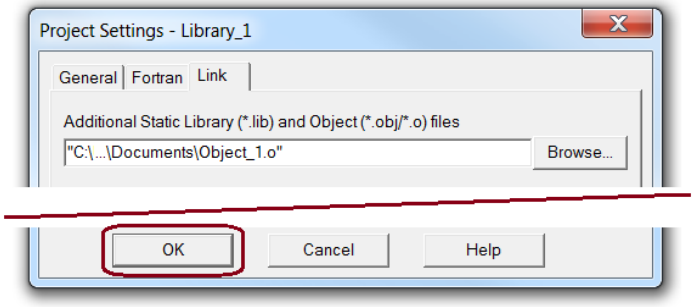
- Open the LIBRARY'S Project Settings, and delete any links in the field:



- Still in the library project settings, select the “Browse” button, browse to and select the object, then select “Open”.



- Select “OK”. The object will be linked in your library.



Problem 3

When compiling a case, the following compiling error is displayed:

Will execute: call C:\Program Files (x86)\Gfortran\4.2.1\bin\gf42vars.bat
Will execute: make –*.mak
Will execute: C:\...*.BAT
Creating EMTDC executable…
C:\Users….*BAT’ is not recognized as an internal or external command
Operable program or batch file.
Unable to generate a simulation executable for namespace ‘*’
Make failed to generate a simulation executable for namespace ‘*’. Binary file was not found.

Cause 3

This is caused by a bug in the PSCAD software in v4.5.1.

Solution 3

Update your software to v4.5.5 or later.

7.9 Receiving a "make: *** [SomeFile.exe] Error 1" Error

Problem 1

When compiling a case, the following compiling error is displayed:

```
C:\Users\user\DOCUME~1\tmp\hello\gf46\multiply.o:multiply.f:(.text+0x4): undefined reference to
  `__e_s1_MOD_timezero'
collect2: ld returned 1 exit status
make: *** [fortran_test.exe] Error 1
```

(Where test case e_s1 is the module name and timezero is one of the variables in that module.)

Cause

This error occurs if you are using GFortran 4.6, and linking in a GFortran 4.2 library when using any of EMTDC include files.

Solution 1

Use GFortran 4.2 to compile this project.

Solution 2

Recompile the library using GFortran 4.6, and run the case with these files using GFortran 4.6.

Problem 2

The following compiling error is displayed:

```
C:\Users\user\DOCUME~1\tmp\hello\gf42\multiply.o:multiply.f:(.text+0x4): undefined reference to
  `__e_s1__timezero'
collect2: ld returned 1 exit status
make: *** [fortran_test.exe] Error 1
```

(Where test case e_s1 is the module name and timezero is one of the variables in that module.)

Cause

This error occurs if you are using GFortran 4.2, and linking in a GFortran 4.6 library when using any of EMTDC include files.

Solution (a)

Use GFortran 4.6 to compile this project.

> **Note**
> GFortran 4.6 is compatible with PSCAD v4.6 and later.

Solution (b)

Recompile the library using GFortran 4.2, and run the case with these files using GFortran 4.2.

Continued…

Problem 3

The following compiling error is displayed:

```
C:\Users\<user>\DOCUME~1\ tmp\hello\gf46\*.o:*.f:(.text+0x59): undefined reference to
  `_gfortran_transfer_character_write'
collect2: ld returned 1 exit status
make: *** [fortran_test.exe] Error 1
```

Cause

This error occurs if you are using GFortran 4.2, trying to run a simple program with just a PRINT statement in it, and trying to link in a GFortran 4.6 library.

Solution (a)

Use GFortran 4.6 to compile this project.

> **Note**
> GFortran 4.6 is compatible with PSCAD v4.6 and later.

Solution (b)

Recompile the library using GFortran 4.2, and run the case with these files using GFortran 4.2.

Problem 4

When trying to compiler a PSCAD project using GFortran 4.2, the following messages display in the Build Pane

```
Generating 'C:\...\*.gf42\*.map'.
Will execute: "C:\...\*.gf42\*.mak.bat"
Creating EMTDC executable...
Linking objects and libraries into binary '*.exe'
C:\...\gf46\...undefined reference to '*'
collect2: ld returned 1 exit status
make: *** [*.exe] Error 1
Unable to generate a simulation executable for namespace '*'
```

Cause

Incompatible compilers: Using GFortran 4.2 to run a model that was developed using GFortran 4.6:

- GFortran 4.2 cannot run models created using GFortran 4.6.
- Likewise, GFortran 4.6 cannot run models created using GFortran 4.2.

Solution 1 – Change your Compiler

Compile the project using GFortran 4.6.

Setup and test GFortran 4.6.

Solution 2 – Change your Lib File

Obtain the lib file built with GFortran 4.2, and use GFortran 4.2 to run the project.

7.10 Receiving an "Error U1052 – file'data.mak' not found " Error

Problem

When compiling a case, the following compiling error is displayed:

build Will execute: call C:\Program Files\Intel\Compiler\11.0\061\fortran\bin\ifortvars.bat ia32
build Will execute: nmake -f BRK_Test.mak
build Will execute:
\\UKGLASFP02\Edrive$\Projects\PSB\EEG\2010PR~1\269371~1\PSCAD\BRK_Test.if9\BRK_Test.bat
build Creating EMTDC executable...
build '\\UKGLASFP02\Edrive$\Projects\PSB\EEG\2010 Projects\Some folder\PSCAD\BRK_Test.if9'
build CMD.EXE was started with the above path as the current directory.
build UNC paths are not supported. Defaulting to Windows directory.
build C:\WINDOWS>call C:\PROGRA~1\Intel\Compiler\11.0\061\fortran\bin\IFORTV~1.BAT ia32
build Intel(R) Visual Fortran Compiler Professional for applications running on IA-32, Version 11.0.061
build Copyright (C) 1985-2008 Intel Corporation. All rights reserved.
build Microsoft (R) Program Maintenance Utility Version 8.00.50727.42
build Copyright (C) Microsoft Corporation. All rights reserved.
build NMAKE : fatal error U1052: file 'BRK_Test.mak' not found
build Stop.
build Unable to execute make.

Software

PSCAD X4, Intel 11.0

Cause 1

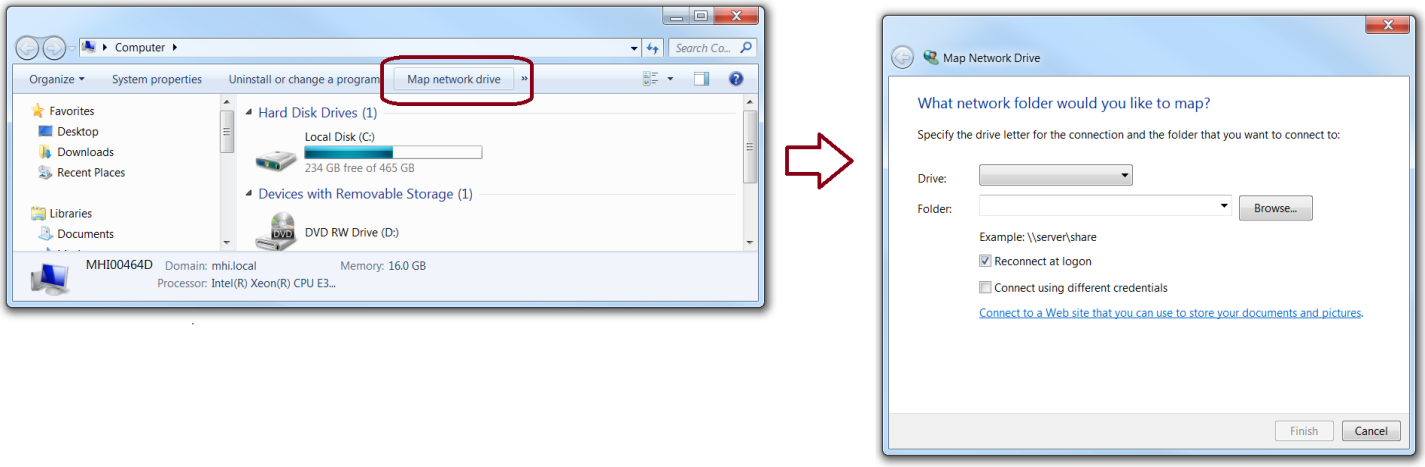
The error occurred when using PSCAD X4 to compile a case located on a network drive (no error occurs if PSCAD 4.2.1 is used). PSCAD X4 does not allow you to compile cases stored on a network drive.

Solution 1

- Copy the case to your local machine and then open it with PSCAD.

or

- Mapping your network drive to a local drive might resolve the issue:



Cause 2

PSCAD case names should NOT contain spaces.

Solution 2

Rename the PSCAD case so that it does not contain spaces.

Cause 3

User was trying to re-compile a previously compiled case, but was not running with elevated privileges.

Solution 3

Launch PSCAD with elevated privileges, then re-compile the case.

7.11 Receiving a "Make File Error" Error

Problem

When compiling a case, a Build Message error similar to the following is displayed:

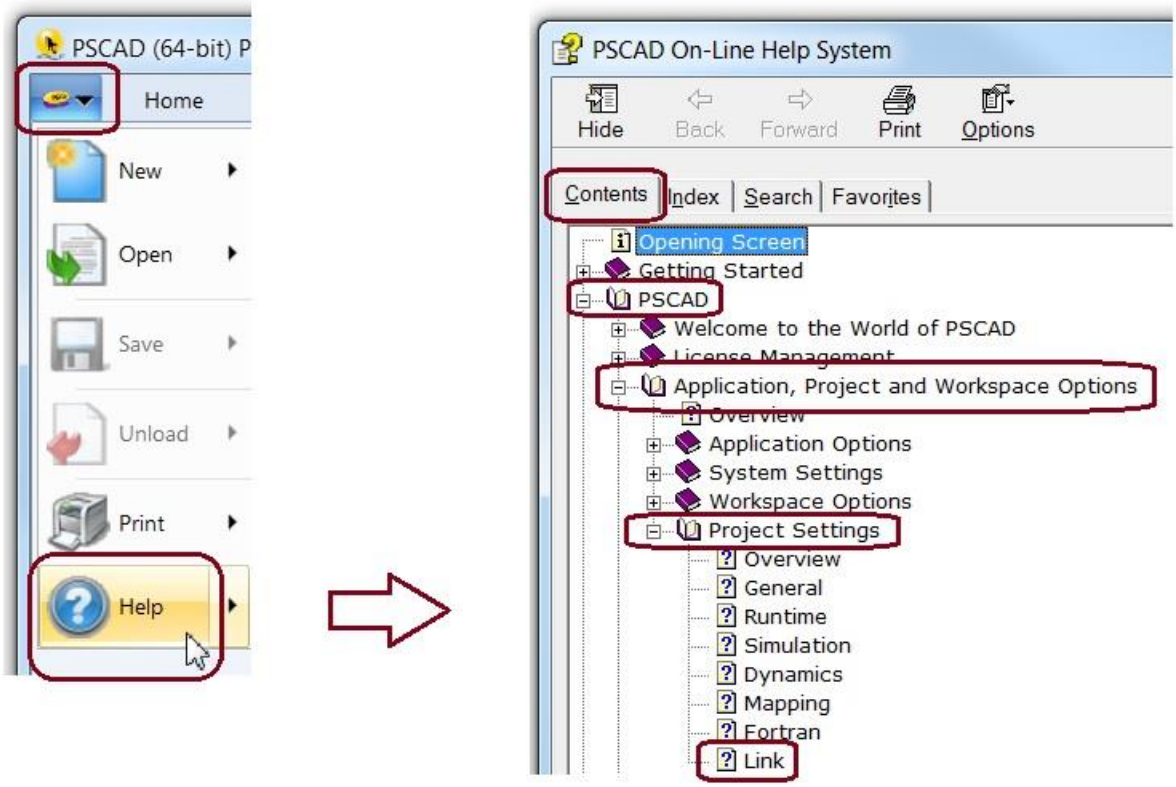
> Make File Error: The local project requires a link library:
> 'C:\….o'
> that does not exist with the given path.

Cause

An object is not properly linked to the project in the PSCAD settings.

Solution

Link the object in the PSCAD settings as per the section "Link" in the On-Line Help System:



7.12 Receiving an "Unable to solve line constants" Build Message

Problem

When compiling a case, the following message is displayed in the Build Messages:

Unable to solve line constants, check the log file for details.

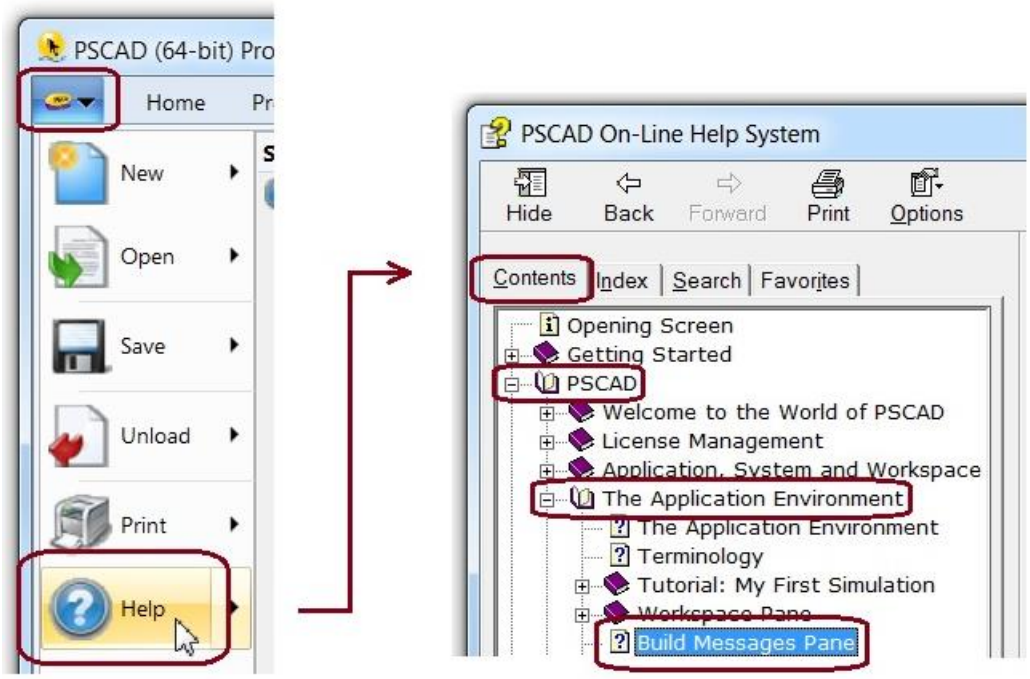
Cause

There is a flaw with the design of the circuit. For example, improper grounding, or a component open at one side.

Solution

Review the case and build messages and all T-Line messages.

For more information on reviewing the messages, refer to the PSCAD On-Line Help System:



For further analysis of a particular error message, please forward a request for assistance to our Support Desk (support@mhi.ca), along with all Build messages, and a snampshot of any error dialog boxes.

Note
Certain restrictions apply related to PSCAD license and version.

7.13 Receiving a "'cl.exe' is not recognized as an internal or external command" build message

Problem

When compiling a case, the following build messages display:

Compiling "<file path and object name>.c".
cl.exe … "<object name>.c"
'cl.exe' is not recognized as an internal or external command.

Cause

The case is trying to build an object containing c-code, but a c-compiler is not installed. "<object>.c" is the object prorgrammed in c-code; "cl.exe" is the c-compiler.

Solution

Install a c-compiler (the full version of Microsoft Visual Studio) along with Intel Fortran composer. The Visual Studio that comes bundled for free with Intel Fortran (i.e. premier partner) is not a c-compiler; the full version is commercially available.

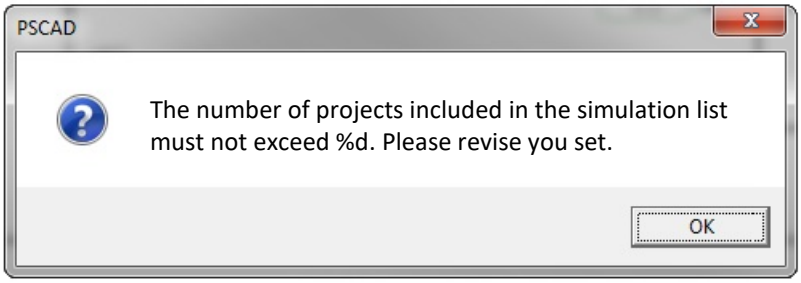
Once Intel Fortran and the c-compiler have been installed, test your setup by trying to run one of the c-code examples located at the following path:

C:\Users\Public\Documents\Pscad<version>\examples\CInterface

7.14 Receiving a "The number of projects included in the simulation list must not exceed %d" Error Message

Problem

When compiling a case, the following error message displays:



Version

This is a bug present in PSCAD v4.5.1 and earlier.

Solution

Update software to v4.5.2 or later.

7.15 Receiving an "Unable to open file…" Error Message

Problem 1

When compiling a case, the first compile passes and the case runs fine, however, some subsequent compiles fail with the following error message:

File Write Error
Unable to open file "<drive>:\<path>\<case>.mak.bat

Cause 1

Aggressive anti-virus scanner

Solution 1

Attempt to disable the virus scanner or reduce it aggressiveness, or add PSCAD to the scanner's "exceptions" list as a trusted program.

Cause 2

System process with PID 4 is holding onto this previously created file.

Solution 2

In the Windows Services Control Panel, re-enable Application Experience Service, which is used to process application compatibility cache requests for applications as they are launched.

Problem 2

When compiling a case, an error message similar to the following displays:

File Write Error Unable to open file [SOME DRIVE]:\[SOME PATH]\[SOME PROJECT]

For example:

File Write Error: Unable to open file F:\.....\\Project1

Cause

The user does not have Windows administrator permisssions for the specified drive.

Solution

Copy the project files to a different folder on which you have permissions (e.g. desktop, My Documents…).

7.16 Receiving a "Could not read symbols: Malformed archive"

Problem

When compiling a case, the compile fails with the following error message:

<path>\emtdc\gf42\windows\emtdc.lib: could not read symbols: Malformed archive

Cause

Possible corruption of PSCAD installation due to reasons unknown.

Solution

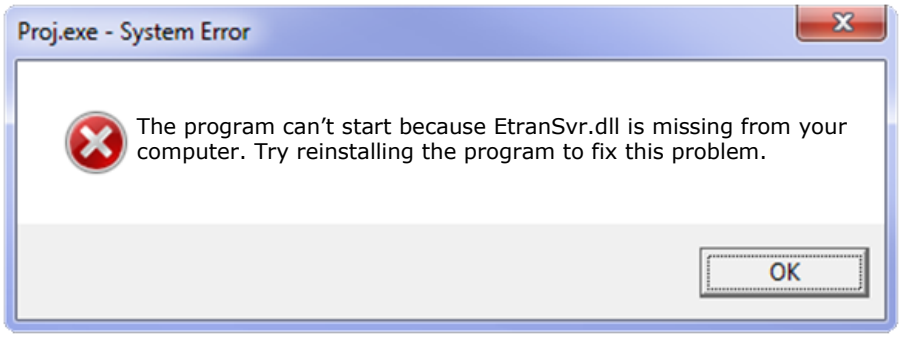
Uninstall, then re-install, PSCAD.

7.17 PSCAD Stops Unexpectely, with no Runtime Error

Problem

When compiling a case, PSCAD stops unexpectely, with no runtime error.

Running the PSCAD case manually outside of the PSCAD application results in the following error message:



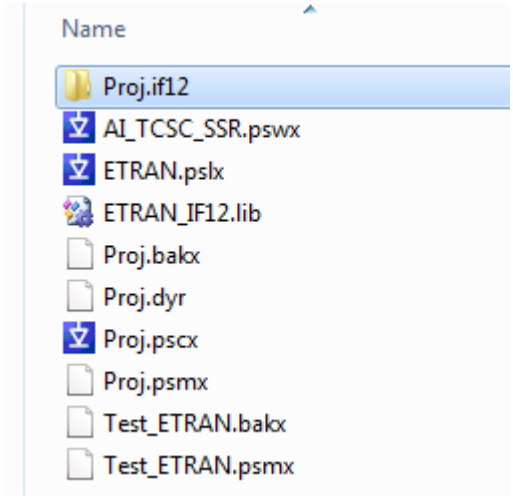
> **Note**
> Running a project manually outside of PSCAD is performed by first building the case in PSCAD, closing PSCAD, then running the .exe file from the build folder.

Cause

A required DLL file is missing.

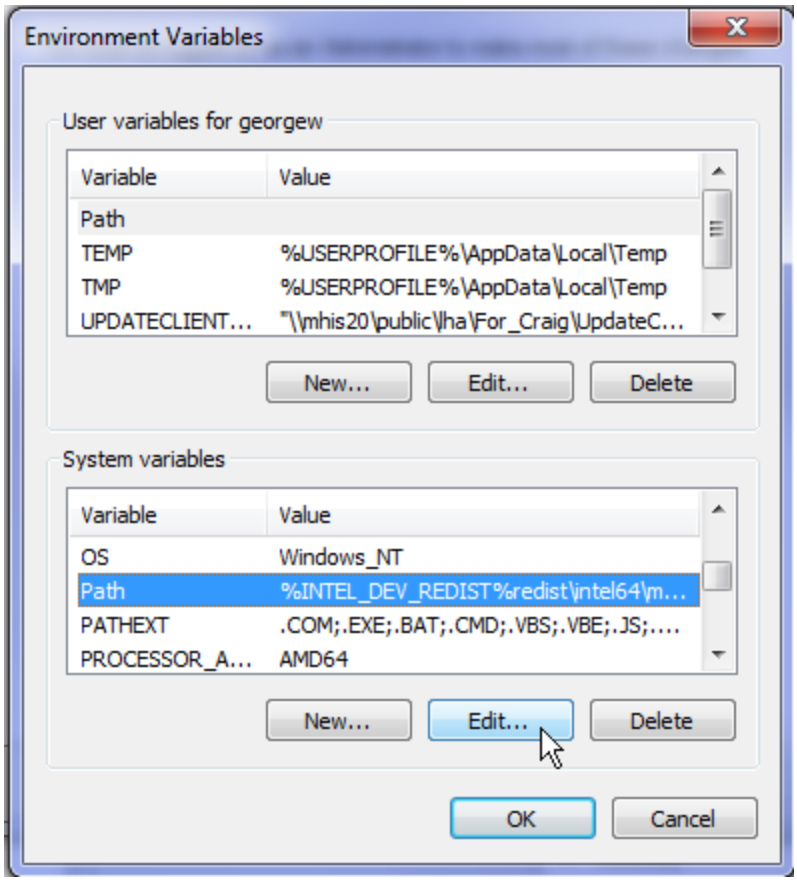
Solution 1

Locate this DLL and place it inside the .if12 folder



Solution 2

Locate this DLL and add the location to your PATH environment variable. That way the simulation will know where to look when it tries to link in the DLL.



7.18 Receiving “Error code = 112” Message

Problem

When building a case, the following error message displays:

Coding aborted due to Error: Unable to create output directory. Error code = 112

Cause 1

User is not an administrator on the computer and has limited access.

Solution 1

Obtain administrator rights on the computer.

Cause 2

The user is an administrator, but the Network is imposing limits on folder permissions.

Solution 2

- Move case files to a folder for which user has permissions, or
- Remove limits on folder permissions (this might require the assistance of your organization’s IT staff).

Cause 3

Overly aggressive virus scanner.

Solution 3

Turn your virus scanner down or off, then retry the build.

7.19 Receiving a "Transmitter '*' cannot be added" Build Message

Problem

When building a case, the following message displays in the Build Messages:

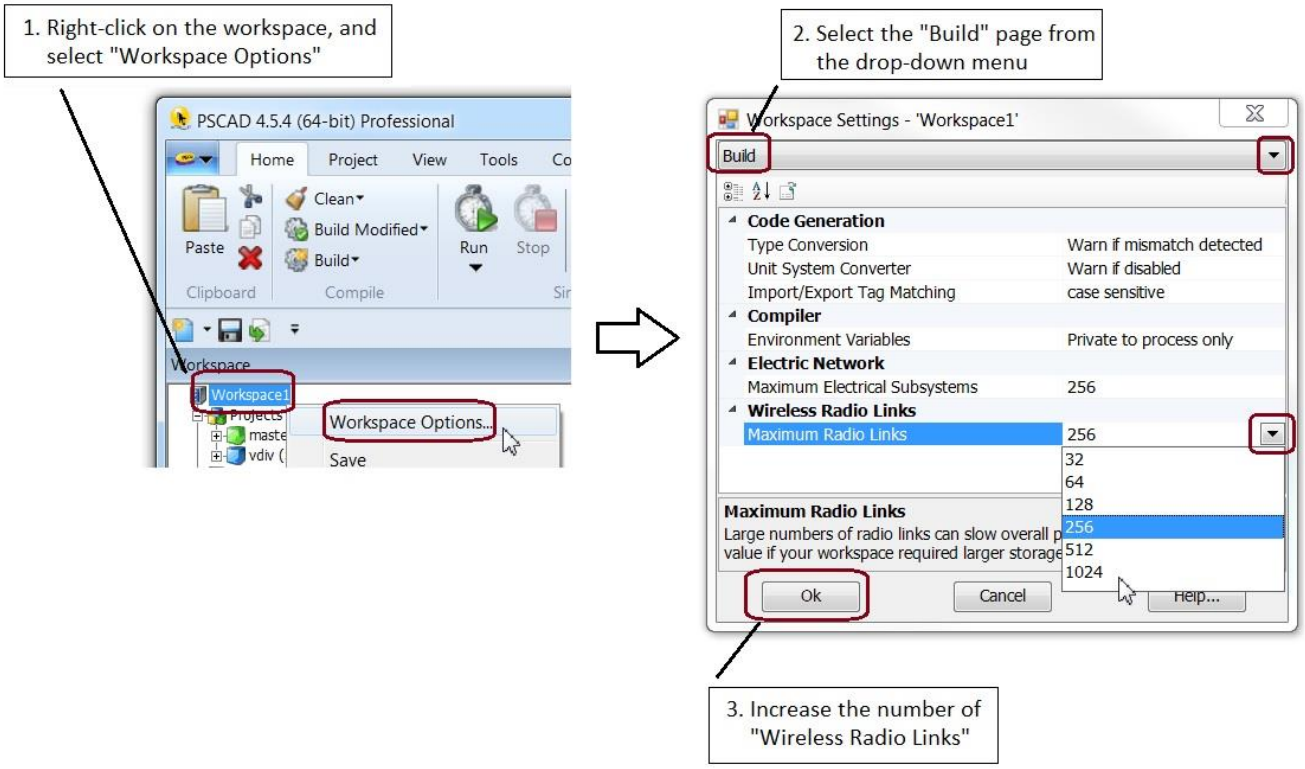
Transmitter '*' cannot be added. The storage table is at capacity of 256.

Cause

The number of allowed radio links is exceeded.

Solution

Increase the number of allowed wireless radio links as shown:



7.20 Receiving a "EMTDC: Error while opening file for read" Build Message

Problem

When compiling a case that is declaring an external file, the following build error is displayed:

EMTDC: Error while opening file for read: [some file name]
ERROR: Abnormal termination of EMTDC by OPENFILE
Non-standard Messages:
Abnormal progress termination
EMTDC Runtime Error: abnormally terminated
Simulation stopped

Cause 1

The error is related to the project being unable to open the declared file, because the declared file is residing in a folder that is different from where the PSCAD project file resides.

Solution 1

Copy the declared file to the same folder as the project resides in.

Cause 2

The error is related to the project being unable to open the declared file, because there is an error in the declared path.

Solution 2

Correct any errors in your path.

7.21 Receiving a "make: *** [Station.o] Error 126" Error Message

Problem

When compiling a case in the Free Edition using the GFortran compiler, the following error is displayed:

make: *** [Station.o] Error 126
Unable to generate a simulation executable for namespace '***'

Cause

WinAVR is installed, and is somehow interfering with compiling the PSCAD case.

Solution

- Locate the WinAVR folder (e.g. C:\WinAVR-20081205).
- Slightly change the folder name so that it won't be able to be executed. For example:

  Note
  You must be an administrator on your machine to perform this step.

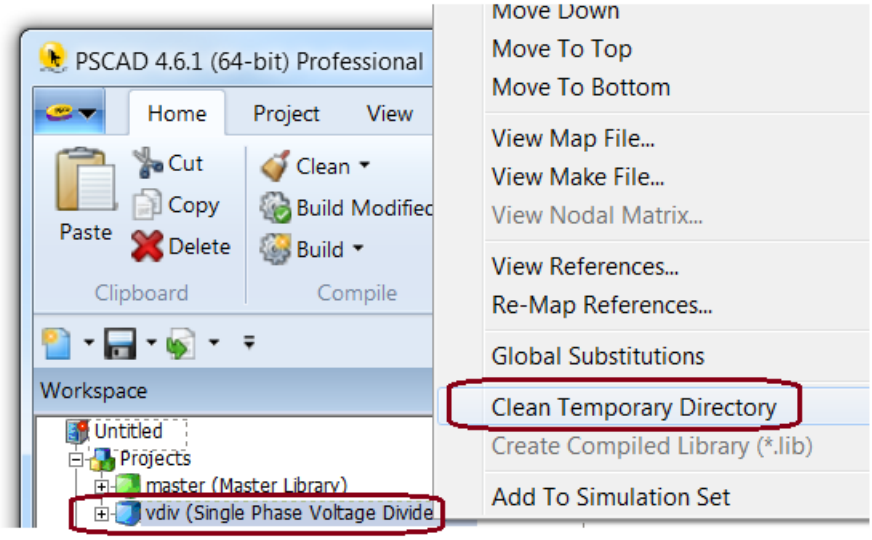
  - Change this:

    C:\WinAVR-20081205

  - To this:

    C:\WinAVR-20081205_1

- Load your project in PSCAD.
- Clean the directory (right-click on the project in the Workspace, and select "Clean Temporary Directory"):



- Re-compile your case.

7.22 Receiving a "Winsock Error #10048" Error Message

Problem

When compiling a case in PSCAD v4.5.3, the following error is displayed:

Possible Causes and Solutions

When PSCAD (GUI) starts a simulation (project), a socket connection is created so that PSCAD and the simulation can exchange data. PSCAD will randomly pick a communications socket on your computer to do this and there are times when this socket may not be available. The following are three typical scenarios that cause this error, along with their solutions.

1. Crashed simulation

Sometimes when a simulation has crashed, it will be stuck running in the background and will still be holding that communications socket open. If you try to run PSCAD again and it tries to use that same socket, then you will see this error. Try looking in your task manager to see if there are any simulations running in the background, and if so, end the process. This should free up the socket. Another sure way to do this is to restart your computer.

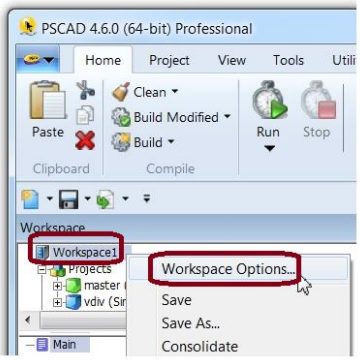
2. Aggressive anti-virus or firewall

Try shutting off the anti-virus and firewall and see if the simulation works. If the simulation works, then PSCAD should be added to the "white list". You may require assistance from your IT department for this setup.

3. Two PSCAD cases are trying to run simultaneously

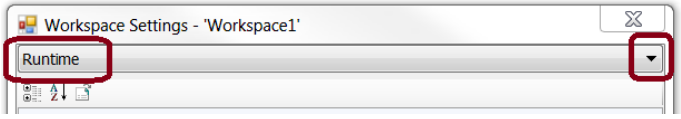
Sometimes, when two instances of PSCAD are started up on the same computer, they both try to use the same communications socket. You can prevent this by simply changing the port range on one of the instances. As the default is the 30000 range of ports, change the second instance of PSCAD to use 31000. This may be performed as follows:

a. Display the Workspace Setting dialog box (right-click on the Workspace, and select "Workspace Options"):

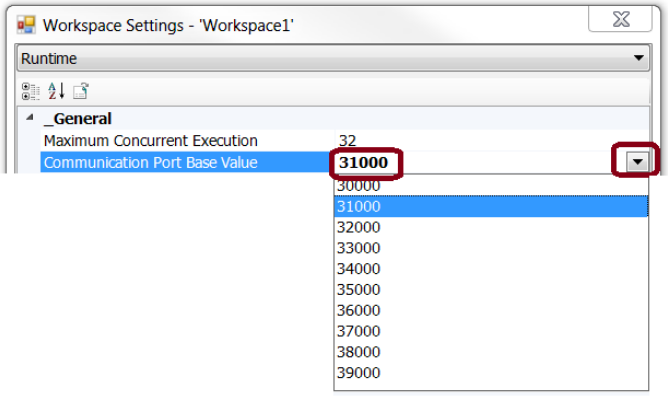


b. When the Workspace Settings dialog box displays, perform the following:

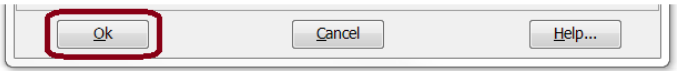
- Select the "Runtime" page from the drop-down menu.



- Select "31000" from the "Communication Port Base Value" drop-down menu:



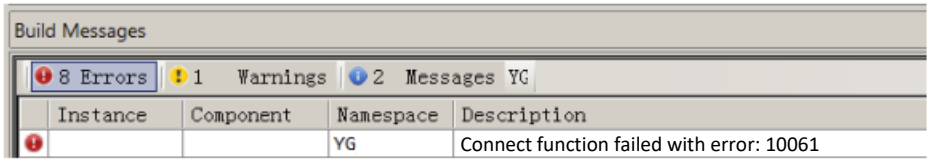
- Select "Ok" to apply the change:



7.23 Receiving a “connect function failed with error: 10061” Error Message

Problem

When compiling a case in PSCAD v4.6.0 or later, the following error is displayed:



Cause

This is an error thrown by the EMTDC Client side of Electric Network Interface (ENI) configuration. The description is as follows:

WSAECONNREFUSED 10061	Connection refused. No connection could be made because the target computer actively refused it. This usually results from trying to connect to a service that is inactive on the foreign host—that is, one with no server application running.

This error can occur if the Windows firewall or anti-virus program is too aggressive. This may be tested by shutting off the anti-virus and firewall, then running PSCAD again.

Solution

Add PSCAD to the firewall or anti-virus whitelist of trusted programs.

7.24 Receiving an "unable to execute a simulation executable for namespace '*'" Error Message

Problem 1

When compiling a PSCAD case using an Intel Fortran compiler, the following errors are displayed:

Description
LINK : fatal error LNK1104: cannot open file 'dfor.lib'
NMAKE : fatal error U1077: ' "C:\Program Files (x86)\Microsoft Visual Studio *\VC\BIN\linkexe" ' : return code '0x450'
Unable to generate a simulation executable for namespace '*'

Cause 1

These errors are due to a compatibility issue: You are using an Intel Fortran compiler to compile a case that contains a library that was precompiled using the Compaq Fortran compiler. It is not supported to run a case using a compiler that is different from the precompiled library.

Solution 1

The preferred solution is to obtain an Intel Fortran-precompiled copy of this library, and set it up as follows:

- Unload the Compaq Fortran-precompiled library and the project from your PSCAD workspace.
- Load the Intel Fortran-precompiled library and the project (in that order) into your PSCAD workspace.
- Clean the project directory.
- Run your case.

The least preferred solution is to run your case using Compaq Fortran 6. There are two things to note with this solution:

- Compaq is not supported as of PSCAD v4.6 and later.
- Compaq is old technology and may be difficult to install.

Problem 2

When compiling a PSCAD case using an Intel Fortran compiler, the following build message is displayed:

> \S was unexpected at this time.
> Unable to generate a simulation executable for namespace '*'

Or similar to the following message:

> 'MySQL' is not recognized as an internal or external command,
> Unable to generate a simulation executable for namespace '*'

The Fortan Medic utility will detect this issue: refer to Appendix A.5, Item #14.

Cause 2

The PATH environment variable key for a software contains the "&" symbol. This symbol can affect PSCAD, Intel Fortran, Microsoft Visual Studio, and other software.

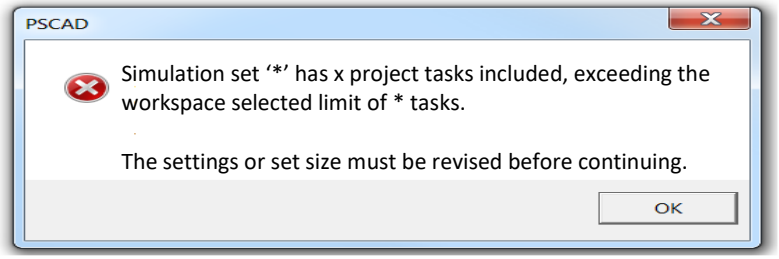
Solution 2

- Uninstall the software that contains the "&" symbol in the PATH environment variable key.
- Re-install this software, but during the installation, select a folder path name that does not include the "&" symbol. For example, replace "&" with "and".

7.25 Receiving a "Simulation set '*' has xx project tasks included, exceeding the workspace selected limit of * tasks" Error Message

Problem

When running a simulation set in PSCAD, the following error is displayed:



Cause

This error is due to exceeding the maximum number of projects that may be included in a simulation set.

Solution 1

Decrease the number of projects in your simulation set as required.

Solution 2

Update your PSCAD version and/or licensing configuration in order to accommodate the desired number of projects in your simulation set. The following specifications may be used to determine the best setup:

- For PSCAD v4.5.3, a limit of 8 projects per simulation set is imposed.
- For PSCAD v4.5.4 and later using lock-based licensing, four projects per simulation set are allowed.
- For PSCAD v4.5.4 and later using certificate-based licensing:
  - A base of 4 projects per simulation set is allowed.
  - If additional instances of EMTDCs[1] are purchased, a simulation set may run the equivalent number of additional simulation projects.

[1] A new feature introduced at v4.6.0 called Electric Network Interface (ENI) is an example of one feature that allows the use of multiple processor cores for running PSCAD. The number of cores that may be used is limited to the number of "Instances of EMTDCs". Additional EMTDCs may be purchased from our sales desk (sales@pscad.com). For further information on this feature, refer to the following references:

https://hvdc.ca/knowledge-base/read,article/41/network-splitting-using-electric-network-interface-eni/v:

https://hvdc.ca/knowledge-base/read,article/365/computer-cpu-cores-and-instances-of-emtdcs/v:

"What's New in PSCAD v4.6.0": https://hvdc.ca/knowledge-base/read,article/229/what-s-new-in-pscad-at-each-version/v:

7.26 Running a PSCAD v4.6 Simulation on Windows 10 is slower than running earlier versions of PSCAD

Problem

When running simulations on Windows 10, using v4.6 is sometimes slower than using an earlier version of PSCAD. Earlier versions of PSCAD may also be affected.

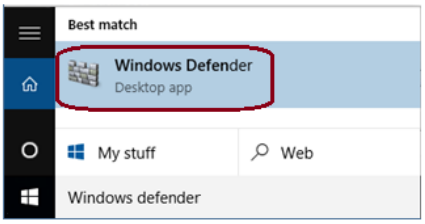
Cause

Windows 10 defender, which is the anti-virus that is released with Windows 10, can slow the simulation for v4.6.

Solution 1

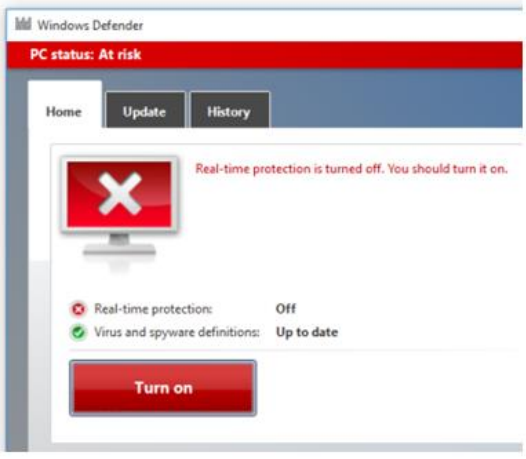
Turn off the Windows 10 Defender, as follows:

- From the Start menu, open Windows Defender.



- In the Windows Defender dialog box, select “Settings”.

- Turn off Windows Defender, otherwise known as “Real-time protection”:

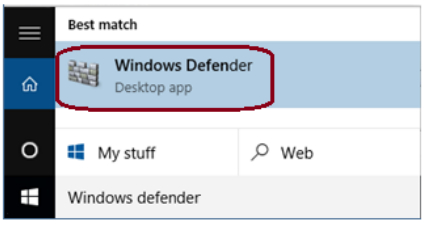


- Restart the computer.
- Retry running the simulation, to see if there is an improvement in the speed.

Solution 2

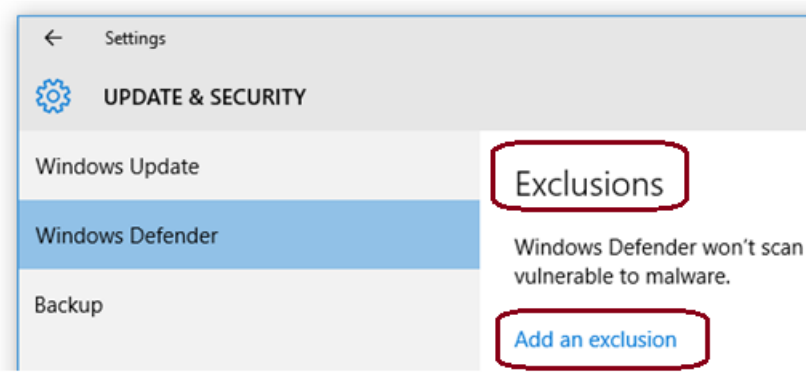
Add the PSCAD v4.6 program folder to the Windows Defender exclusion list, as follows:

- From the Start menu, open Windows Defender.

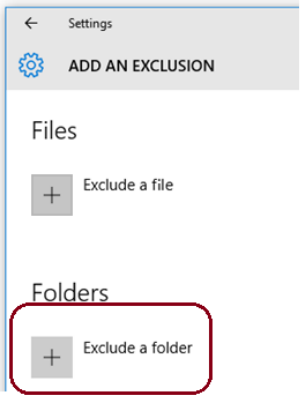


- In the Windows Defender dialog box, select “Settings”.

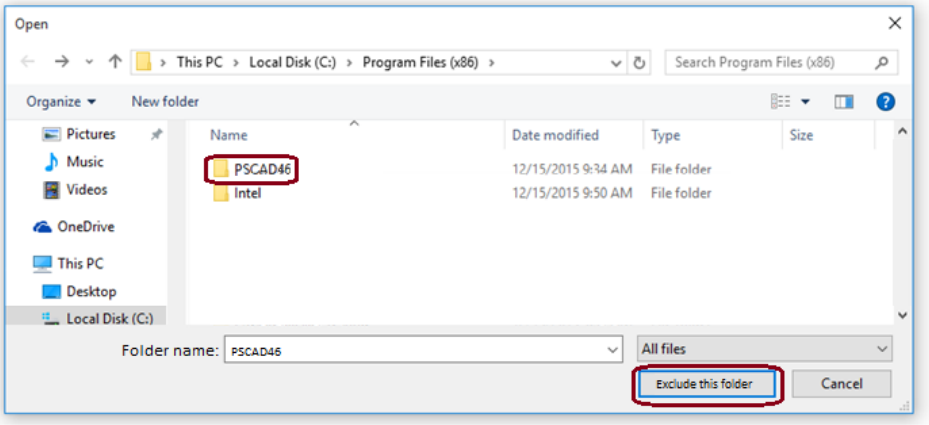
- Scroll down to “Exclusions”, and select “Add an exclusion”.



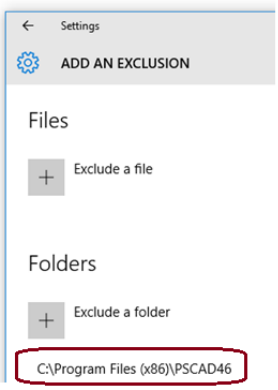
- Select “Exclude a folder”.



- When prompted, select a folder to exclude, and then select “Exclude this folder”.



- The selected folder should be listed in the excluded folders listing.



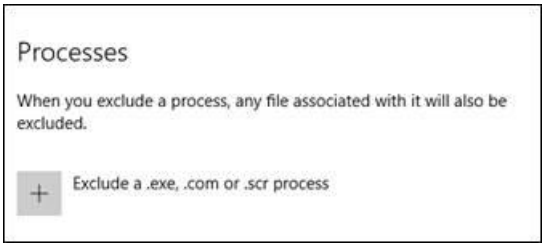
- Restart the computer.
- Retry running the simulation, to see if there is an improvement in the speed.

Solution 3

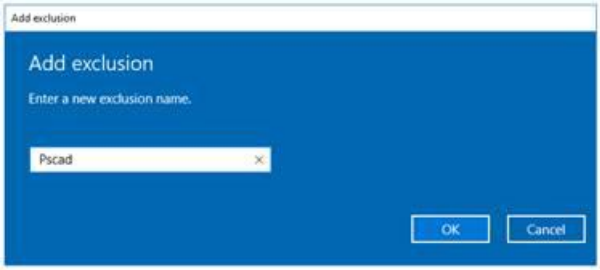
Add the PSCAD executable to the Windows Defender exclusion list, as follows:

- From the Windows Start menu, browse to and select the “Windows Defender Settings” link.
- Select “Add an exclusion”:

- Select “Exclude a .exe, .com or .scr process”:



- Enter “PSCAD” in the field, and select “OK”:



- Exit the Windows Defender settings.

7.27 Receiving an "'nmake' is not recognized as an internal or external command…" Error Message

Problem (1)

When compiling a PSCAD case using the Intel Fortran compiler, the following build errors display:

> Creating EMTDC executable…
> C:\Users\...
> ERROR: Visual Studio 2010, 2012 or 11 is not found in the system.
> 'nmake' is not recognized as an internal or external command,
> operable program or batch file.

Cause (1)

The INTEL_LICENSE_FILE environment variable has duplicate path segments, such as:

C:\Program Files (x86)\Common Files\\Intel\Licenses;C:\Program Files (x86)\Common Files\Intel\Licenses

Which is effectively:

C:\Program Files (x86)\Common Files\\Intel\Licenses;

C:\Program Files (x86)\Common Files\Intel\Licenses

Solution (1)

The Fortran Medic may be used to replace the duplicated path segments with a single path segment, which in the above example, would be

C:\Program Files (x86)\Common Files\Intel\Licenses

> **Note**
> Double slashes, \\, are converted to single slashes prior to any comparison for duplicates.

For more information on using the Fortran Medic for this repair, see Appendix A.5, Item 11.

Cause (2)

Previous additions to, or modifications to, the Windows PATH by installing a FORTRAN compiler have not been recognized by Windows.

Solution (2)

Log out then log back in on your machine (preferred), or restart your machine. Then retry compiling PSCAD.

Problem (2)

The following build messages display in the PSCAD application when trying to run a case using PSCAD v4.6 and Intel Fortran v17:

Creating EMTDC executable…
C:\…
Intel® MPI Library 2017 Update 1 for Windows* Target Build Environment for Intel® 64…
Copyright (C) 2007-2016 Intel Corporation. All rights reserved.
ERROR: Visual Studio 2012, 2013 or 2015 is not found in the system.
'nmake' is not recognized as an internal or external command,
operable program or batch file.
Unable to generate a simulation executable for namespace '*'

Cause (2)

Prerequisite software was missing. Specifically, Microsoft Windows SDK for Windows 8.1 should have been installed prior to installing the compiler.

Solution (2)

Install missing prerequisite software, then re-install Intel Fortran. Ensure to log out and log back in following these installations, to apply the changes.

7.28 Receiving a " 'C:Program' is not a valid internal or external command" Error Message

Problem

When building a PSCAD case, the following build error is displayed:

'C:Program' is not a valid internal or external command

When the latest Fortran Medic tool is run, an error message is displayed as indicated in Appendix A.5, Item 15.

Cause

The computer is not correctly generating short filenames.

Version

This issue affects PSCAD v4.5.3.

Solution

This issue has been resolved in PSCAD v4.5.4. Update your software to v4.5.4 or later.

7.29 Receiving a "libeng – Library is missing" Error Message

Problem

When building one of the Matlab examples that comes bundled with PSCAD, a build error similar to the following is displayed:

libeng – Library is missing

Cause

When Matlab R2015a (64-bit) or better is installed, but not the 32-bit version, PSCAD incorrectly thinks that the 32-bit version of Matlab was also installed. PSCAD would then incorrectly list them both as the following in the application:

R2015b, 8.5, (64-bit): This points to the 32-bit path, in which a 32-bit version SHOULD be installed, but is not installed.

R2015b, 8.5, (64-bit): This correctly points to the 64-bit path to where the 64-bit version IS installed.

The issue arises because regardless which installation is selected in the PSCAD application, the code will always select the first match from the list, which points to the non-existent 32-bit installation, and the error will display when the case is run.

Version

PSCAD v4.6.0 (64-bit), together with Matlab R2015a or better

Solution

- Use PSCAD v4.6.1 or later (this issue is corrected in these versions), or
- Use PSCAD v4.6.0 or earlier, and install both the 32-bit and the 64-bit versions of Matlab R2015a or better (unverified)

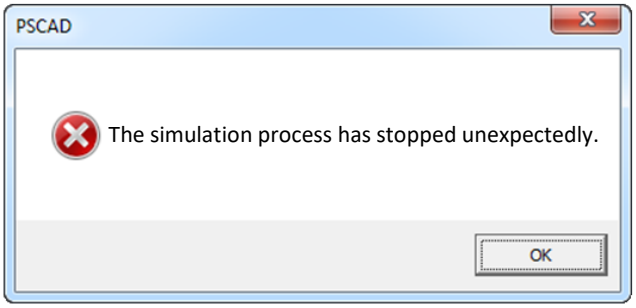
7.30 Receiving a "The simulation process has stopped unexpectedly" error message when running any simulation

Problem

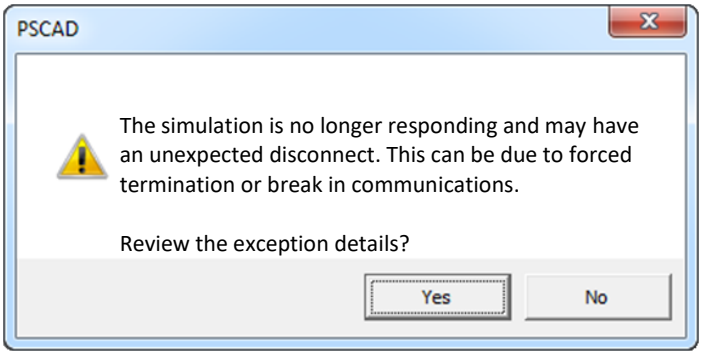
The following build error message is displayed when running any simulation:

> The simulation has stopped unexpectedly.
> The simulation is no longer responding and may have an unexpected disconnect. This can be due to forced termination or break in communications.
> Review the exception details?

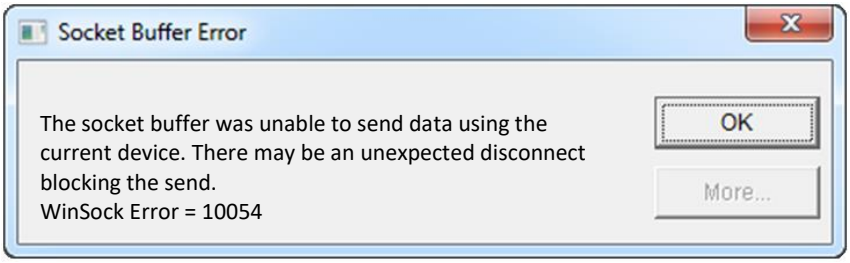
The following notification diplays:



When "OK" is selected above, the following dialog displays:



When "Yes" is selected above, the following dialog displays:



Furthermore, the build messages indicate that Visual Studio 2015 version 221 is being used in the simulation:

> Intel(R) Parallel Studio XE 2015 Update 4 Composer Edition (package 221)
> Microsoft (R) Program maintenance utility version 14.00.23506.0
> Copyright (C) Microsoft Corporation. All rights reserved.

Furthermore, the run messages display the following error message:

> Spawn_command "*.exe –v4 localhost * locale=English-us"

Also refer to Appendix A, Item 16 to see the message that is displayed in the Fortran Medic utility.

Cause

This error is due to incompatible software:

If Intel Fortran Composer 15.0.221 and earlier detects that Microsoft Visual Studio 2015 (v14) is installed, it will try to use that software for the compilation. However, these software programs do not appear to be compatible, and as a result, the simulation terminates.

> **Note**
> Intel Fortran Composer 15.0.208 and later is compatible with Microsoft Visual Studio 2015, according to Intel Fortran Release Notes.

Solution

Two options are as follows:

- To retain Microsoft Visual Studio 2015 (v14), switch out your compiler:
  - Install a compatible version of Intel Fortran (e.g. v15.0.208 or later, or v16). During this installation ensure to integrate Intel Fortran with the installed VS2015.
  - Log out then log back in on your computer to apply the changes.
  - Launch PSCAD, and set PSCAD to use the newly installed compiler (in the Application Options dialog box, display the Dependencies page, and select the compiler from the "Fortran Compiler…Version" field).
- Or, to retain Intel Fortran 15.0.221 or earlier, switch out your Visual Studio version:
  - Uninstall Visual Studio 2015 (or at least rename the VS installation folder in C:\Programs Files (x86) so that Intel Fortran does not try to use it).
  - Install a compatible version of Visual Studio software (VS2010, VS2012 or VS2013).
  - Re-install Intel Fortran 15.0.221 or earlier, ensuring to integrate it with the new VS software.
  - Log out then log back in on your computer to apply the changes.
  - Launch PSCAD, and ensure PSCAD is still set to use this compiler (in the Application Options dialog box, display the Dependencies page, and select the compiler from the "Fortran Compiler…Version" field).

7.31 Receiving a “No rule to make target ‘*.mak’. Stop” error message

Problem

The following build error messages are displayed when running any PSCAD case:

```
C:\Windows>call “*.bat”
CMD does not support UNC paths as current directories.
make: *.mak: No such file or directory
make: *** No rule to make target ‘*.mak’. Stop.
Unable to generate a simulation executable for namespace '*'
```

Or, the following, slightly different build error messages are displayed when running any PSCAD case:

```
make: *** No rule to make target ‘*.exe’. Stop.
Unable to generate a simulation executable for namespace '*'
```

Cause

The project files are being run either from “D” drive, or from a network folder:

- PSCAD can never run projects from network folders.
- PSCAD can sometimes not run projects from “D” drive.

Solution

Move your project files into a local folder on your C drive (e.g. on your desktop), load the project from your local folder, and run the simulation.

7.32 Receiving a Syntax Error Message when Compiling with GFortran

Problem

A build error message similar to the following is displayed when running any PSCAD case:

```
CALL E_VARIMPBRANCH( (IBRCH(3)+10),SS(3),1,0,0,R_B,,0.0)
                                                  1
Error: Syntax error in argument at (1)
```

The correct text should look something like this:

```
CALL E_VARIMPBRANCH( (IBRCH(3)+10),SS(3),1,0,0,R_B,1.0,1.0)
```

Cause

PSCAD is incorrectly generating Fortran code with missing parameters when using GFortran 4.2.1 or 4.6.2, which generates compile errors.

Solution

- Update to PSCAD 4.6.1 or better (if available), or
- Install and use an Intel FORTRAN compiler

7.33 Receiving a "severe (41): insufficient virtual memory" Error Message

Problem

When running a PSCAD case, the following error displays in the build messages:

forrtl: severe (41): insufficient virtual memory

Cause 1

Your project is exceeding the 2 GB memory limit imposed on any 32-bit application. Your simulation is demanding more memory than is physically possible to adress. For more information on this, please refer to the following link:

https://en.wikipedia.org/wiki/2_GB_limit

Solution 1

- Decrease the size of your network, or
- Switch to a PSCAD 64-bit application, available as of PSCAD v4.5. To pdate your software, please forward your license number to our Sales Desk (sales@pscad.com).

  Note

  See Cause 2 to ensure that you have enough RAM for your 64-bit application.

Cause 2

Your computer does not have enough RAM installed.

Solution 2

- Use PSCAD v4.5.3 or later, as this is optimised to handle large cases, and/or
- Ensure that you have adequate RAM for your case size. As a general rule, the Windows operating system requires 2 GB, the PSCAD GUI requires 2 GB to load the case and set up the memory for the simulation run, and the simulation and frequency scan require 2 to 4 GB of RAM, depending on network size.

7.34 Compiling any Case Containing Many Transmission Lines is Extremely Slow

Problem

Compiling cases containing many transmission lines is extremely slow to solve.

Version

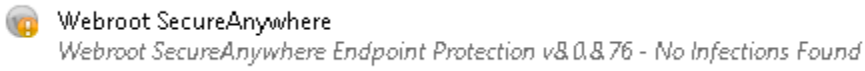
Applicable to PSCAD v4.5.0 and later.

Cause

The anti-virus, Webroot, is installed on your machine, and is causing extremely slow simulations. This software forces PSCAD to solve all lines in series rather than in parallel, and it starts solving them quite slowly.

This issue is applicable to PSCAD v4.5.0 and later, as the feature of solving transmission segments in parallel was first introduced at v4.5.0.

To determine whether you have Webroot installed on your machine, look for the icon in the notifications taskbar, similar to the following:



Solution

Webroot should be removed from your computer if you are running PSCAD cases with many transmission lines.

7.35 Receiving a "gcc.exe: C:PROGRA~…lib: No such file or directory" Error Message

Problem

Compiling a PSCAD case results in the following error in the Build Messages:

gcc.exe: C:PROGRA…lib: No such file or directory

Possible Cause

An installed compiler, OpenModelica, might be conflicting with the GFortran compiler that is used with PSCAD.

Solutions (unverified)

Try one of the following:

Note
You will require admin privileges on your machine to perform the first two options.

- Rename the OpenModelica folder:
  - From: C:\OpenModelica1.9.1
  - To: C:\OpenModelica1.9.1_
- Uninstall OpenModelica.
- Run PSCAD and GFortran on a different machine.

7.36 Receiving an "Error code 0x458" Message

Problem

Error messages similar to the following display when compiling PSCAD cases with Intel Fortran:

```
Compiling 'station.f' into object code.
Compiling 'main.f' into object code.
Linking objects and libraries into binary '*.exe'
 ...error LNK****: unresolved external symbol…
*.exe : fatal error LNK1120: * unresolved externals
NMAKE : fatal error U1077: '"C:\Program Files (x86)\Microsoft Visual Studio… \link.exe"' : Return code
  '0x460'
Stop.
Unable to generate a simulation executable for namespace '*'
```

Note
Refer to Section 7.43 for another issue related to Visual Studio 2015.

Cause

Due to some changes made to the libraries as of Microsoft® Visual Studio 2015, PSCAD must be configured to use one of two program files:

- The default configuration upon PSCAD installation is for using earlier versions of Visual Studio (VS2010, VS2012, VS2013).
- The new configuration is for using newer versions of Visual Studio (VS2015+).

If PSCAD is configured to use the wrong VS version, the build will fail, and errors similar to the above will display. The solution, listed below, contains instructions for configuring PSCAD to use the appropriate program file.

Note
The program file for VS2015+ is only supported in PSCAD v4.6.1 and newer; older versions of PSCAD do not support using Visual Studio 2015+.

Note
Objects or libraries containing c-code precompiled with the earlier versions of Visual Studio (VS2010, VS2012, VS2013) are not compatible with VS2015. The reverse is true as well (objects or libraries containing c-code precompiled with VS2015 are not compatible with VS2010, VS2012, VS2013).

For any objects of libraries not containing c-code, they may be precompiled and then run using any of the above versions. There should not be any compatibility issue.

Applicability

The solution to this issue is applicable to the following:

- PSCAD v4.6.1 and newer;
- A supported version of Intel Fortran compiler (as specified in the Compatibility Charts, Chart 4);
- Visual Studio 2015 and newer

Solution (1.a) – Configuring for Visual Studio 2015+ on a Windows 64-bit Machine (Automatically using the Fortran Medic)

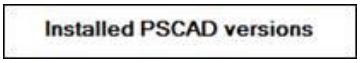
Use the Fortran Medic tool to automatically configure PSCAD to use Visual Studio 2015+ on a Windows 64-bit machine as follows:

Warning
If you proceed with this action, PSCAD will no longer be configured for using Visual Studio 2010, 2012, or 2013.

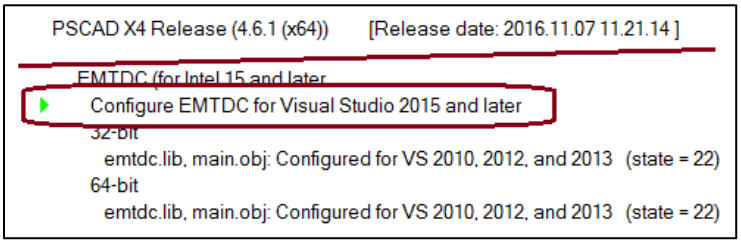
Notes
1. Windows Administrator privileges are required.

2. Applicable for v4.6.1 and newer (not supported with earlier versions).

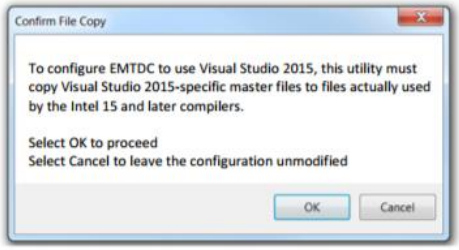
- Run the Fortran Medic tool as per Appendix A.2, and scroll down to the following heading:



- Under the PSCAD 64-bit installation (v4.6.1 or newer), right-click on the message as shown below:



- The following dialog will display, select "OK" to proceed:



- Your environment will be configured to use Visual Studio 2015 and newer.

Notes
1. This function does not install Visual Studio software, it merely configures PSCAD to be able to use Visual Studio 2015 and newer.

2. Configuration with Visual Studio 2010, 2012, and 2013 may be restored as per Appendix A.6, Function 2.

Solution (1.b) – Configuring for Visual Studio 2015+ on a Windows 32-bit Machine (Automatically using the Fortran Medic)

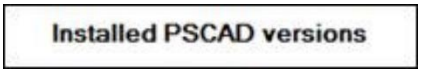
Use the Fortran Medic tool to automatically configure PSCAD to use Visual Studio 2015+ on a Windows 32-bit machine as follows:

Warning
If you proceed with this action, PSCAD will no longer be configured for using Visual Studio 2010, 2012 or 2013.

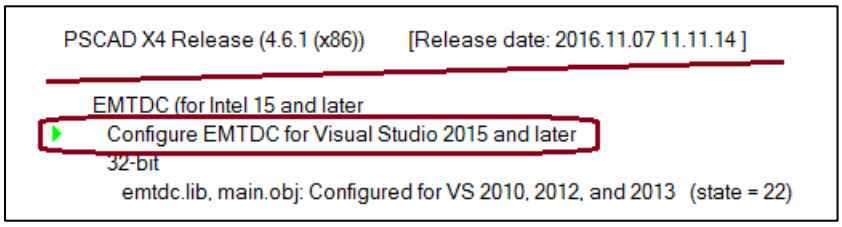
Notes
1. Windows Administrator privileges are required.

2. Applicable for v4.6.1 and newer (not supported with earlier versions).

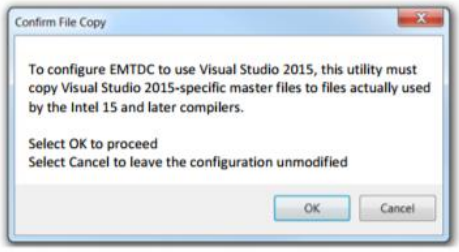
- Run the Fortran Medic tool as per Appendix A.2, and scroll down to the following heading:



- Under the PSCAD 32-bit installation (v4.6.1 or newer), right-click on the message shown below:



- The following dialog will display. Select "OK" toproceed:



- Your environment will be configured to use Visual Studio 2015 and newer.

Notes
1. This function does not install Visual Studio software, it merely configures PSCAD to be able to use Visual Studio 2015 and newer.

2. Configuration with Visual Studio 2010, 2012, and 2013 may be restored as per Appendix A.6, Function 2.

Solution (2) – Configuring for Visual Studio 2015+ on a Windows 32-bit or 64-bit Machine (Manually)

Manually change your environment in order to use Visual Studio 2015 and newer.

Warning
This is not the recommended solution, as errors could be introduced to your environment when performed manually. We recommend following the automatic instructions in Solution (1.a or 1.b) above, instead.

Notes
1. Windows Administrator privileges are required
2. Applicable for v4.6.1 and newer (not supported with earlier versions)
3. Ensure to perform all changes, for both the main.obj and emtdc.lib files, and for both the 32-bit and 64-bit applications.

1. Changes to the "main.obj" files (64-bit):

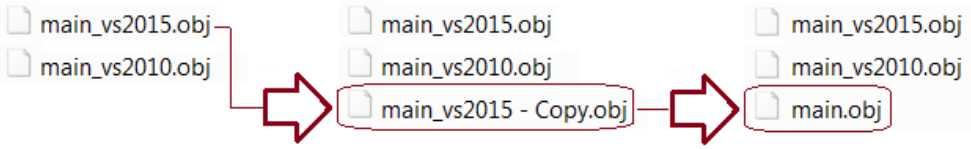
- Open a Windows file browser to the following folder:

  <PSCAD Install folder>\emtdc\if15\windows

- If using v4.6.1, locate the following two files, and rename the "main.obj" file as "main_vs2010.obj":

- If using v4.6.2 or newer, locate the following three files, and delete the "main.obj" file:

- Then, for v4.6.1 and newer, make a copy of the "main_vs2015.obj" file, and rename it as "main.obj":



2. Changes to the "main.obj" files (32-bit):

- Open a Windows file browser to the following folder:

  <PSCAD Install folder>\emtdc\if15_x86\windows

- Make the same changes to the main object files as performed in Step 1 above.

Continued…

3. Changes to the "emtdc.lib" files (64-bit):

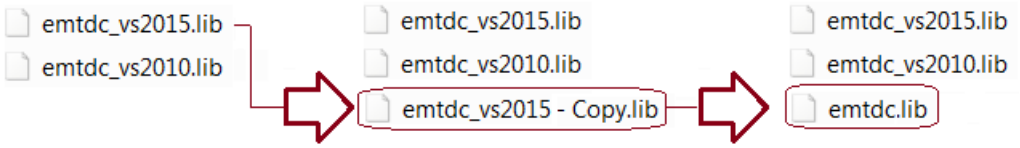
- Open a Windows file browser to the following folder:

  <PSCAD Install folder>\emtdc\if15\windows

- If using v4.6.1, locate the following two files, and rename the "emtdc.lib" file as "emtdc_vs2010.lib":

- If using v4.6.2 and newer, locate the following three files, and delete the "emtdc.lib" file:

- Then, for v4.6.1 and newer, make a copy of the "emtdc_vs2015.lib" file, and rename it as "emtdc.lib":



4. Changes to the "emtdc.lib" files (32-bit):

- Open a Windows file browser to the following folder:

  <PSCAD Install folder>\emtdc\if15_x86\windows

- Make the same changes to the emtdc library files as performed in Step 3 above.

Notes

1. This function does not install Visual Studio software, it merely configures PSCAD to be able to use Visual Studio 2015 and newer.

2. Configuration with Visual Studio 2010, 2012, and 2013 may be restored as per Appendix A.6, Function 2.

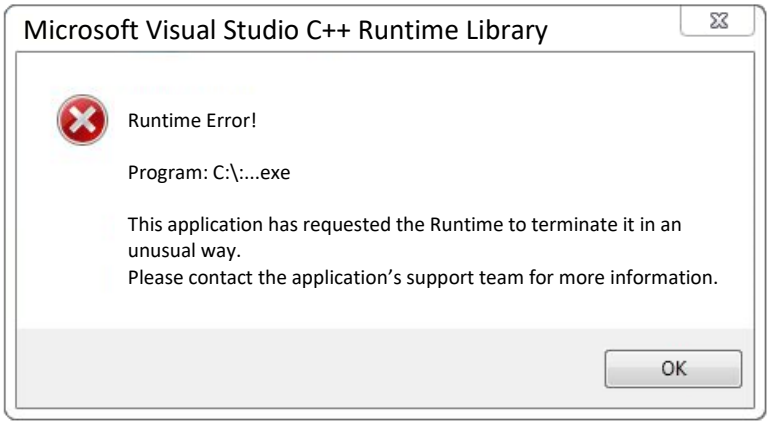
Solution (3) – Configuring for Visual Studio 2013 and older (Automatically using the Fortran Medic)

Refer to Appendix A.6 Item 2.

7.37 Receiving a "The simulation process has stopped unexpectedly" Message

Problem

The following dialog displays:



The following build messages display in the PSCAD application:

Creating EMTDC executable…
C:\....bat
Linking object code and libraries into binary '*.exe'
The simulation process has stopped unexpectedly.

Cause

A "Single Sign On Engine" (SSO) application is installed on this machine, and apears to halt PSCAD from building a simulation.

It is suspected that SSO restricts which executable files are allowed to be run on a machine. If so, SSO prevents PSCAD from running the executable files that PSCAD creates when building a case. There is no practical way to whitelist these executables ahead of time as the executable names depend on the existing PSCAD case names and new case names as the user creates more PSCAD cases.

To see the messages that are displayed when the Fortran Medic tool is run, refer to Appendix A.5, Item 20.

Solution (1)

A possible solution would be to confine all PSCAD cases to a particular folder, (e.g. C:\PSCAD), and to see if the SSO may be configured to allow all executable files to run from that folder without restriction. This would be analogous to giving a 'sandbox folder' to PSCAD which would not be monitored by the SSO.

Solution (2)

Remove SSO from your computer.

7.38 Receiving a "Main.f(6) : fatal error C1034" Error Message

Problem

The following build messages display in the PSCAD application when trying to run a PSCAD v4.2.1 case using Intel Fortran:

```
Creating EMTDC executable
...
    Compiling Main.c
        cl: Command line warning D9035 : option 'GX' has been deprecated and will be removed in a future release
        cl : Command line warning D9036 : use 'EHsc' instead of 'GX'
        cl : Command line warning D9002 : ignoring option '/ML'
        Main.f(6) : fatal error C1034: f2c.h: no include path set
        NMAKE : fatal error U1077: "'c:\program files (x86)\microsoft visual studio 9.0\vc\bin\cl.exe"' : return code '0x2'
        Stop.
```

See Appendix A.5, Item 21 to see the corresponding Fortran Medic error.

Cause

GIT and Mingw are installed and are causing this error, and are causing PSCAD to fail when compiling cases with Intel Fortran.

Solution (1)

Rename the GIT folder.

Solution (2)

Remove GIT and MingW.

Solution (3)

Install PSCAD and Intel Fortran on a different machine, one that does not also have GIT and MingW.

7.39 Receiving a "Unable to Generate a Simulation" Error Message

Problem

The following build messages display in the PSCAD application when compiling a PSCAD case with an Intel Fortran compiler:

Unable to generate a simulation executable for namespace '*'

See Appendix A.5, Item 24 to see the corresponding Fortran Medic error.

Cause

A file path contains a '+', which causes the Intel Fortran compiler to halt.

> **Note**
> The & symbol is also not allowed in a file path.

Solution (1)

Run the Fortran Medic utility as per Appendix A.2, then use the utility to remove the path containing the '+' as per Appendix A.5, Item 24.

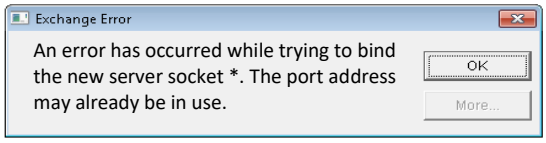
Solution (2)

Identify the software which requires the '+' character, then remove the software, and re-install the software into a folder which does not contain the '+' character.

7.40 Receiving an "Exchange Error" Message

Problem

When building a PSCAD case, the error below displays. Restarting PSCAD sometimes resolves this.



Background

When PSCAD (GUI) starts a simulation (project), a socket connection is created so that PSCAD and the simulation can exchange data. PSCAD will randomly pick a communications socket on the computer to do this. When this socket is not available, the above error will display. Three typical causes for a socket not being available are listed below.

Cause (1): A Crashed Simulation is still running in the Background

Sometimes when a simulation has crashed, the simulation will be stuck running in the background, and will still hold a communications socket open. If PSCAD is run again, it may try to use that same socket, but it will not be available.

Solutions (1): A Crashed Simulation is still running in the Background

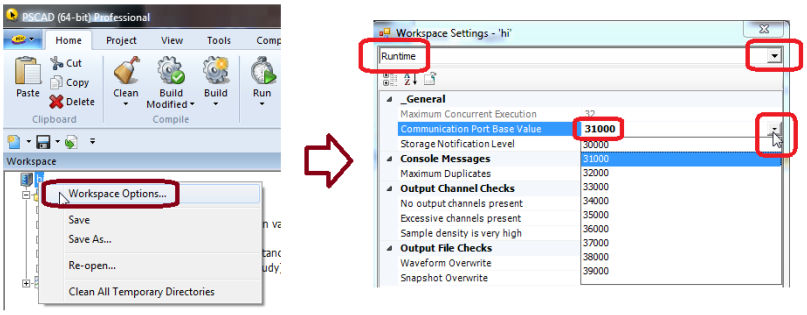
- Display the Task Manager and end any simulations running in the background to free up the socket; or
- Simply restart the computer to end any simulations running in the background to free up the socket.

Cause (2): Running a Second Instance of PSCAD

Two instances of PSCAD are being run, and both instances are trying to use the same communications socket.

Solution (2): Running a Second Instance of PSCAD

Change the port range for one of the PSCAD instances as shown:



Cause (3): Aggressive Protection Software

Aggressive anti-virus or firewall software is preventing this communication.

Solution (3): Aggressive Protection Software

Test whether aggressive protection software is the cause by shutting it off, then building the project. If this resolves the matter, ask your IT staff to white list PSCAD.

7.41 Receiving an "Unable to generate a simulation executable for namespace" Error Message

Problem

When building a PSCAD case, the following error displays:

Unable to generate a simulation executable for namespace '*'

Cause

Anti-virus software is preventing PSCAD from compiling the case.

Specifically, Avast has been known to do this.

Solution

White-list PSCAD activities in Avast.

7.42 Receiving an "Error LNK2005" Error Message

Problem

When building a PSCAD case, the following error displays in the Build messages:

LIBCMTD.lib (*.obj) : error LNK2005: * already defined in LIBCMT.lib (*.obj)
Unable to generate a simulation executable for namespace '*'
Make failed to generate a simulation executable for '*'. Binary file was not found.

Cause

The EMTDC configuration file is outdated.

Applicability

PSCAD v4.5.1 to v4.5.3

Intel Fortran Composer XE 2013

Solution

- Update your software (sales@pscad.com), or
- Retain your software version, but update your EMTDC configuration file:

  - Request the file from PSCAD Support (support@mhi.ca). Ensure to include your PSCAD version and license number.
  - Delete the existing "emtdc.cfg" file from this path:

    C:\Program Files (x86)\PSCAD453\emtdc\if12\windows

  - Save the file you receive from Support to the above path.

7.43 Receiving an "Error LNK2001" Error Message

Problem

When using VS2015 to compile a pre-VS2015 build library containing c-code, some errors similar to the following display in the Build messages:

error LNK2001: unresolved external symbol ___iob_func

error LNK2001: unresolved external symbol _printf

error LNK2001: unresolved external symbol _sprintf

error LNK2001: unresolved external symbol _sscanf

Or

When using pre-VS2015 to compile a VS2015 build library containing c-code, some errors similar to the following display in the Build messages:

error LNK2005: _sscanf already defined in emtdc.lib(datetime.obj)

error LNK2005: _printf already defined in main.obj

error LNK2001: unresolved external symbol ___acrt_iob_func

error LNK2001: unresolved external symbol ___stdio_common_vfprintf

Notes

1. "Pre-VS2015" refers Microsoft® Visual Studio 2013 and earlier, for example VS2010, VS2012 and VS2013.
2. "VS2015" refers to Microsoft® Visual Studio 2015.
3. These errors do not occur if the library contains no c-code.
4. Refer to Section 7.36 for another issue related to Visual Studio 2015.

Cause

Due to changes made to the Microsoft® Visual Studio library at VS2015, any PSCAD libraries created with an earlier version of Visual Studio (VS2013 and earlier) containing c-code are not compatible with VS2015 (and later).

Conversely, any PSCAD libraries created with VS2015 (and later) containing c-code are not compatible with VS2013 and earlier.

Note
For PSCAD libraries that do not contain c-code, this compatibility issue is not present.

Solution

Switch to a compatible version of Visual Studio. See Section 7.36 for PSCAD configuration.

7.44 Receiving a "The simulation process has stopped unexpectedly" message when Windows goes to sleep while a case is running

Problem

The following error is displayed when Windows goes to sleep (or hibernating) while a case is running:

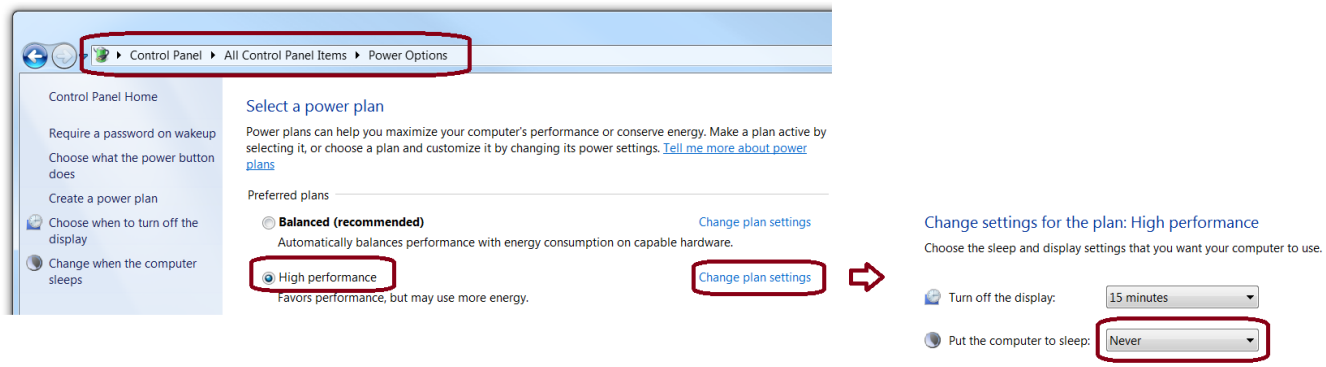
> The simulation has stopped unexpectedly.
> The simulation is no longer responding and may have an unexpected disconnect. This can be due to forced termination or break in communications.
> Review the exception details?

Cause

Windows is likely set to go to sleep after a specified period of time. This halts all processes, including the PSCAD simulation.

Solution

Whenever a computer will be unattended during a simulation (e.g. over lunch break, overnight, etc…), the Windows settings could be switched to not go to sleep. The settings for this are in the Windows Control Panel | Power Options. It is recommended to set Windows to high or full performance, and to never go to sleep, as shown in the following example:



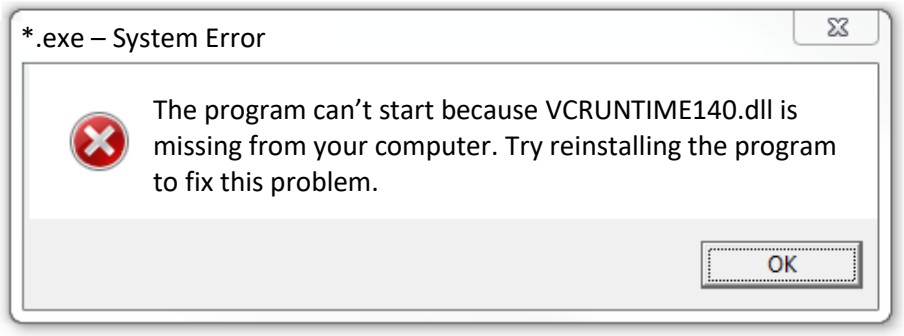
The settings may be switched back when the simulation is complete.

> **Note**
> If the above solution does not prevent the computer from going to sleep during a simulation, then a network policy is likely being imposed which overrides the Windows settings. In this case, your IT staff should be consulted.

7.45 Receiving a "System Error" message when compiling a PSCAD case

Problem

The following error is displayed when a simulation is run:



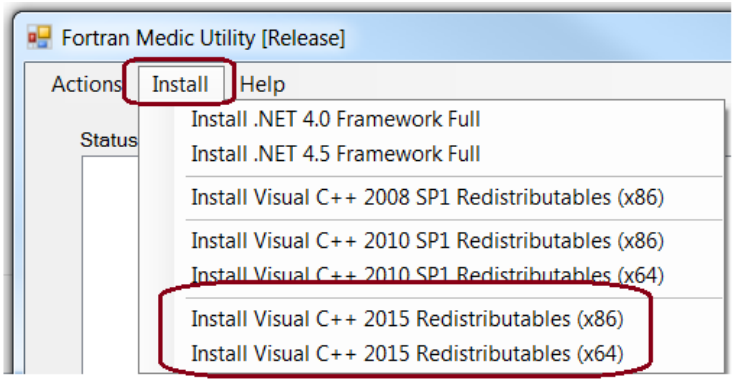
Cause

Required prerequisite software for using PSCAD v4.6.2 and later is missing. Specifically, Visual Studio C++ 2015 Redistributables are missing.

Solution

Install this software as follows:

- Download and launch the Fortran Medic tool as per Appendix A.2 (Steps (a) and (b)).
- From the "Install" menu, install both the x86 and the x64 editions of Visual Studio C++ 2015 Redistributables.



7.46 Receiving a "Visual Studio 2010, 2012 or 11 is not found in the system" Error Message

Problem

When compiling a PSCAD case, the following build errors display:

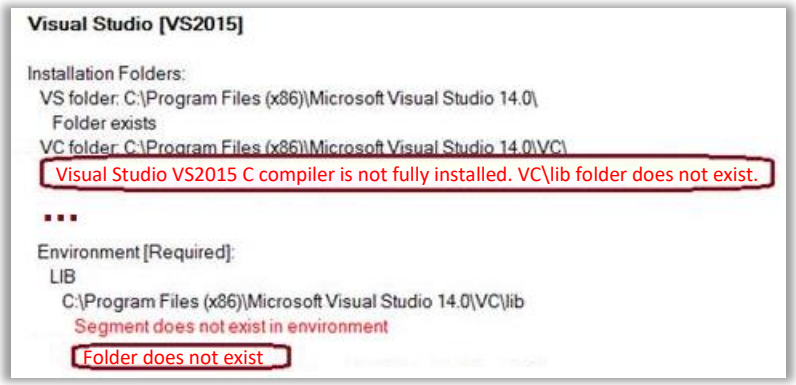
ERROR: Visual Studio 2010, 2012 or 11 is not found in the system.
NMAKE : fatal error U1077: 'ifort.exe' : return code '0x1'
Unable to generate a simulation executable for namespace '*'

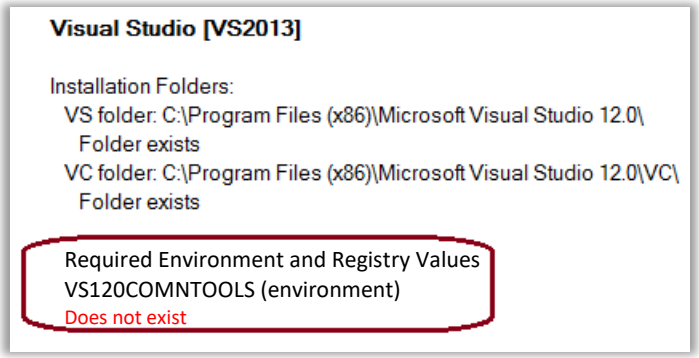
The above errors are present when using the following software:

- An Intel Fortran compiler, and
- One of the standalone editions of Visual Studio 2015+, for example the Professional Edition.

Note
The above errors are not present when using the Premier Partner (Shell) Edition of Visual Studio, which is free and comes bundled with a licensed Intel Fortran compiler.

And when the Fortran Medic tool is run on this machine (as per Appendix A.2), this tool displays errors similar to either of the following:





Cause

The Visual Studio software installation is incomplete, and required software is missing. This can occur when a default installation is performed, rather than a custom installation.

Solution

Reinstall the Visual Studio software, ensuring to select a “Custom” installation, and ensuring to install all of the required software. The following screenshot shows the required software, and should be similar to a user’s setup:

Following the installation, log out then log back in on the machine, then load and try compiling one of the simple PSCAD cases (C:\Users\Public\Documents\PSCAD\4.x\Examples\tutorial\vdiv.pscx).

If there are no errors, the setup is ready. Or, if there are any errors, Intel Fortran might require re-installation. During this installation:

- Select the “custom” rather than “default” installation on the “Options” page.
- If this is a licensed edition of the Intel Fortran compiler, de-select the Visual Studio software.
- On the integration page, select the appropriate Visual Studio edition and version.
- Log out then log back in on the machine.
- Test the setup by trying to run the simple PSCAD case again.

7.47 Receiving a "'make' is not recognized" error message when compiling with GFortran

Problem

When compiling a PSCAD case using a GFortran compiler, build errors similar to the following display:

```
C:\Users\Public\...\*.gf42>call…
'make' is not recognized as an internal or external command, operable program or batch file.
Unable to generate a simulation executable for namespace '*'
```

And when the Fortran Medic tool is run, the following error displays:

```
The PATHEXT environment variable is missing the .EXE extention
```

(see Appendix A.5 Item 27 for further information on this error in the Fortran Medic tool)

Cause

The ".exe" is missing from the PATHEXT environment variable.

Solution

Use the Fortran Medic tool to easily fix the PATHEXT environment variable as per Appendix A.5 Item 27.

7.48 Receiving a “make (e=5): Access is denied” Build Error

Problem

When compiling a PSCAD case when user has no Windows Administrator privileges, build errors similar to the following display:

```
Creaing EMTDC executable
C:\...\*.gf46>call “C:\Program Files (x86)\GFortran\4.6\bin\gf46vars.bat”
process_begin: CreateProcess(C:\...)…failed.
make (e=5): Access is denied.
make: *** [Station.o] Error 5
Unable to generate a simulation executable for namespace ‘*’
```

Cause

When a simulation is launched, executable files are created and launched as indicated in Appendix F. However, network policies can detect and block these actions if the user has no Windows Administrator privileges.

Solution (1)

Always launch PSCAD with Windows Administrator privileges (from the Windows Start menu, browse to and right-click on the PSCAD link, and select “Run as administrator”).

Solution (2)

Ask your IT staff to modify your network policy to allow a user to create and run batch files with normal Windows “User” privileges.

7.49 Security Software is preventing Simulations residing outside the Program Directories from running

Problem

Simulations will not run because security software, for example AppLocker, is configured to prevent creating or running executable files that reside outside the Program directories.

See Appendix F for a list of executable files that are created and launched during a simulation.

Solution (1)

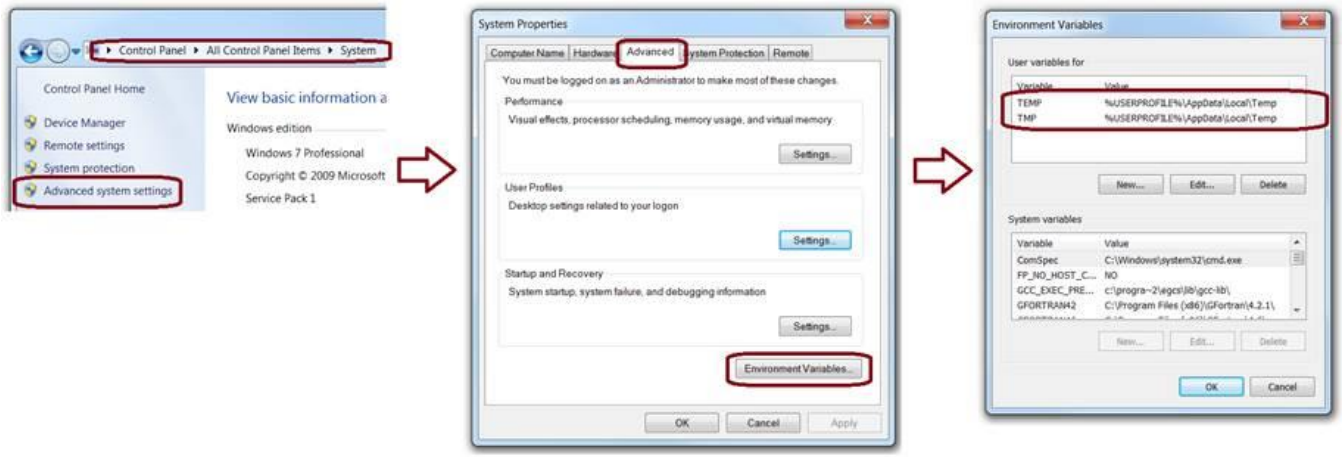
For future development (i.e. after v4.6.2)

The long term solution for future patches and releases is for PSCAD to create a modified <case name>mak.bat file which redirects the TMP and TEMP variables. When the compile is complete, the TMP and TEMP user variables revert automatically to the user's environment settings.

Solution (2)

For PSCAD v4.6.2 and earlier

Redirect where these batch files are created by changing where the TMP and TEMP user environment variables point to:



Here is an overview of this process (details follow):

- Create a folder called C:\XXX
- Set both the TMP and TEMP user environment variables to C:\XXX
- Log out then log back in on the machine
- Compile a PSCAD case using GFortran 4.2, or GFortran 4.6.2, or some Intel Fortran compiler, and the temp batch files will be created in the C:\XXX folder.

> **Caution**
> If the TMP and TEMP user environment variables are changed, then ALL temp files from ALL other programs will also be stored in the redirected folder, which in the above situation, would be C:\XXX.

Solution (3)

For PSCAD v4.6.2 any version

Run your PSCAD project files from within Program Files. This setup may be tested to see if this resolves the issue as follows:

> **Note**
> Windows Administrator privileges are required for this.

- Make a copy of the “simpleac” file, which resides in the following location:

  C:\Users\Public\Documents\PSCAD\4.6\Examples\tutorial

- Save the copied file to a new folder in the Program Files.
- Launch PSCAD with elevated privileges (from the Windows Start menu, browse to and right-click on the PSCAD link, and select “Run as administrator”).
- Load the simpleac example from Program Files, and run it.
- If the anti-virus does not prevent the case from running, you could move all of your cases to Program Files, and run them from there.

> **Note**
> To avoid having to have administrator rights, your IT staff could make the Program Files accessible to all users, rather than to just administrators.

7.50 Receiving an “Unable to generate a simulation executable for namespace ‘*’” Error Message

Problem

When trying to compile a PSCAD project, the build fails, with the following messages:

```
Creating EMTDC executable...
…
Intel(R) MPI Library 2018 Update 1 for Windows* Target Build Environment for Intel(R) 64 applications
Copyright (C) 2007-2017 Intel Corporation. All rights reserved.
Copyright (C) 1985-2017 Intel Corporation. All rights reserved.
Intel(R) Compiler 18.0 Update 1 (package 156)
**********************************************************************
** Visual Studio 2017 Developer Command Prompt v15.5.2
** Copyright (c) 2017 Microsoft Corporation
**********************************************************************
[vcvarsall.bat] Environment initialized for: 'x64'
Microsoft (R) Program Maintenance Utility Version 14.12.25831.0
Copyright (C) Microsoft Corporation. Tous droits réservés.
Linking objects and libraries into binary ‘*.exe'
Unable to generate a simulation executable for namespace '*'
======== Build: 3 errors, 0 warnings ========
```

System

Intel Fortran Composer 18.0.156

Visual Studio 2017 Professional Edition 14.12.25827

PSCAD v4.6.2

Cause

The “*” characters related to Visual Studio 2017 cause the compiling error.

Solution (1)

Update your softare - this issue is expected to be resolved in PSCAD v4.6.3+.

Solution (2)

Downgrade your version of Visual Studio to v2015.

7.51 Receiving an "Unable to generate a simulation executable for namespace '*'" Error Message

Problem

When trying to compile a PSCAD project with an Intel Fortran compiler, the editor is opened with a file called "echo.vbe", at which point the compiling pauses, and only resumes once the window is manually closed. This sequence repeats itself several times, depending on the project size.

Ultimately, the build fails, with the following message:

Unable to generate a simulation executable for namespace '*'

When the latest Fortran Medic tool is run as per Appendix A.2, the messages as shown in Appendix A.5 Item 30 are displayed.

This issue is not present when using the GFortran compiler to compile a PSCAD case.

Cause

A file called echo.vbe, installed by some unknown and unrelated third party software, is interfering when compiling a PSCAD project with Intel Fortran composer.

Solution

Perform a search on your machine for the file called echo.vbe. This file may be in C:\Windows. Then, either:

- Remove this file if you do not require it for another application, or
- Temporarily rename this file when using PSCAD. For example:

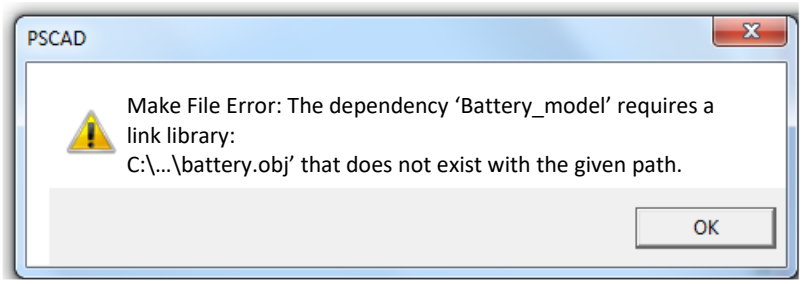
  Rename this file from echo.vbe to echo_1.vbe

  Note
  Windows administrator privileges are required to perform this.

7.52 Receiving a "Make File Error" Message when compiling a Project

Problem

When compiling a PSCAD project, the simulation halts, and a message similar to the following displays:



Cause

Some object files that the PSCAD project requires are missing. For example, the above screenshot indicates that the library associated with the Battery Model is missing.

Solution

Locate and link in the associated library. See Linking Objects and Libraries into PSCAD for more details.

7.53 Receiving an “Unable to execute make” Error Message

Problem

In PSCAD v4.5.4 or v4.6.0, when compiling a project that contains a blackbox module using the GFortran compiler, the simulation halts, and a message similar to the following displays:

"filename.mak:line numer *** target pattern contains no '%'. Stop.
Unable to execute make.

Note
This model runs in PSCAD v4.5.4 or v4.6.0 when using the Intel Fortran compiler.

Applicability

PSCAD v4.5.4 v4.6.0

GFortran compiler

Cause

The associated object (*.o) file fails to generate when using the GFortran compiler due to blackbox incorrectly using quotations in some of the file paths in the *.mak file.

Solution

Update your software to PSCAD v4.5.5, v4.6.1 or later.

7.54 Receiving a "…machine type 'X86' conflicts with target machine type 'X64'" Error Message

Problem

When compiling a case using the Intel Fortran compiler, build errors similar to the following display:

```
C:\...\*.obj : fatal error LNK1112: module machine type '86' conflicts with target machine type 'x64'
NMAKE : fatal error U1077: '"C:\...\Intel Fortran\...\link.exe"' : return code '0x458'
Unable to generate a simulation executable for namespace '*'
```

Or

```
Linking objects and libraries into binary '*.exe'
Main.obj : error LNK2019: unresolved external symbol * referenced in function *
C:\...\*.LIB : warning LNK4272: library machine type '86' conflicts with target machine type 'x64'
*.exe : fatal error LNK1120: 1 unresolved externals
NMAKE : fatal error U1077: '"C:\...\Microsoft Visual Studio 14.0\VC\BIN\amd64\link.exe"' : return code '0x460'
Stop.
Unable to generate a simulation executable for namespace '*'
```

Cause

A 64-bit edition of Intel Fortran is being used to run a project, however, the project has precompiled libraries that were created using a 32-bit edition of Intel Fortran. The same edition of Intel Fortran compiler used to create the libraries must also be used to run the project.

Solution (1)

Re-run your project using the 32-bit edition of Intel Fortran.

Solution (2)

If you have access to the source code for the libraries, re-compile the libraries using Intel Fortran 64-bit edition, then re-run the project using the 64-bit edition of Intel Fortran.

7.55 Receiving a "This version of *\gcc.exe is not compatible with the version of Windows you're running" Error Message

Problem

When compiling a case using the GFortran 4.2 compiler, the build fails, and the following messages display in the Build pane:

> Creating EMTDC executable…
>
> C:\...*.gf42><call "C:\Program Files (x86)\GFortran\4.2.1\bin\gf42vars.bat"
>
> Linking objects and libraries into binary '*.exe'
>
> This version of C:\Program Files (x86)\GFortran\4.2.1\bin\gcc.exe is not compatible with the version of Windows you're running. Check your computer's system information and then contact the software publisher.
>
> make: *** [*exe] Error 1
>
> Unable to generate a simulation executable for namespace '*'

Cause

Unknown at this time

Applicability

PSCAD v4.5.4 and v4.6.1 (may be applicable to other versions)

GFortran v4.2

Solution

Uninstall GFortran 4.2, re-install GFortran 4.2, log out and log back in your machine.

Test whether you can run one of the simple PSCAD examples with GFortran 4.2:

> C:\Users\Public\Documents\PSCAD*\examples\tutorial\vdiv.pscx

7.56 Receiving a "Visual Studio 2005, 2008 or 2010 is not found in the system" Error Message

Problem

When compiling a case using the Intel Fortran compiler, the build fails, and the following error displays in the Build pane:

ERROR: Visual Studio 2005, 2008 or 2010 is not found in the system.

When the Fortran Medic tool is run (Appendix A.2), an error similar to the one shown in Appendix A.5, Item 17 is displayed.

Cause

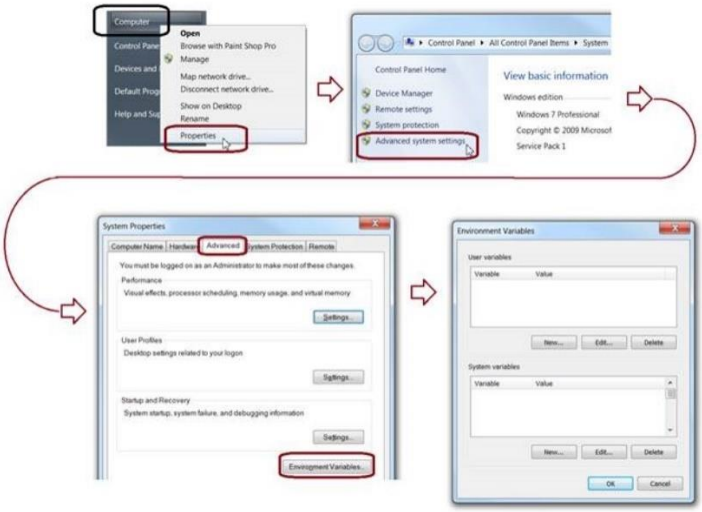
The required common tools environment variable, for example, VS90COMNTOOLS, is not defined on the machine.

Solution

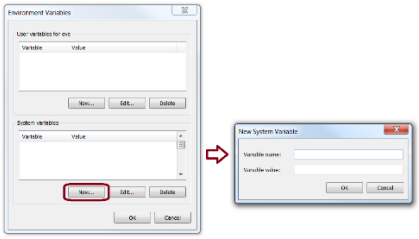
Preferred Solution: Use the Fortran Medic tool to create the environment variable as per Appendix A.5, Item 17.

If not possible to use the Medic tool to create the environment variable, this may be done manually as follows:

a. Display your Environment Variables:



b. Select New to create a new System Variable:



c. Enter the variable name and value as applicable:

Visual Studio version	Environment variable name	Default Variable value
VS 2008	VS90COMNTOOLS	C:\Program Files (x86)\Microsoft Visual Studio 9.0\Common7\Tools
VS 2010	VS100COMNTOOLS	C:\Program Files (x86)\Microsoft Visual Studio 10.0\Common7\Tools
VS 2012	VS110COMNTOOLS	C:\Program Files (x86)\Microsoft Visual Studio 11.0\Common7\Tools
VS 2013	VS120COMNTOOLS	C:\Program Files (x86)\Microsoft Visual Studio 12.0\Common7\Tools
VS 2015	VS140COMNTOOLS	C:\Program Files (x86)\Microsoft Visual Studio 14.0\Common7\Tools

d. Save the variable.

7.57 Receiving a "Cannot determine the location of the VS Common Tools folder" Error Message

Problem 1

When compiling a case using the Intel Fortran compiler and Visual Studio, the build fails, and the following Build Messages display:

```
ERROR: Cannot determine the location of the VS Common Tools folder.
LINK : fatal error LNK1181: cannot open input file '*.lib'
NMAKE : fatal error U1077: '"C:\Program Files (x86)\Microsoft Visual Studio 12.0\VC\bin\amd64\link.exe"' : return code '0x49d'
Unable to generate a simulation executable for namespace '*'
```

When the Fortran Medic tool is run (Appendix A.2), the error as indicated in Appendix A.5, Item 32 is displayed.

> **Note**
> By contrast, refer to Section 7.57 Problem 2, below, if the Medic does not detect an error with Visual Studio common tools folder.

Cause

The PATH environment is missing the following standard system paths:

- C:\Windows
- C:\Windows\System32

System

This error was detected for the following software combination, but is certainly not limited to this combination:

PSCAD v4.5.4, Intel Fortran compiler 17, Microsoft Visual Studio 2012

Solution

- Run the Fortran Medic tool as per Appendix A.2, then create / add the values as per Appendix A.5, Item 32.
- Log out, then log back in to Windows. No need to reboot.
- Try compiling your PSCAD cases again.

Continued...

Problem 2

When attempting to compile a project using Intel Fortran and Visual Studio, the compile fails, and the following message displays:

```
ERROR: Cannot determine the location for the VS Common Tools folder.
'nmake' is not recognized as an internal or external commend,
Unable to generate a simulation executable for namsepace '*'
```

And when the Medic log file is run as per Appendix A.2, the VS Common Tools folder is properly detected. For example, the following shows that Visual Studio 2010 is properly detected:

```
Required Environment and Registry Values
  VS100COMNTOOLS (environment)
    C:\Program Files (x86)\Microsoft Visual Studio 10.0\Common7\Tools\
```

> **Note**
> By contrast, refer to Section 7.57 Problem 1, above, if the Medic does detect an error with Visual Studio common tools folder.

Cause

It is suspected that different editions of the Visual Studio version are somehow causing some confusion with regard to the required Common Tools folder. For example, for the Visual Studio 2010 version, the Professional, Premier Partner, and Express Editions are all installed.

Solution

- Uninstall all editions of Visual Studio
- Log out and log back in on your machine.
- Install VS2010 Pro (please do not install the other editions)
- Re-install Intel Fortran
- Test your setup as per this article:

  https://www.pscad.com/knowledge-base/article/566

7.58 Receiving an "Unable to generate a simulation executable for namespace '*'" Error Message

Problem

When compiling a case using the Intel Fortran compiler, the build fails, and the following Build Messages display:

> Creating EMTDC executable
>
> …
>
> Intel Parallel Studio XE 2011
>
> …Setting environment for using Microsoft Visual Studio 2008 x86 tools.
>
> Unable to generate a simulation executable for namespace '*'

Cause

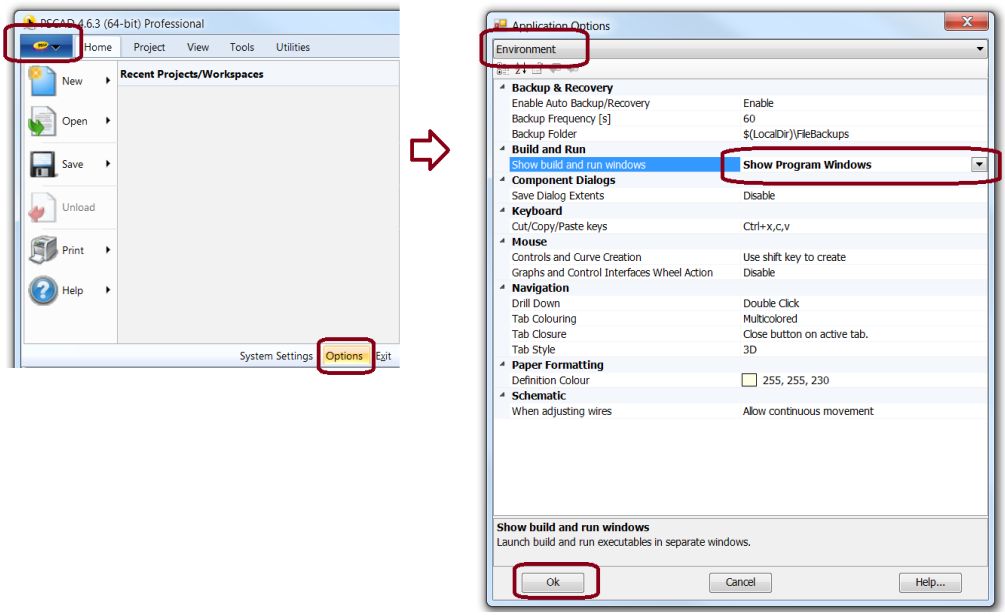
The cause is unknown at this time. It is only known that if the customer did not have Windows Administrator Privileges, then a setting in PSCAD, that of hiding the Program Windows, was generating an error.

System

Using PSCAD v4.6.2, the PSCAD case could be compiled using GFortran, but not using Intel Fortran 12 / Microsoft Visual Studio 2008.

Solution

1. Obtain Windows Administrator Privileges, then run the case using Intel Fortran 12 / VS 2008, or
2. Continue with simple Windows User Privileges, but set PSCAD to "Show Program Windows" during a run:



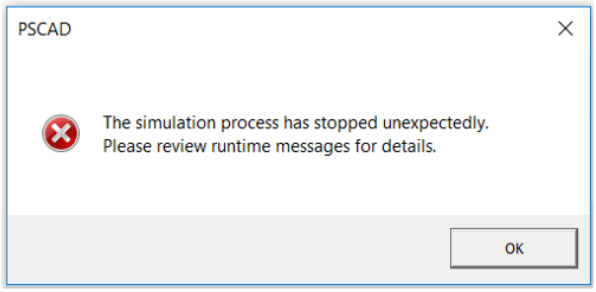
7.59 Receiving an "engOpen invalidCheck Matlab installation!'" Warning Message

Problem

When attempting to co-simulate a case with Matlab, the build fails, and the following Build warnings display:

> Error Status returned from MATLAB engine when…
> engOpen invalidCheck Matlab installation!

Also, the following message displays:



Cause

This error may be related to trying to use one version of Matlab (for example R2016a), but a different version of Matlab (for example R2017b), is registered and integrated with external software.

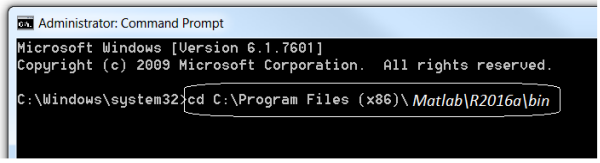
Solution (1)

Switch the registered version of Matlab to the desired version. For example, if trying to register R2016a, perform as follows:

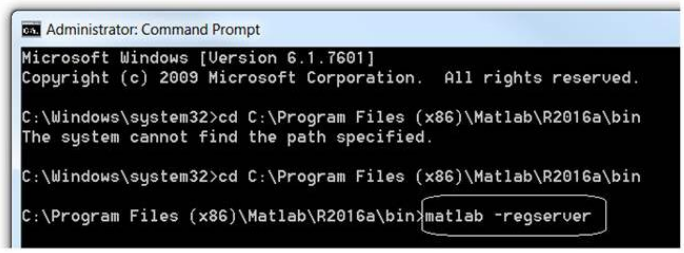
> **Note**
> Windows Administrator privileges are required.

a. Open the Windows Command prompt (from the Windows Start menu, right-click on "Command Prompt" and select "Run As Administrator"):

b. Change the directory to the "bin" folder of the desired Matlab version. For example:



c. Type in matlab –regserver:



d. This should register Matlab R2016a, and allow you to run one of the simple Matlab cases.

Solution (2)

Switch the registered version of Matlab to the desired version. For example, if trying to register R2016a, perform as follows:

Notes

1. Windows Administrator privileges are required.
2. This is applicable for Windows 10

a. Start MATLAB.

b. and typed regmatlabserver into the Command Window.

c. Register from MATLAB Command Prompt

d. To register MATLAB as an Automation server from within MATLAB, first start MATLAB, with the Run as administrator option, if necessary.

e. Because you open MATLAB to run this command, you know what MATLAB version you will register.

7.60 Receiving a “make: *** No rule to make target ‘*.mak’” Error Message

Problem

When attempting to run a PSCAD project with GFortran 4.2, the build fails, and the following Build messages display:

Creating EMTDC executable…
‘[SOME NETWORK PATH AND FILE NAME].gf42’
CMD.EXE was started with the above path as the current directory.
UNC paths are not supported. Defaulting to Windows directory.
C:\Windows>call “C:\Program Files (x86)\GFortran\4.2.1\bin\gf42vars.bat”
CMD does not support UNC paths as current directories.
Make: *.mak: No such file or directory
Make: *** No rule to make target ‘*.mak’. Stop
Unable to generate a simulation executable for namespace ‘*’

Background

The PSCAD project was originally loaded and being run from a local drive. The local drive was backed up while PSCAD was being used, which caused the project to appear to be loaded and run from the backup location (on a network drive). Compiling failed because a project may not be run from a network drive, it must be run from a local drive.

Cause

A file backup process running on the user’s machine, and is somehow changing the Working Directory, which affects PSCAD.

Solution

Modify the backup process so that it does not operate while PSCAD is running.

Perhaps the backup process could be configured to operate outside company hours.

7.61 Unable to Compile a Project

Problem

When attempting to run a PSCAD project with GFortran 4.6, the build fails.

Cause

Software that is incompatible with GFortran is installed. For example:

Cygwin, NutC, Winavr or Octave

Solution

Uninstall the offending software, or the issue might be resolved by simply changing the installation directory name of the offending software.

See Section 7.1 for further options for Cygwin.

7.62 Unable to Compile a Project

Problem

When attempting to run a PSCAD project, the build fails.

The build message is similar to the following:

Unable to find C:/Program

When the Fortran Medic utility is run as per Appendix A.2, the message as shown in Appendix A.5 Item 44 will display.

Cause

The 8.3 filenames are not enabled.

Solution (1)

- Ask your IT staff to enable 8.3 filenames, or

Solution (2)

- Ensure that all folder and file names in the path to the PSCAD project file:
  - Are 8 characters or less, and
  - Contain no spaces.

7.63 Receiving an "Unable to find Microsoft Visual C++ *** or higher" Build Error

Problem

When attempting to run one of the simple PSCAD projects using Intel Fortran, the build fails, with build messages similar to the following:

```
Will execute (1): call "C:\Program Files (x86)\Intel\Compiler\Fortran\10.1.034\IA32\bin\ifortvars.bat"  ia32
Will execute (2): nmake -f *.mak
Will execute (2): "C:\Users\Public\Documents\PSCAD\4.6\Examples\*\*.if9\*.mak.bat"
Creating EMTDC executable...
C:\Users\Public\Documents\PSCAD\4.6\Examples\*\*.if9>call "C:\Program Files (x86)\Intel\Compiler\Fortran\10.1.034\IA32\bin\ifortvars.bat"  ia32
Intel(R) Visual Fortran Compiler for applications running on IA-32, Version 10.1.034
Copyright (C) 1985-2010 Intel Corporation. All rights reserved.
Unable to find Microsoft Visual C++ 7.1 or higher.
```

> **Note**
> The above messages occurred with Intel Fortran 10.1.

Cause

This error message can occur for many reasons. One reason is that the following software was installed in the wrong order: Intel Fortran compiler (IVF) and Visual Studio (VS).

The VS software should be installed before IVF. Then when IVF is installed, IVF detects the installation of VS.

> **Note**
> This issue is likely to happen with older versions of Intel Fortran, e.g. IVF 10.1.

Solution

Re-install IFV so that it detects VS.

7.64 Unable to Build a PSCAD Project using GFortran 4.2.1 or GFortran 4.6.2

Problem

When building one of the simple PSCAD projects with GFortran 4.2.1 or GFortran 4.6.2, the build fails, with a message similar to the following:

Unable to generate a simulation executable for namespace '*'

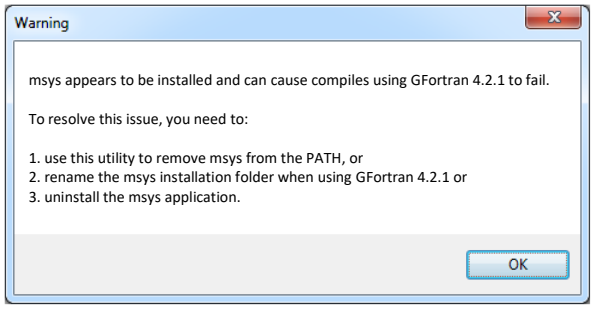
When the Medic utility is run (Section A.2), an error is displayed, similar to the following:

Detected GFortran Versions

...

GFortran 4.2.1 compiles might fail - msys appears to be installed

Right-clicking on the error displays the following warning:



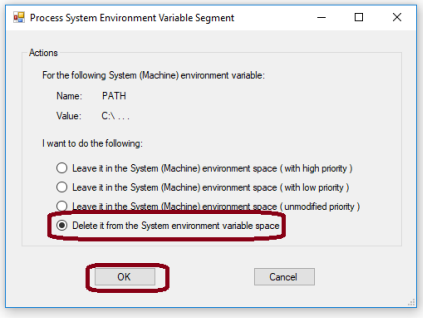
Cause

MSYS/MinGW is installed, and is causing an issue with the GFortran compiler.

Solution

There are three options for resolving this issue:

- Use the Medic utility to remove msys from the PATH: (or)
    - Run the Medic utility (Appendix A.2), and scroll to the heading “PATH (Machine) Environment Variable”. Locate and right-click on the msys path, then select to remove it from the PATH, similar to the following:



    - Scroll to the heading “PATH (User) Environment Variable”, locate and right-click on the msys path, then select to remove it from the PATH (similar to above screenshot).
- Rename the msys installation folder when using GFortran, (or)
- Uninstall the msys application.

7.65 Unable to Build a Project in the PSCAD Free Edition using GFortran 4.2.1 or GFortran 4.6.2

Problem

When building one of the examples with GFortran 4.2.1 or GFortran 4.6.2, the build fails, with a message similar to the following:

```
\\case name.gf46>call "D:\Program Files (x86)\GFortran\4.6\bin\gf46vars.bat"
Compiling 'Station.f' into object code.
gfortran.exe: error: CreateProcess: No such file or directory
make: *** [Station.o] Error 1
Unable to generate a simulation executable for namespace '*'
```

Cause

The GFortran compiler is installed on the D drive, instead of on the default drive, drive C.

On some machines, if the GFortran compiler is installed on the D drive, compiling can fail.

Solution (1)

Move GFortran to C Drive as follows:

- Completely uninstall GFortran.
- Re-install GFortran, taking care to install it to C drive.
- Log out then log back in on your machine (or restart your machine) to apply the changes.
- Test your setup by trying to run one of the simple example cases using the newly installed compiler:

  C:\Users\Public\Documents\PSCAD\<version>\Examples\tutorial\vdiv.pscx
- If there are no build errors, the issue should be resolved.

Solution (2)

Obtain Windows Administrator privileges, and always launch PSCAD with Windows Administrator privileges (right-click on the PSCAD launching link, and select the option to “Run as administrator”).

7.66 Receiving an "ERROR: Visual Studio 2013, 2015 or 2017 is not found in the system" Message

Problem

When trying to compile a PSCAD project using the following software combination:

- PSCAD v4.6.3 / Intel Fortran 18 / Visual Studio 2017 Professional Edition

The following Build Messages display:

```
ERROR: Visual Studio 2013, 2015 or 2017 is not found in the system.
'nmake' is not recognized as an internal or external command, operable program or batch file.
Unable to generate a simulation executable for namespace '*'
```

According to the Fortran Medic utility (Appendix A.3), the setup all looks good:

- The software is properly installed (PSCAD v4.6.3 / Intel Fortran 18 / Visual Studio 2017).
- PSCAD is configured to use Visual Studio 2017 (see Section 7.36).
- Intel Fortran 18 and Visual Studio 2017 are detected as being integrated.

Also, the user has logged in and out of the machine following the above installations.

Cause

There is likely an issue with integration between Intel 18 and Visual Studio 2017, even though no issue integration was detected by the Medic utility.

Solution

Re-install Intel Fortran 18. During this installation, select the option to customize the installation when prompted, to be able to:

- Ensure that the Visual Studio 2015 Shell Edition that comes bundled in the Intel 18 installation is not selected for installation.
- Ensure that Visual Studio 2017 is selected to be integrated with Intel Fortran 18.
- Launch PSCAD and test your setup by trying to compile the vdiv example case:

  C:\Users\Public\Documents\PSCAD\<version>\Examples\tutorial\vdiv.pscx

- If the above case build with no errors, your setup is ready for your own cases.

7.67 Receiving a "'C' is not recognized as an internal or external command operable program or batch file" Message

Problem

When trying to compile a PSCAD project using a GFortran compiler, the build fails, with the following messages:

'C' is not recognized as an internal or external command, operable program or batch file.
...
Linking objects and libraries into binary '*.exe'
Unable to generate a simulation executable for namespace '*'

Cause

The PATH environment has the ampersand symbol, "&", which is causing the build to fail. For example:

"C:\Program Files (x86)\<Some folder containing the & symbol>\<some installed program>\"

This issue may be detected by the Fortran Medic:

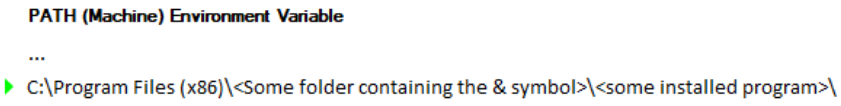
- Run the Medic as per Appendix A.2.
- Review Appendix A.5, Item 41.

Solution (1): Remove the offending path segment from your environment

> **Note**
> Obviously the above path segment is there for a reason and if you remove it, it may negatively affect the installed IntelliLink6 program which appears to need it. We recommend that you should probably consult with your IT department before removing it.

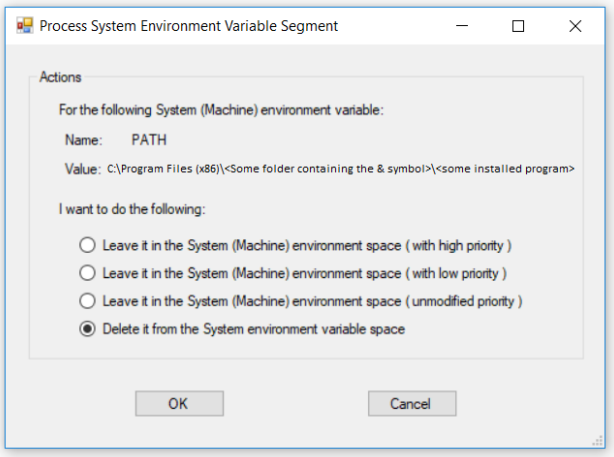
To remove the offending PATH segment, please do the following:

- Launch the FortranMedic, select Actions | Start
- When done, scroll down to the PATH (Machine) Environment Variable section, and locate the path with the "&" symbol.



Continued…

- Right-click on the green arrow corresponding to that path, and a dialog similar to the following should display:



- Select the option to “Delete it from the System environment variable space”, then select OK.
- Log out of Windows (to ensure that the PATH environment is completely updated), then log back in.
- Launch PSCAD and test your setup by trying to compile the vdiv example case:

  C:\Users\Public\Documents\PSCAD\<version>\Examples\tutorial\vdiv.pscx
- If the above case build with no errors, your setup is ready for your own cases.

Solution (2): Remove any programs that are saved to the folder with the “&” symbol

- Open the Windows Programs and Features.
- Uninstall the program(s) residing within the folder with the “&” symbol, and any other programs that were installed by it.
- Re-install the uninstalled software, but do not install it to the folder with the “&” symbol. Instead, install it to a folder which does not contain the “&” symbol in the folder name.
- Log out of Windows (to ensure that your PATH environment is completely updated), then log back in.
- Launch PSCAD and test your setup by trying to compile the vdiv example case:

  C:\Users\Public\Documents\PSCAD\<version>\Examples\tutorial\vdiv.pscx
- If the above case build with no errors, your setup is ready for your own cases.

7.68 Receiving an “Export argument '*' cannot be declared since it is already declared as a local signal” Error Message

Problem

When trying to compile a PSCAD project, the build fails, with the following messages:

```
Generating network and source code '*\*.f'
Export argument '*' cannot be declared since it is already declared as a local signal.
```

Cause

The same name has been used to define two items, which is not permitted. Each item must be uniquely identified. For further information, please refer to the following article:

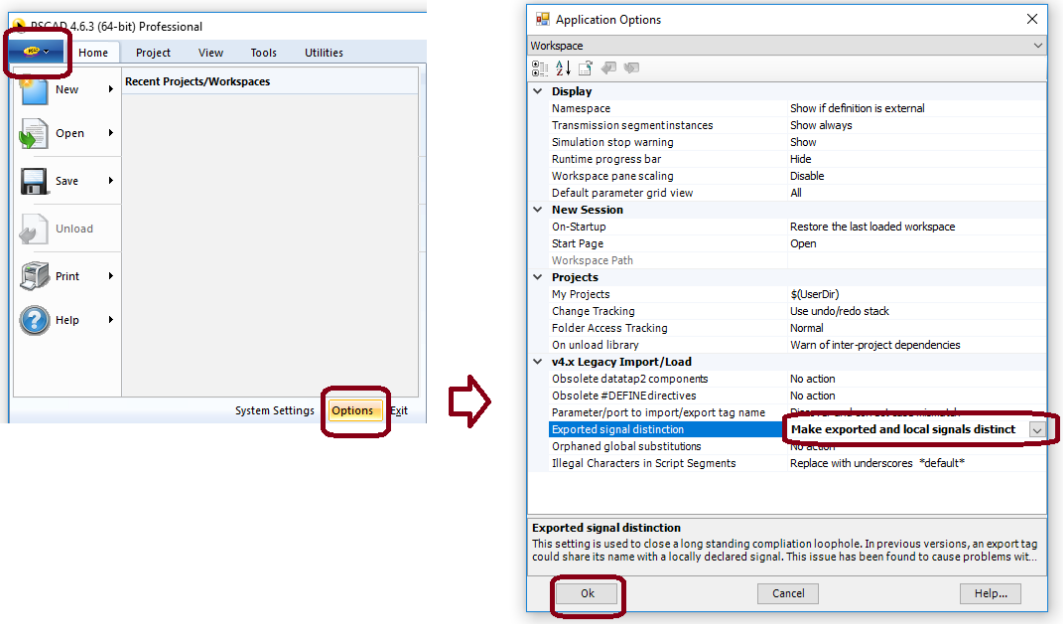
https://hvdc.ca/webhelp/ol-help.htm#Master_Library_Models/Miscellaneous/Export.htm

Solution 1 – Manually Rename one of the Items

Simply rename one of the items.

Solution 2 –Automatically Rename any Common Items

- Unload the PSCAD project.
- Select the setting “Make exported and local signals distinct”:



- Reload the PSCAD project. Any items with a common name will be renamed.
- Save the project.
- Further information on this setting may be found in this article:

  https://hvdc.ca/webhelp/PSCAD/Application_Project_and_Workspace_Options/Application/Workspace.htm

7.69 Receiving a “connect function failed with error: 10060” Error Message

Problem

When a machine is connected over VPN, a PSCAD build fails, with the following Build messages:

```
Linking objects and libraries into binary '*'
...
connect function failed with error: 10060
```

Then, if trying to run the project using Python scripting, the following message is displayed:

```
Exchange::Connect:Connection Timed Out: 10060
Failed to connect to [some IP address]:[some port]
Simulation has ended. Status code = 1
```

Also, an attempt to run the simulation from the DOS windows is successful (*.exe -startup:alone).

Background

Connection over VPN is somehow causing communication to be blocked between the PSCAD GUI (graphical user interface) and the EMTDC (solution engine). This communication is required during a PSCAD simulation, and is performed over the following port range: 30,000 to 40,000.

Even if VPN is later disabled, this communication can remain blocked (restarting the machine may restore communication).

Software

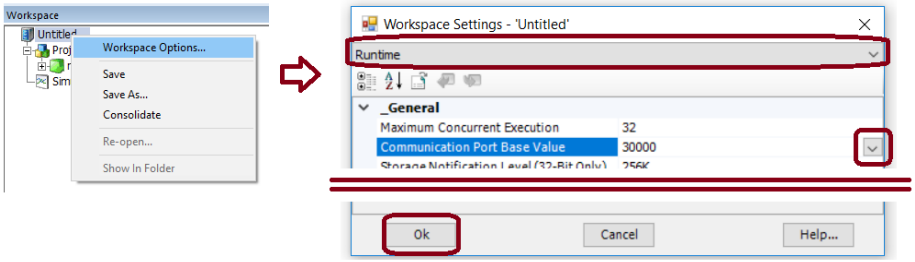
This issue has been seen with VPN software called Check Point Endpoint Connect.

Cause (1)

Protection software is causing this block, for example through the group policy or through the firewall.

Solution (1)

Configure PSCAD to use a specific port, and whitelist that port in the protection software.



Cause (2)

Other applications may be using all available ports.

Solution (2)

Determine which software is using the required ports, and turn off the software. To help determine this, you could try using a software that oversees port usage (e.g. “CurrPorts”, https://www.nirsoft.net/utils/cports.html).

Cause (3)

In older versions, if PSCAD crashed, the ports being used by PSCAD could still be held up even after restarting the machine.

Solution (3)

Either update your software (preferred), (sales@pscad.com (provide your license number)), or clear the ports manually.

7.70 Receiving Error "make: *** [<some file name>.exe] Error 1" Error Message with an ETRAN precompiled library

Problem

When attempting to run a project that is using an ETRAN precompiled library, the following Build messages display:

Linking objects and libraries into binary <some file name>.exe'
*: undefined reference to `electranix_*_'
collect2: ld returned 1 exit status
make: *** [<some file name>.exe] Error 1
Unable to generate a simulation executable for namespace '<some file name>'

Cause

The ETRAN precompiled library was not linked in PSCAD.

Solution

- Download the (free) ETRAN library from the following website
  (scroll down to "Download the E-TRAN Runtime library for PSCAD" on the webpage):

  http://www.electranix.com/E-TRAN/support_downloads.htm

- Load the E-TRAN library into PSCAD.
  For older PSCAD versions, the library must appear first (above the PSCAD project), in the workspace pane.

- Link the ETRAN precompiled library. Refer to the following article for tips:

  https://hvdc.ca/knowledge-base/read,article/478/linking-objects-and-libraries-into-pscad/v:

7.71 Receiving “Error 1”

Problem

When attempting to run a PSCAD project, Error 1 displays in the Build Messages.

Cause

An access violation has occurred. The user does not have permissions for the file path in which the project files reside, resulting in PSCAD not being to read or write to the file or directory, and the build fails.

Solution

Obtain full permissions for the file path, or

Move the project files to a location with full permissions.

7.72 Receiving “The system cannot find the path specified” Error Message on a Machine with Anaconda and PowerShell

Problem

On a machine on which Anaconda Python (aka Conda) and PowerShell are being used, when attempting to build a PSCAD project, the build fails, and messages similar to the following display in the Build Messages pane:

```
Creating EMTDC executable…
The system cannot find the path specified.
C:\...\<filename>.gf46>call "C:\Program Files (x86)\GFortran\4.6\bin\gf46vars.bat"
The system cannot find the path specified.
Compiling 'Station.f' into object code.
Unable to generate a simulation executable for namespace ‘<filename>’
```

Cause

PowerShell is being used to activate Anaconda environments. However, this affects software which run terminal commands, such as PSCAD, which for example invokes the Fortran compiler and linker. As a result, PSCAD builds fail.

Solution

Disable PowerShell from activating Anaconda environments (for example, run the command “conda init --reverse powershell”). Then, use a different method for activating Conda environments, for example:

- Launch the Anaconda Prompt from the Windows Start menu, or
- Use a different framework. For example, one customer uses Git Bash, which is enabled by running command “conda init bash”.

7.73 Receiving a "LINK : fatal error LNK1181: cannot open input file 'ws2_32.lib'" Error Message

Problem

When attempting to build a PSCAD project with the Intel Fortran compiler, the build fails, and messages similar to the following display in the Build Messages pane:

```
Linking objects and libraries into binary '*.exe'
LINK : fatal error LNK1181: cannot open input file 'ws2_32.lib'
NMAKE : fatal error U1077: '"C:\Program Files (x86)\Microsoft Visual Studio 12.0\VC\BIN\link.exe"' : return code '0x49d'
Stop.
Unable to generate a simulation executable for namespace '*'
```

Cause

A prerequisite for using the Intel Fortran compiler, 'ws2_32.lib', is missing from the environment variable. This error will occur in either of the following situations:

- If the Windows 10 SDK kit is not installed, install the Windows 10 SDK kit as per Solution 1, below, or
- If the Windows 10 SDK kit is installed, add the 'ws2_32.lib' path to the environment variable as per Solution 2, below.

Solution 1

If the Windows 10 SDK kit is not installed, install the Windows 10 SDK kit as follows:

1. Install the Windows 10 SDK kit:

   https://developer.microsoft.com/en-us/windows/downloads/windows-10-sdk
2. Log out then log back in on your machine to apply the changes.
3. Test your setup as per this article:

   https://www.pscad.com/knowledge-base/article/566
4. If the 'ws2_32.lib' error persists, try the suggestion listed in Solution 2, below.
5. If the issue persists, please feel free to obtain assistance from the MHI support desk (support@mhi.ca). Include the following in your request:
   - A description of the issue.
   - The items as listed in steps 2.c and 2.f of the following article:

     https://www.pscad.com/knowledge-base/article/164
   - Your PSCAD license number

Solution 2

If the Windows 10 SDK kit is installed, add (hardcode) the 'ws2_32.lib' path to the environment variable as follows:

Warning:

1. Users should be very careful when modifying environment variables, to ensure programs are not affected.
2. This solution will only allow running either the 32-bit or 64-bit edition of Intel Fortran.

Notes

This solution requires Windows Administrator privileges

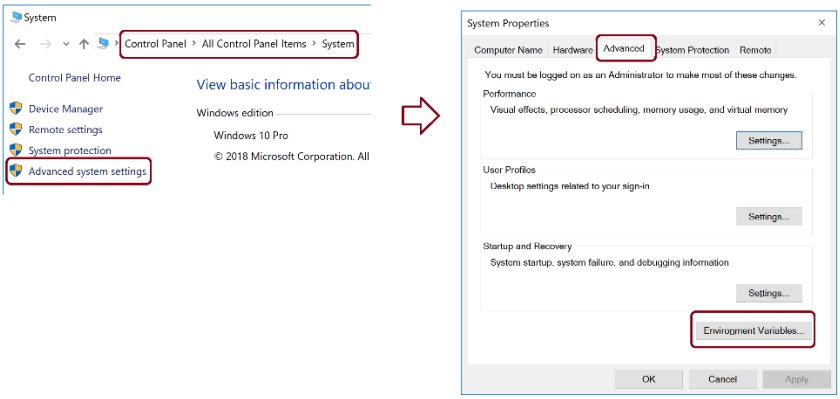
Steps:

1. Determine ws2_32.lib path for either the 32-bit or 64-bit edition on your machine:

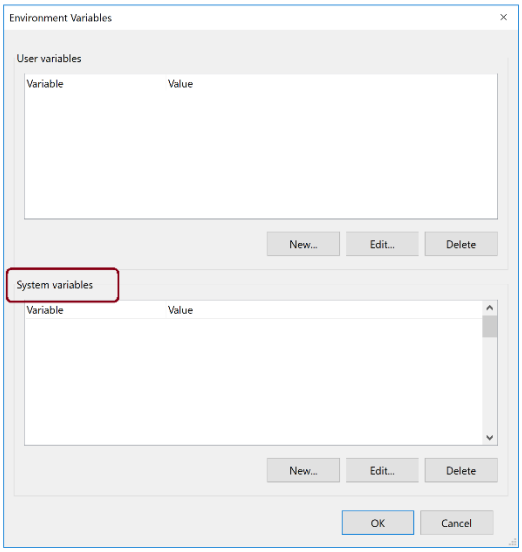
   For example, if you want to be able to use the 32-bit edition of Intel Fortran, this path would be similar to the following:
   C:\Program Files (x86)\Windows Kits\10\Lib\10.**<your version>**\um\x86

   For example, if you want to be able to use the 64-bit edition of Intel Fortran, this path would be similar to the following:
   C:\Program Files (x86)\Windows Kits\10\Lib\10.**<your version>**\um\x64

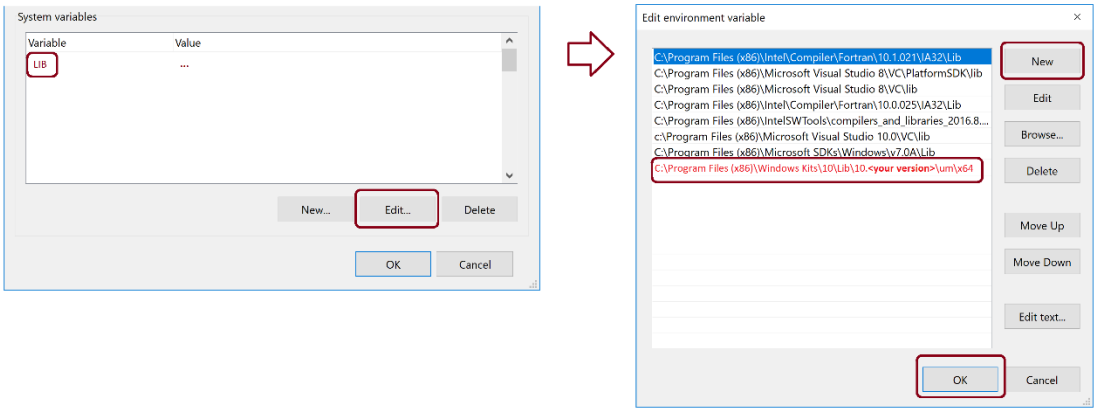
2. Display your Windows Environment Variables:



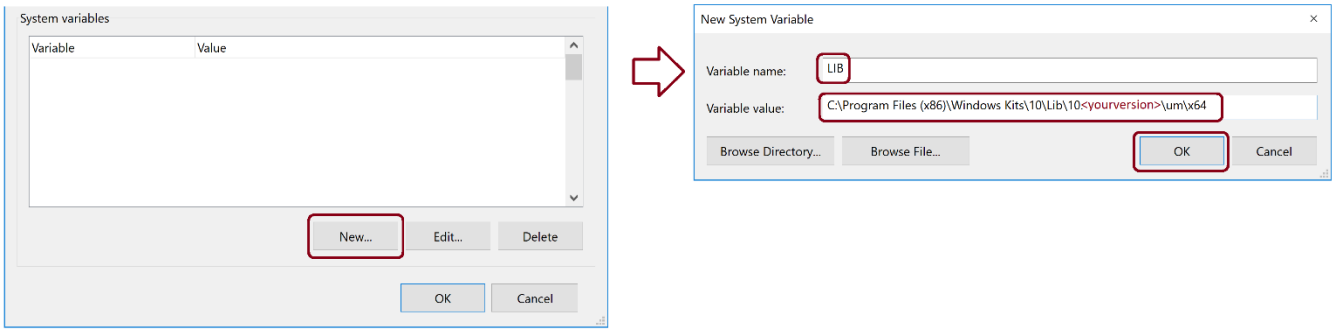
3. Add the path that you selected in Step 1 above to “System variables” as follows:



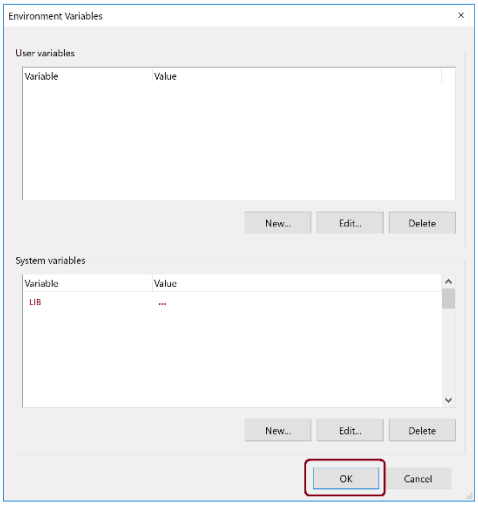
a. If you already have an existing “LIB” variable, highlight the LIB, select “Edit”, add the “New” path, and select “OK”. For example, if the 64-bit edition of Intel Fortran will be used, the path would be entered similarly to the following:



b. Or, if you do not already have an existing LIB variable, select “New”, enter LIB in the "Variable name" field, enter the path into the "Variable value" field, and select “OK”. For example, if the 64-bit edition of Intel Fortran will be used, the path would be entered similarly to the following:



4. Select “OK” as shown below to apply the changes.



5. Log out then log back in on your machine to apply the changes.

6. Test your setup as per this article:

   https://www.pscad.com/knowledge-base/article/566

7. If the issue persists, please feel free to obtain assistance from the MHI support desk (support@mhi.ca). Include the following in your request:

   - A description of the issue.
   - The items as listed in steps 2.c and 2.f of the following article:

     https://www.pscad.com/knowledge-base/article/164

   - Your PSCAD license number

[case 35465]
[Note – IF I MAKE ANY CHANGES TO THIS SOLUTION, SHOULD CHECK 7.76, SIMILAR!!! SAME CASE!!! ALMOST SAME MESSAGES, BUT VIRTUALLY SAME SOLUTION]

7.74 Receiving a “Fatal Error: Parse error when checking module version for file 'ndde.com' opened at (1)” Error Message

Problem

When attempting to build a PSCAD project with a GFortran compiler, the build fails, and messages similar to the following display in the Build Messages pane:

```
Fatal Error: Parse error when checking module version for file 'ndde.mod' opened at (*)
make: *** [Station.o] Error 1
Unable to generate a simulation executable for namespace '<some file name>'
```

Cause

Two or more GFortran versions have been installed into the same program folder. This causes the compilers to interfere with each other because they are both becoming invoked, and compiling fails.

Solution

Uninstall all versions of GFortran, then reinstall them, ensuring that each version is saved to its own folder.

Please see the following references:

- Software download and setup tips
- Tips for testing your setup

7.75 Receiving a “Reached Buffer Limit:” Error Message When Calling in a Blackboxed Module

Problem

When attempting to call a blackboxed module from an external file, the following error displays in the Build Messages pane:

FORTRAN Script Error: Reached Buffer Limit: The buffer limit for a single line has been reached.

Running the blackboxed module on its own results in no warnings or errors.

Cause

Perhaps the length of some of the signal names are too long.

Solution

Use shorter signal names.

7.76 Receiving a "cannot open file 'libucrt.lib'" Error Message

Problem

When trying to compile a PSCAD project using Intel Fortran, the simulation fails, and the following build errors display:

```
Linking objects and libraries into binary '*.exe'
LINK : fatal error LNK1104: cannot open file 'libucrt.lib'
NMAKE : fatal error U1077: '"<some path>\link.exe"' : return code '0x450'
Stop
Unable to generate a simulation executable for namespace '*'
```

Cause

The Libucrt.lib path, which is part of Windows 10 SDK, is missing from the environment variables, causing the simulation to fail.

Solution

Add (hardcode) the Libucrt.lib path into the environment variables.

Warning:

1. Users should be very careful when modifying environment variables, to ensure programs are not affected.

2. This solution will only allow running either the 32-bit or 64-bit edition of Intel Fortran.

Notes

This solution requires Windows Administrator privileges

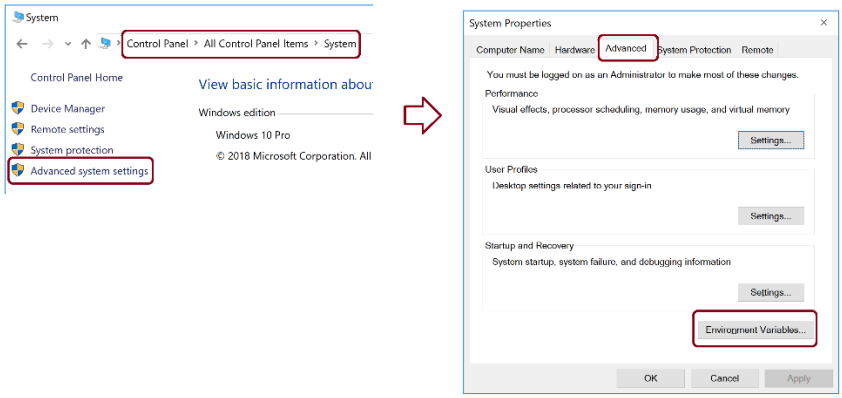
Steps:

1. Determine the Libucrt.lib path on your machine:

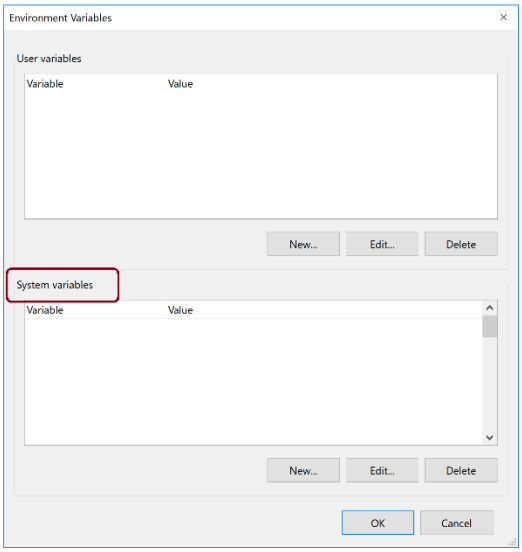
   For example, if you want to be able to use the 32-bit edition of Intel Fortran, this path would be similar to the following:
   C:\Program Files (x86)\Windows Kits\10\Lib\10.**<your version>**\ucrt\x86

   For example, if you want to be able to use the 64-bit edition of Intel Fortran, this path would be similar to the following:
   C:\Program Files (x86)\Windows Kits\10\Lib\10.**<your version>**\ucrt\x64

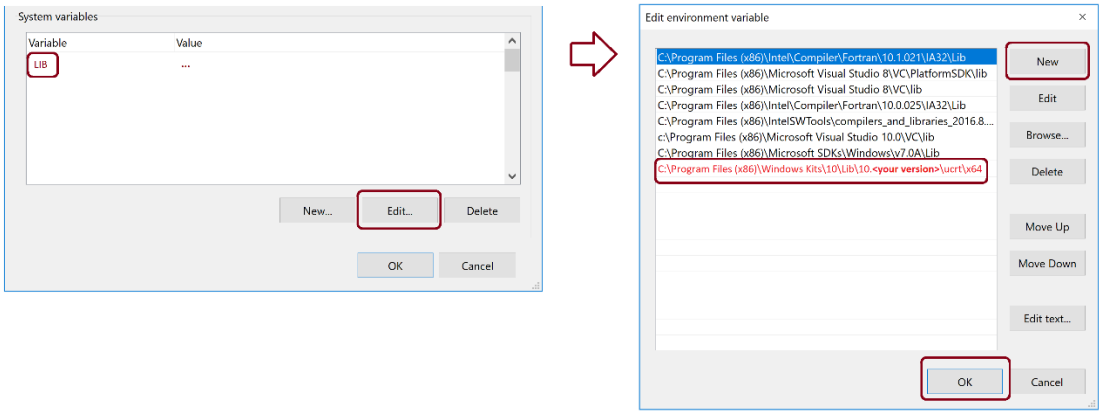
2. Display your Windows Environment Variables:



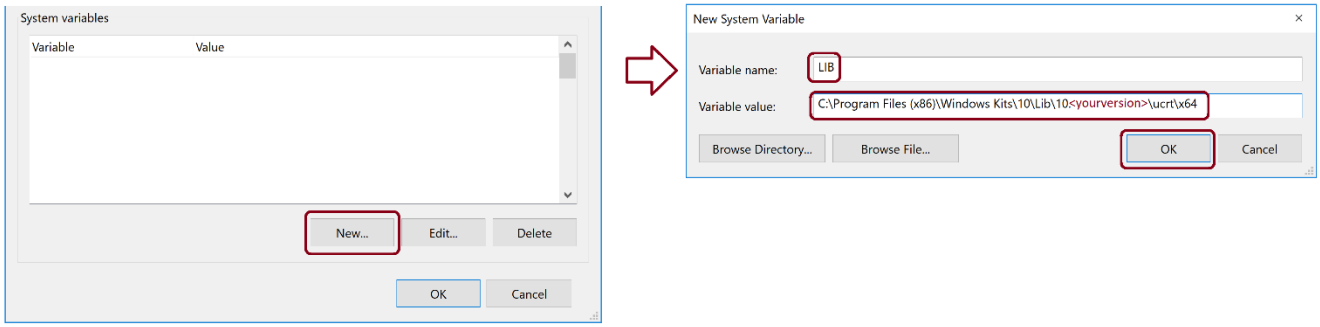
3. Add the path that you selected in Step 1 above to “System variables” as follows:



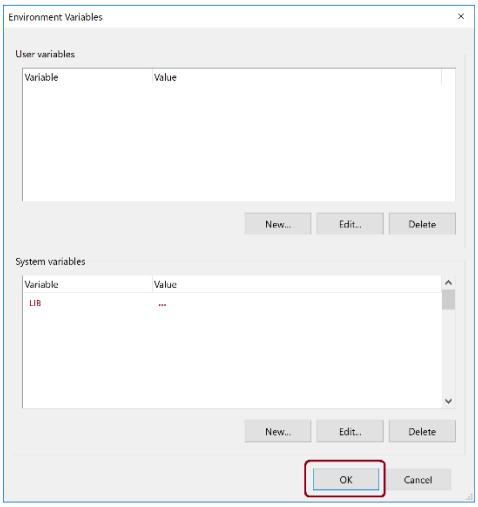
c. If you already have an existing “LIB” variable, highlight the LIB, select “Edit”, add the “New” path, and select “OK”. For example, if the 64-bit edition of Intel Fortran will be used, the path would be entered similarly to the following:



d. Or, if you do not already have an existing LIB variable, select “New”, enter LIB in the "Variable name" field, enter the path into the "Variable value" field, and select “OK”. For example, if the 64-bit edition of Intel Fortran will be used, the path would be entered similarly to the following:



4. Select “OK” as shown below to apply the changes.



5. Log out then log back in on your machine to apply the changes.
6. Test your setup as per this article:

   https://www.pscad.com/knowledge-base/article/566
7. If the issue persists, please feel free to obtain assistance from the MHI support desk (support@mhi.ca). Include the following in your request:
   - A description of the issue.
   - The items as listed in steps 2.c and 2.f of the following article:

     https://www.pscad.com/knowledge-base/article/164
   - Your PSCAD license number

[case 35465]
[Note – IF I MAKE ANY CHANGES TO THIS SOLUTION, SHOULD CHECK 7.73 Solution 2, SIMILAR!!! SAME CASE!!! ALMOST SAME MESSAGES, BUT VIRTUALLY SAME SOLUTION]

7.77 Receiving an "unresolved external" Error Message

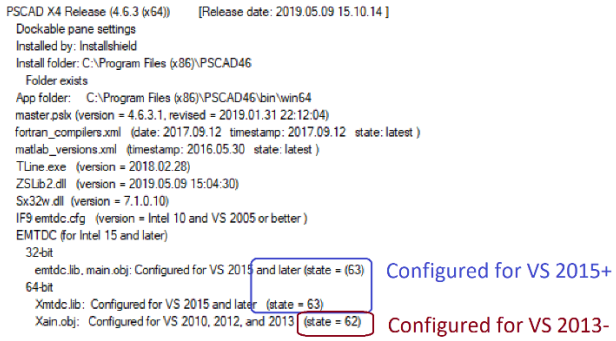
Problem

When trying to compile a PSCAD project using Intel Fortran, the simulation fails, and the following build errors display:

```
inking objects and libraries into binary '*.exe'
*.lib(*.obj) : error LNK2019: unresolved external symbol __iob_func referenced in *
*.exe : fatal error LNK1120: 1 unresolved externals
NMAKE : fatal error U1077: '"<some path>\link.exe"' : return code '0x460'
Stop.
Unable to generate a simulation executable for namespace '*'
```

Cause

The settings for using Visual Studio within the PSCAD program files contain conflicting information. This is determined by running the Medic utility as per Appendix A.2, and viewing the output for the PSCAD installation. As shown below, the "state" is set to a mixture of 62 and 63; whereas the state should be set either to 62 (PSCAD is configured to use VS 2013 and older) or 63 (PSCAD is configured to VS 2015 and newer).



Solution 1 - Fix this in your current PSCAD Installation

- Download the most recent version of the Medic utility and run it (Appendix A.2).
- Scroll down to the PSCAD v4.6.3 (x64) installation, right-click on the switch to reconfigure PSCAD (see above screenshot). If prompted whether to reconfigure PSCAD for VS 2013 and older select the option to proceed.
- Re-run the Medic utility (select "Start" from the Actions menu).
- Scroll down to the PSCAD v4.6.3 (x64) installation, and check that the states are all set "62". If they are, then select one of the following options:
    - If you will be using Visual Studio 2013 and older with your Intel Fortran compiler, leave the setting as is. Your setup is properly configured.
    - If you will be using Visual Studio 2015 and newer with your Intel Fortran compiler, right-click on the switch, and select the option to proceed when prompted whether to configure for VS 2015+.
- Launch PSCAD, and test your setup by trying to run one of the simple examples:

    https://www.pscad.com/knowledge-base/article/566
- If the issue is not resolved, please feel free ton contact the MHI support desk (support@mhi.ca). In your request, include your license number, your Medic log file (Appendix A.3), and the build messages from trying to run the test example in the above paragraph.

Solution 2 - Reinstall PSCAD

- Download and install PSCAD. Links to setup instructions are available in this article: https://www.pscad.com/knowledge-base/topic-504/v-
- After PSCAD has been installed, select one of the following options:
  - If you will be using Visual Studio 2013 or older with your Intel Fortran compiler, there is no configuration change to be made to PSCAD; PSCAD will be configured to use VS 2013 and older by default upon installation.
  - Or, if you will be using Visual Studio 2015 or newer with your Intel Fortran compiler, download the most recent version of the Medic utility and run it (Appendix A.2).right-click on the switch as shown in the above screenshot, and select the option to proceed when prompted whether to configure for VS 2015+.
- Launch PSCAD, and test your setup by trying to run one of the simple examples:

  https://www.pscad.com/knowledge-base/article/566
- If the issue is not resolved, please feel free to contact the MHI support desk (support@mhi.ca). In your request, include your license number, your Medic log file (Appendix A.3), and the build messages from trying to run the test example in the above paragraph.

7.78 Receiving the error message "Windows is not generating short (8.3) pathnames" Error Message in the Fortran Medic Utility

Problem

When building a PSCAD case, the build fails, and the Fortran Medic (Appendix A) displays the following error (see Appendix A.5, Item 15):

> Folder Info
> …
> Conflicts
> Windows is not generating short (8.3) pathnames
> 8.3 filename creation is disabled on all volumes

Also, your PSCAD project file and/or path do not conform to short 8.3 pathnames, which requires that filenames and pathnames are no longer than eight characters each nor contain any spaces:

- Example of project name and path containing eight characters or less, and containing no spaces: C:\12345678\123\123456_8.pscx

Cause

Windows is not generating short (8.3) pathnames on this machine. Consequently, compiling will fail for any PSCAD projects when the above-listed 8.3 pathnames are not conformed to.

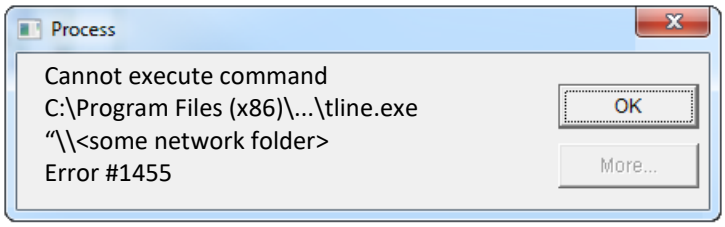
Solution

Ensure all PSCAD project files and path folder names are each a maximum of eight characters, and contain no spaces.

7.79 Receiving the error message “” Error Message When Trying to Build a Project

Problem

When building a PSCAD case, the build fails, and the following error displays:



Cause

From the above screenshot, it is shown that the file is being run a network shared folder, and there may be an issue with the network share.

Solution

Build and run the PSCAD project from a local drive:

- Copy the PSCAD case files to a local drive (for example, a new folder on your desktop).
- Load the case file residing on your desktop into PSCAD
- Build/run the project.

Please note that it is not supported to run projects that are saved to a network drive; project files should be saved to and run from a local drive.

7.80 Build Start is Delayed

Problem

When trying to build a PSCAD project, a delay occurs before building commences.

Cause

Protection software might be blocking the simulation; in particular, the compiling software or PSCAD is being blocked.

Solution

Refer to the information, "Execution Permissions" listed in the Requirements documents, to help with whitelisting PSCAD and compiler activities. The Requirements for PSCAD may be displayed from this topic.

7.81 Microsoft® Visual Studio is not Detected

Problem

When using a newer Intel Fortran compiler that can use VS 2019 or VS 2019, a build error displays indicating that VS 2017 or VS 2019 is not detected.

And, running the Fortran Medic as per Appendix A.2 results in messages similar to the following:

Visual Studio 201X Professional (###)

…

VS201XINSTALLDIR

Path not set

Cause

The Visual Studio installation did not create the environment variable VS2017INSTALLDIR or VS2019INSTALLDIR (as applicable) needed for the scripts to work. If this variable is not present, Intel will not detect Visual Studio, and PSCAD compiling will fail, unless

Note
Compiling might work if another installation of Visual Studio that is compatible with the Fortran compiler is present.

Solution

Use the Fortran Medic utility to create the environment variable as per Appendix. A.5 Item 51.

Notes

1. This solution is only applicable if using the Professional or Community Editions of Visual Studio.
2. Re-installing Visual Studio is NOT expected to create the environment variable.

Once the changes have been made, re-start the Fortran Medic (go Actions | Start), and the messages should now be changed to be similar to the following:

Visual Studio 201X Professional (###)

…

VS201XINSTALLDIR

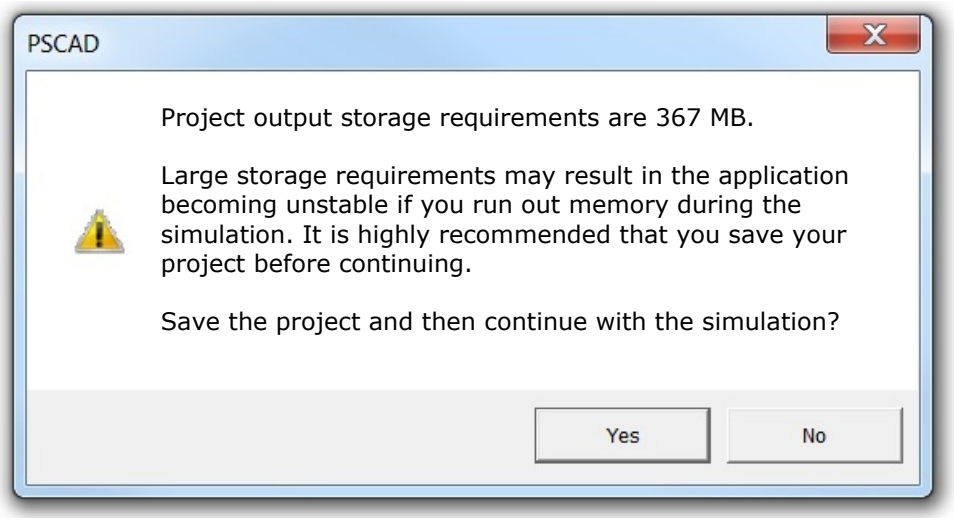
C:\Programs Files (x86)\Microsoft Visual Studio\201X\Professional
Folder exists

8. Issues when Running Cases in PSCAD

8.1 Receiving a "Project output storage requirements are xx MB" Warning

Problem

The following warning displays during a project run:



Cause

This warning is displayed to reduce the risk that any unsaved changes in the project are lost. Unsaved changes will be lost if the operating system becomes unstable and crashes due to the large simulation storage requirements.

Solution

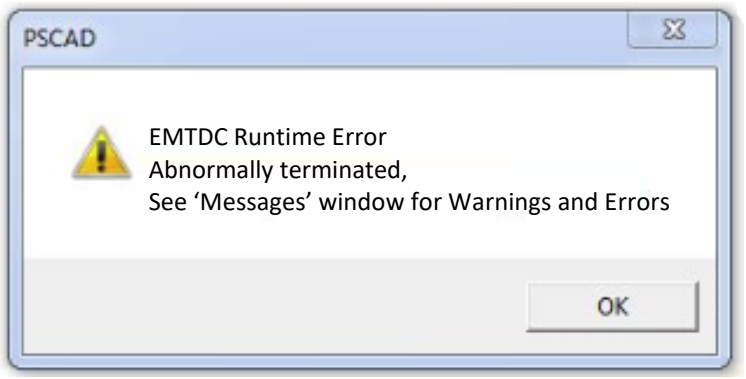
Select from the following options:

- Select "Yes" to save the project and proceed with the simulation.
- Select "No" to not save the project and proceed with the simulation.

8.2 Receiving an "EMTDC Runtime Error" Message

Problem

When a case is run, the following dialog box is displayed:



The build messages report errors similar to the following:

```
[casename]    make: *** [[casename].exe] Error 1
[casename]    Unable to generate a simulation executable for namespace '[casename]'
```

When the Fortran Medic utility is run (Appendix A.2), the conflicts as shown in Appendix A.5 Item 39 are displayed.

Cause

PSCAD is compiling cases using GFortran, and the computer is set to use non-English decimal number format. However, GFortran is affected by Regional and Language Settings.

Solution 1

Switch to Intel Fortran compiler, as this compiler is not affected by regional and language settings.

Solution 2

Continue to use GFortran, but perform the following configurations:

- Launch the Fortran Medic utility (see instructions in Appendix A.2).
- In the utility, locate the following error message:
    - For v4.2, the error will appear as follows:

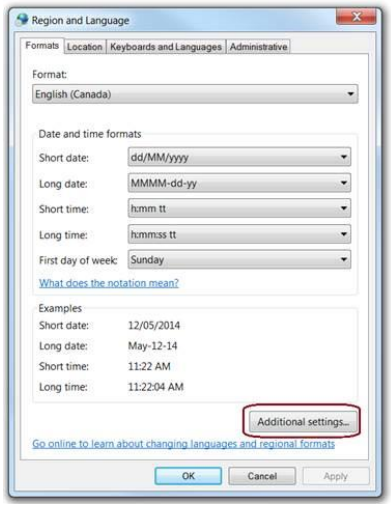
      GNU-compiled cases might not work due to non-English decimal number formats
    - For v4.3 and later, the error will appear as follows:

      GFortran-compiled cases might not work due to non-English decimal number formats
- Right-click on the error, and a "Solution" dialog box will display. Follow the instructions in the dialog box to resolve the issue. Note the following:

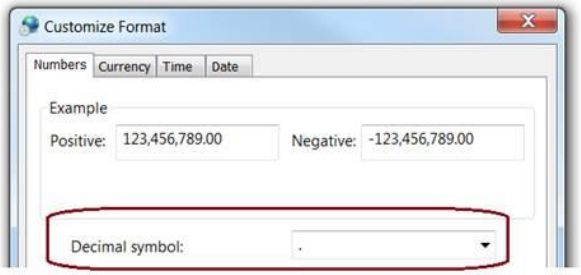
    - If prompted to modify the Control Panel, see instructions below for more details, below (for v4.2.1 and v4.3.0).

- If prompted to Modify the Control Panel (Solution 2): Perform the following:

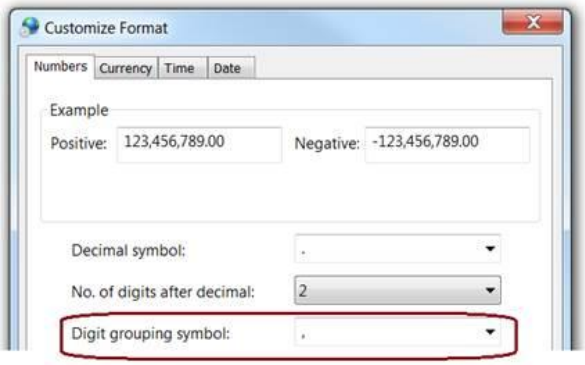
◦ In the Windows Control Panel, go to the “Region and Language” dialog, and select “Additional settings”:



◦ Change the default “Decimal symbol” from comma’s (,) to decimals(.):



◦ Change the default “Digit grouping symbol” from decimals(.) to commas(,):



◦ Press “OK” to save your settings.

8.3 Receiving an "Abnormal termination of EMTDC by *" error message

Problem

The following error message is displayed when performing a multi-run using GFortran 4.2:

Abnormal termination of EMTDC by *

Solution

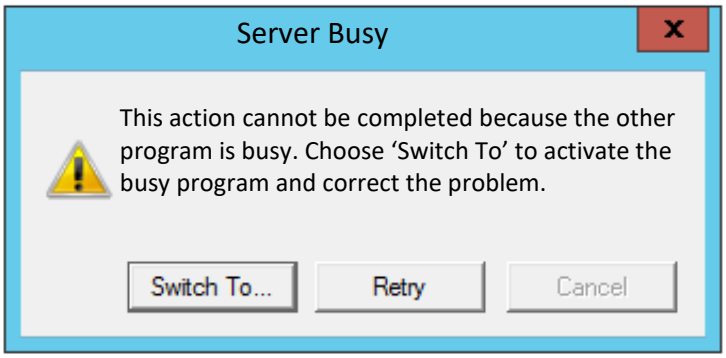
There is a bug in GFortran 4.2, in which GFortran does not release the "handles" when running a multi-run simulation, and it crashes the simulation. The following are work-arounds:

- Switch to a commercial compiler, such Intel Fortran or Compaq Fortran.
- If running PSCAD v4.6.0 and later, switch to GFortran 4.6, which comes bundled free with PSCAD (this version of GFortran is not compatible with PSCAD versions 4.5 and earlier).

8.4 Receiving a "Server Busy" Error Message

Problem

The following error message is displayed after a PSCAD case has been compiled, at the beginning of the run:



System

PSCAD v4.6.1.

Solution

Contact support@mhi.ca for the solution. Ensure to provide your PSCAD license number.

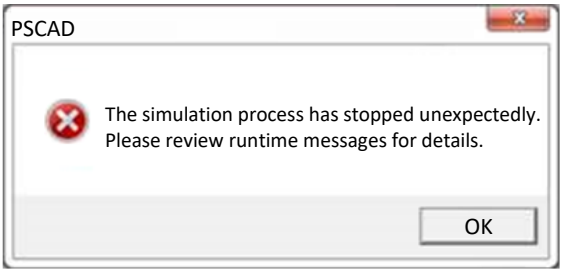
8.5 Receiving a "Result too large" Runtime Error Message

Problem

The following messages are displayed in the Runtime Messages:

Initializing Simulation Run
Executing > "*.bat"
…
*.exe: Result too large

The following PSCAD error displays:



Cause

Security software (anti-virus software or firewall) is blocking PSCAD from communicating with the simulation over the dedicated TCP ports, so the run fails. For example, McAfee has been known to cause this issue.

Applicability

This is applicable to using any compiler for compiling the PSCAD case.

Solution

PSCAD must be able to access the dedicated TCP ports: 30,000 to 30,999

Try either of the following:

- Obtain assistance from your IT staff to allow PSCAD to communicate on the above TCP ports.
- Turn off your anti-virus or firewall software.

8.6 Receiving an "Error: 10013" Runtime Error Message

Problem

The following message displays in the Runtime Messages pane:

connect function failed with error: 10013

Cause

Some aggressive security software (anti-virus or firewall) is preventing the simulation from being run.

Error code 10013 = Permission denied

Solution

Try either of the following:

- Obtain assistance from your IT staff to determine the setting that is causing this issue, and to then disable this setting.
- Turn off your anti-virus or firewall software, and retry running the case.

8.7 Receiving a "Singularity (a zero diagonal) encountered" Runtime Error Message

Problem

The following message displays in the Runtime Messages pane:

ERROR: Singularity (a zero diagonal) encountered

Cause

This error is most likely caused by 'floating' parts of circuit, i.e in which there is no ground reference in that part of circuit. For example, it may occur in the delta side of a three-phase transformer.

Solution

Ensure all circuits are grounded.

8.8 Unable to Run any PSCAD Cases - The Regional Language Issue

Problem

When running any case, the run fails, and the following Build Messages display:

EMTDC Simulation Run
Communications: Command: vdiv.exe -v4 localhost 57153
Communications: Connection established.
Non-standard Messages: Current locale = [SOME COUNTRY]

The issue you have encountered is directly due to your computer's Language and Region (also known as locale) settings, as some regions do not handle decimal numbers in the traditional English (USA) manner.

To date, some users with Dutch, German, Finnish, or Spanish language settings have experienced any or all of the following issues:

- Runtime errors or crashes due to invalid reading of the map file,
- Runtime errors due to invalid reading of the snapshot file,
- Runtime results are incorrect if using GFortran, but correct if using Intel or Compaq, and
- Runtime's Finish time is adjusted to some other value
- Undefined, zero, or negative finish time
- Runtime's Finish time is adjusted to some other value
- Unable to enter floating point numbers.

Affected Versions

This issue affects PSCAD X4 (4.4.0) only when using the GFortran compiler if the Use English (USA) number format is not selected

This issue affects PSCAD X4 (4.3.1) only when using the GFortran compiler if the Use English (USA) locale is not selected

This issue affects PSCAD X4 (4.3.0) only when using the GFortran compiler.

This issue affects PSCAD 4.2.1 only when using the GNU compiler.

Solution (1): Update your Software (preferred)

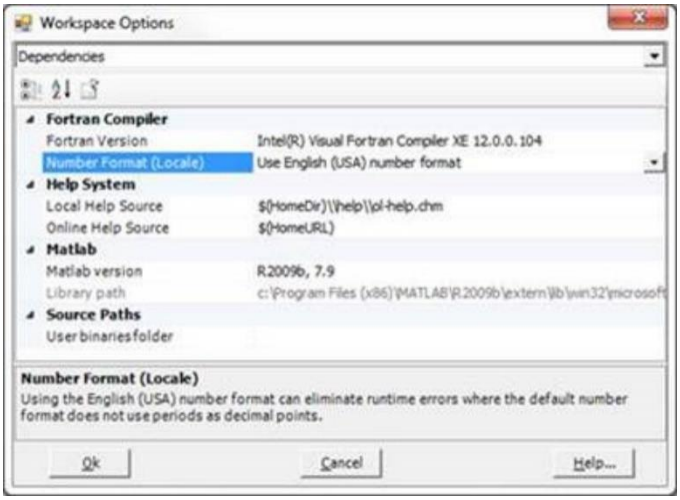
Update your software to PSCAD v4.5.5 or later.

Solution (2): For PSCAD X4 (4.4.0 and better)

The regional language issue has been addressed in PSCAD X4 (4.4.0):

- Launch PSCAD
- Then go *PSCAD Button*  *| Options | Dependencies*
- Configure the *Number Formate (Locale)* to *Use English (USA) number format*.

  This does not affect your computer's Language and Region settings at all or your other programs.
- Compile and run your cases.

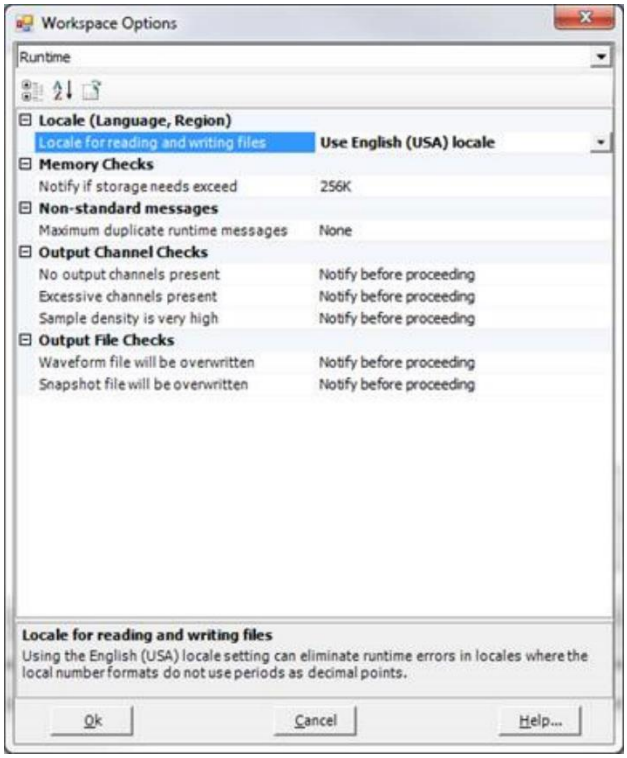


Solution (3): For PSCAD X4 (4.3.1)

The regional language issue has been addressed in PSCAD X4 (4.3.1):

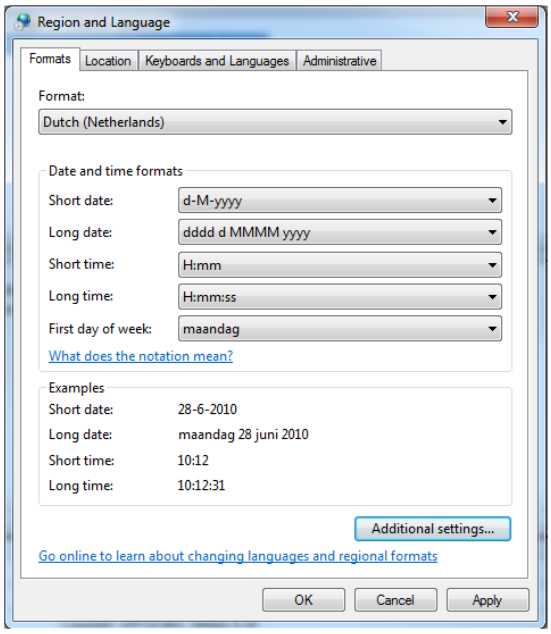
- Launch PSCAD
- Then go *Edit | Workspace Options | Runtime*
- Configure the *Locale for reading and writing files* option to *Use English (USA) locale*.

  This does not affect your computer's Language and Region settings at all or your other programs.
- Compile and run your cases.

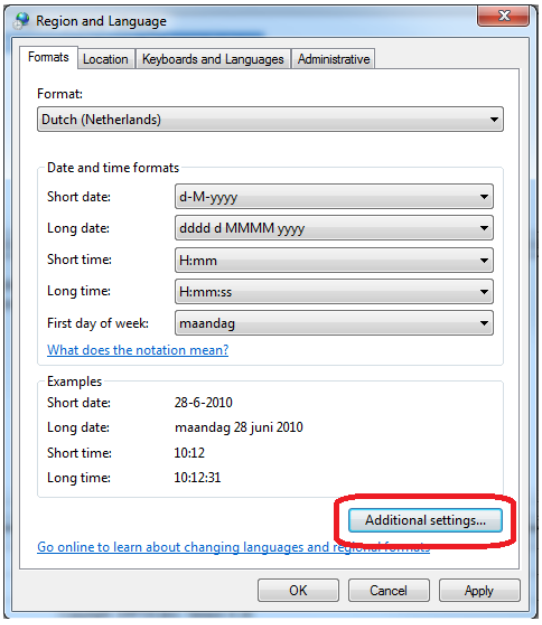


Solution (4): For PSCAD X4 (4.3.0) and PSCAD 4.2.1

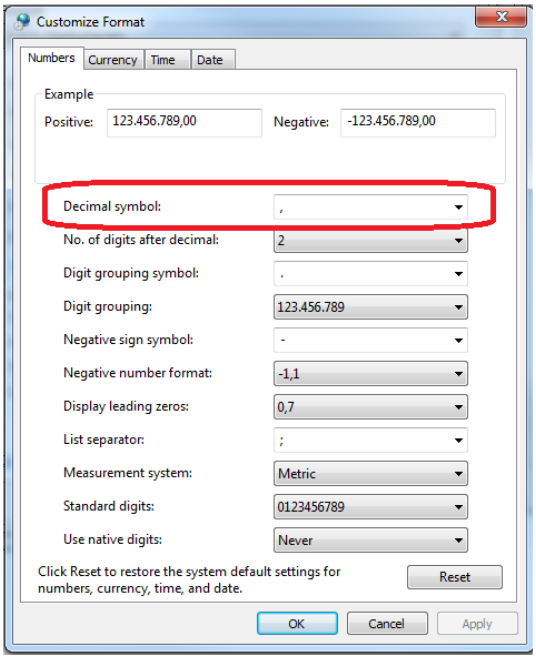
- Use an Intel Fortran compiler, as Intel is not affected by your Regional and Language Settings, **or**
- Modify your Region and Language Settings according to the steps listed below.
- Go to Region and Language settings dialog



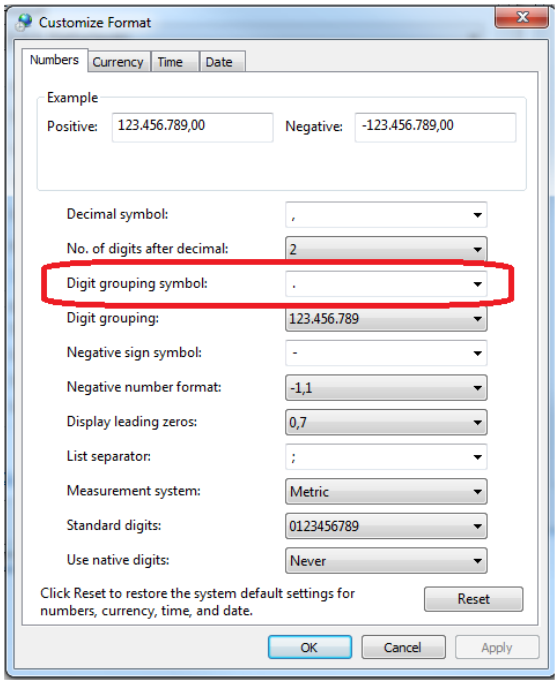
- Click on Additional Settings (Note that we tested this with Dutch settings).



- Change the default “Decimal symbol” from commas (,) to decimals(.)



- Change the default “Digit grouping symbol” from decimals(.) to commas(,)



- Save your settings and PSCAD should work fine with different language settings.

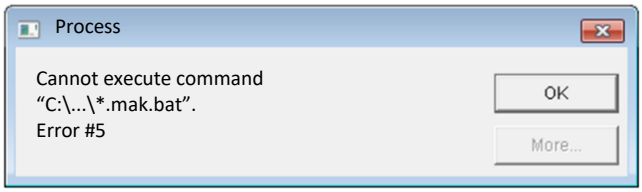
If Unable to Resolve Your PSCAD Issues

If you have any further PSCAD issues, then please download the latest FortranMedic utility and send in the generated log file as per Appendix A.2 and Appendix A.3.

8.9 Receiving a "Process…Cannot execute command…Error #5" Error Message

Problem

When trying to run a PSCAD case, an error similar to the following is displayed:



Note
See Section 7.4 for similar issue.

Problem (1)

Anti-virus software is blocking the execution of files with double extensions (e.g. *.mak.bat), which are present in PSCAD v4.6.3 Update 4 and older.

Specifically, McAfee released an update that detected and blocked files with double extensions in June 2018.

Solution (1.a)

Continue to use McAfee, but update your PSCAD software to v4.6.3 Update 5 or newer.

PSCAD v4.6.3 Update 5 and newer has replaced the double file extensions (*.mak.bat) with a single file extension (*_mak.bat). Further details on this update are listed in this article.

Solution (1.b)

Continue to use McAfee, but somehow configure the anti-virus software to not block the PSCAD files. For example, confine all PSCAD project files to a particular, local folder, (e.g. C:\PSCAD), and configure your protection software to allow all executable files to run from that folder without restriction. This would be analogous to giving a 'sandbox folder' to PSCAD which would not be monitored by the protection software.

Solution (1.c)

Switch to a different anti-virus software. Some examples of protection software that some customers switched to are Microsoft Security Essentials, Malwarebytes, and Windows Defender.

Solution (1.d)

Turn off anti-virus software.

Problem (2)

User was trying to run a project from a remote drive.

Solution (2)

Move the project to a local drive.

8.10 Receiving a "The simulation process has stopped unexpectedly. Please review runtime messages for details" Error Message

Problem

When running a PSCAD project, the simulation appears to build and to start to run. However, the run stops, and the following build message displays:

The simulation process has stopped unexpectedly. Please review runtime messages for details.

Issue (1)

In addition to the information listed above, in "Problem", when the Fortran Medic is run, the following error is listed (see Appendix A.5, Item 45 for details):

Network Information
…
Error: NotConnected (10057), A request to send or receive data was disallowed because the socket is not connected and (when sending on a datagram socket using a sendto call) no address was supplied.

Cause (1)

PSCAD and the EMTDC are not able to communicate over ports 30,000 to 40,000.

Consequently, PSCAD cannot run the sumulations.

Solution (1)

Enable communication between PSCAD and the EMTDC over ports 30,000 to 40,000.

Issue (2)

In addition to the information listed above, in "**Problem**":

- Anti-virus is turned off, so protection software is not blocking the run.
- The PSCAD project contains c-code, however, the edition of Visual Studio that is installed does not have a c-compiler.
- There are no Runtime errors; the Runtime pane simply displays normal messages, similar to the following:

  Initializing Simulation Run
  Executing > <some filepath>
  Intel(R) Parallel Studio XE ####
  Copyright…
  Intel(R) ….

Cause (2)

The project contains c-code, but cannot be compiled because the compiling software (Visual Studio) does not have a c-compiler.

Solution (1)

Install an edition of Visual Studio that has a C-compiler (see this article for further information on this).

8.11 Otaining Different Simulation Results for a Case Compiled with GFortran and Intel

Problem

You are getting different simulation results between a case compiled with a GFortran compiler and the same case compiled with the Intel compiler.

Cause

The differences in simulation results may be due to un-initialised variables used in the code of one of your custom components. Gfortran will set un-initialised variables to a value of Zero when the simulation starts. Intel will not set uninitialized variables to anything, and they will be set to randum numbers by default.

Solution

Try removing/replacing parts of your network until the results match, then you should be able to identify which component is causing you problems. Once you have the component isolated, look at its custom code and see if there are any problems.

8.12 Obtaining Error 0xC0000005 When Attempting to Run PSCAD Cases

Problem

When attempting to run a PSCAD project, the following error displays:

```
NMAKE : fatal error U1077: 'case_name' : return code '0xc0000005'
```

Cause

Anti-virus protections software is blocking the executable files from running.

Solution (1)

Whitelist the folder containing the PSCAD case.

Solution (2)

Whitelist a sandbox folder, then move any PSCAD case files into that folder, and always run any PSCAD cases from that sandbox folder.

Solution (3)

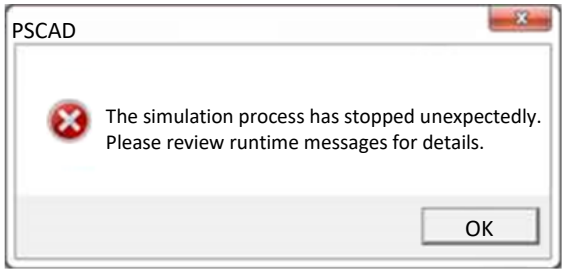
Turn off the anti-virus software when running PSCAD cases.

8.13 Cannot Run a Project When Connected over VPN

Problem

If a user's laptop is connected to its docking station, and they have a wired network connection to their docking station, and they are connected to their corporate VPN, then PSCAD is unable to run cases.

The following dialog might display:



Runtime messages might be as follows:

```
Initializing Simulation Run
Executing > "C:\...\\*.bat"
C:\...\*.gf46>call "C:\...\gf46vars.bat"
Tests.exe: No error
Current locale = C
```

The following error might display:

```
Failed to connect to IP_address:port'
```

However, user can successfully run a simulation regardless whether their laptop is connected to their corporate VPN or not, when either of the following is true:

- They connect to and use their WiFi network, or
- They unplug the laptop from their docking station and plug the network cable directly into their laptop network jack.

Solution

- If possible, connect your laptop to a WiFi network, or
- Unplug the laptop from its docking station and plug the network cable directly into the laptop's network jack, or
- Contact your IT staff for resolution.

9. Issues with MyCentre

Content moved to new manual "Resolving MyCentre Issues"
See Section 1.2

10. Issues with Diagnostic Tools

Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues"

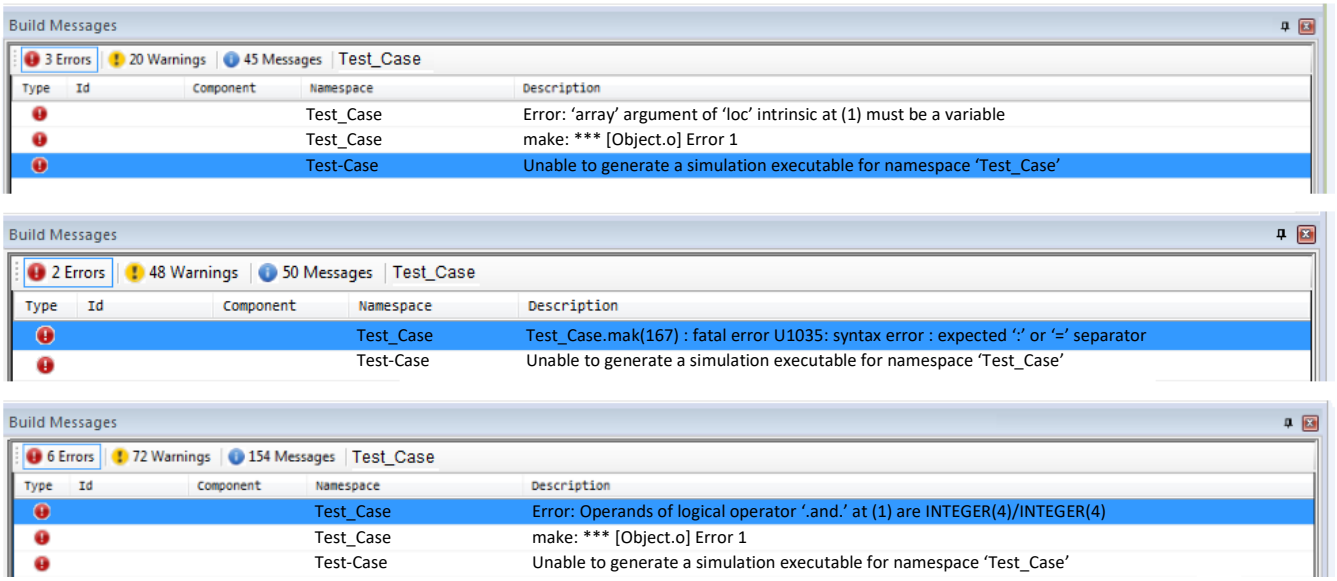
See Section 1.2

11. Resolving FORTRAN Coding Issues

11.1 Receiving an "Unable to generate a simulation executable" Error Message

Problem

When compiling a PSCAD project, build messages similar to any of the following display:



Cause

The errors are most likely a result of FORTRAN code errors in custom models.

Solution

There are three possible solutions:

- If these errors only show up in newer FORTRAN versions, then fix the programming errors, if wanting to use the models with newer FORTRAN versions.
- If these errors only show up in newer FORTRAN versions, then just compile using the FORTRAN versions in which these issues do not result in errors (e.g. earlier versions).
- Fix these errors so that they work in ALL FORTRAN versions.

Appendix A Using the Fortran Medic Utility

A.1 Overview

The Fortran Medic utility is our standard mechanism for gathering and displaying information about a user's machine that is relevant to installing, launching, licensing, and running PSCAD (see Appendix A.2).

This utility does not perform any reporting back to us, other than allowing a user to generate a text log file and forward it to our Support Desk to assist with troubleshooting (see Appendix A.3).

This utility does not modify anything on a machine without a user's explicit permission, which is obtained by clicking on the green or red arrows and confirming the recommended action (see Appendix A.4).

Many of the issues that may be detected by the utility are related to software installation and compatiblity, integration of Intel Fortran compilers with Microsoft Visual Studio, and protection software preventing PSCAD usage (see Appendix A.5 for some of the more common issues).

Appendix A.6 lists some of the functions in the Fortran Medic Utility.

A.2 Running the Fortran Medic Utility

The utility may be run as follows:

a. Download the latest "FortranMedic" from our website:

   http://updater.pscad.com/utilities/FortranMedic.zip

b. Unzip the downloaded file, save it to a local drive, and run the unzipped "FortranMedic.exe" file.

   To run the Medic without Windows Administrator privileges, select "No" when prompted by the Windows User Account Control. (The Medic will not be able to be used to make any changes to your machine)

   To run the Medic with Windows Administrator privileges, select "Yes" when prompted by the Windows User Account Control. (The Medic will be able to be used to make changes to your machine)

c. When the Medic opens, click on the "Actions" menu and select "Start". The utility will retrieve information about your computer.

A.3 Generating the Log File

The utility log file may be generated as follows:

a. Run the utility as per Appendix A.2.

b. Click on the "Actions" menu and select "Save Messages". The FortranMedic message window will display the location of the saved messages text file on your computer.

c. Send this log file to support@mhi.ca, along with any other relevant details.

A.4 Fixing Issues using the Utility

The utility may be used to fix issues as follows:

a. Run the utility as per Appendix A.2.

b. Right-click on any red arrow to display a menu to fix it.

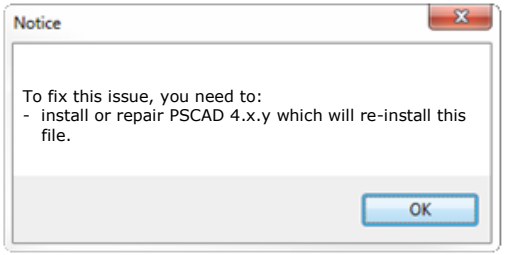
c. Right-click on any green arrow to optionally perform additional actions only if instructed by support@mhi.ca.

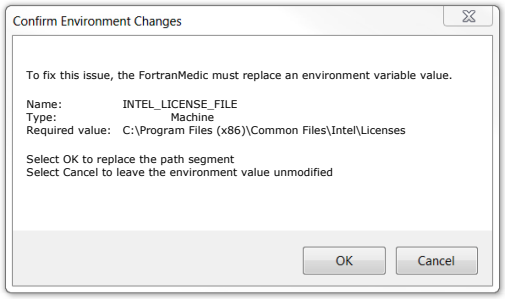
Specific errors within this utility are listed in Appendix A.5.

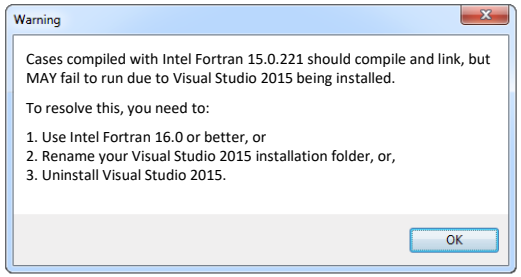
A.5 Errors Listed in the Utility

The following table lists common errors within the utility along with solutions:

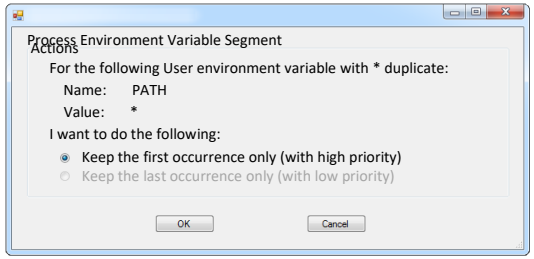
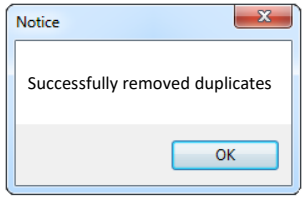
	Error	Solution
1.	**PATH (Machine) Environment Variable** Conflicts Intel MKL Segments C:\Program Files (x86)\Intel\...\bin	Right-click on the error and select the option to remove it. This environment variable will be removed.
2.	**Intel Fortran (xx.x.xxx)** Conflicts Missing expected file File: C:\Program Files (x86)\Intel…\Ifort.exe PSCAD V4 will not recognize this compiler	Right-click on the error and select the option to update the .exe file.
3.	**Installed PSCAD versions** **PSCAD X4 Release…** Workspace file: C:\...user_profile.xml File exists but appears to be empty	Right-click on the message and select the option to delete this file.
4.	**Installed PSCAD versions** **PSCAD X4 Release…** Conflicts PSCAD will not display the latest Start Page.	Right-click on the message and select the option to restore the start page.
5.	**Visual Studio [VS2005]** Installation Folders: VS folder: C:\Program Files (x86)\Microsoft Visual Studio 8\ Folder exists VC folder: Visual Studio VS2005 is not installed. Folder not specified.	You are running an older version of the Fortran Medic, and the following is occurring: • This version does not correctly detect Visual Studio Premier Partner (Shell Edition). • This version interprets this information as an incomplete installation of the full (commercial) Visual Studio. The solution is to: • Download and re-run our latest version of the Fortran Medic utility. • Check the installation of Visual Studio.
6.	**Firewalls and anti-virus programs** Firewalls and anti-virus software can affect PSCAD	See manual posted here: https://hvdc.ca/knowledge-base/read,article/217/update-client-myupdater-issues/v: Also, refer to Appendix F for information on the creation of executable files during a simulation.

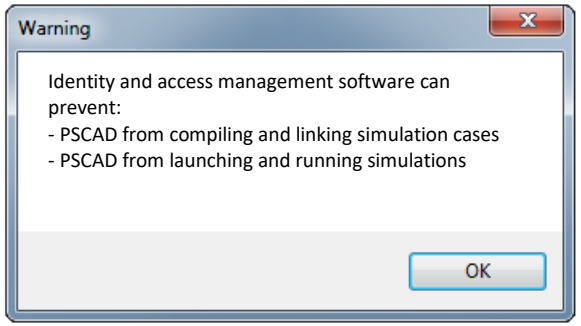
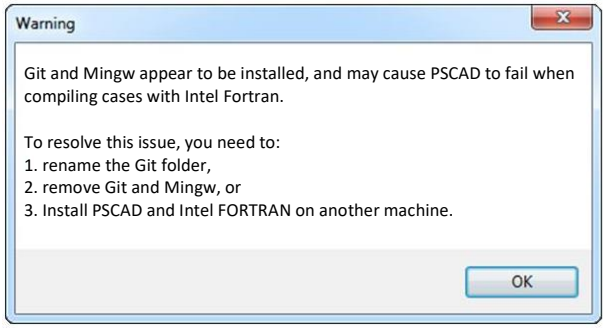
	Error	Solution
7.	**Visual Studio [Premier Partner Edition - VS 2005 Shell]** Conflicts The required VS7CommonDir registry value is not set.	This does not seem to affect performance when compiling in PSCAD X4 v4.5.4.
8.	**Intel Fortran (10.1…)** Environment [Actual]: IFORT_COMPILER10: Environment variable does not exist	Right-click on the error and select "OK" to add the path segment or "Cancel" to leave the environment value unmodified.
9.	*Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues" (See Section 1.2)*	
10.	**"Installed PSCAD versions"…** **"PSCAD X4 Release…"** ZSLib2.dll appears to be from a different version of PSCAD	Right-clicking on the message displays a dialog with the solution steps:  See Section 2.8 for additional information.

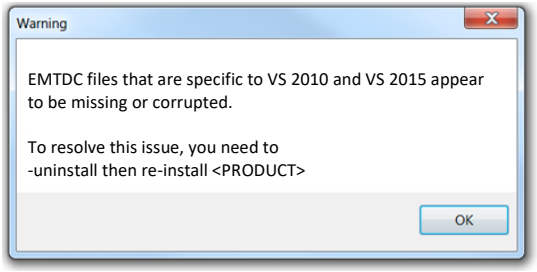
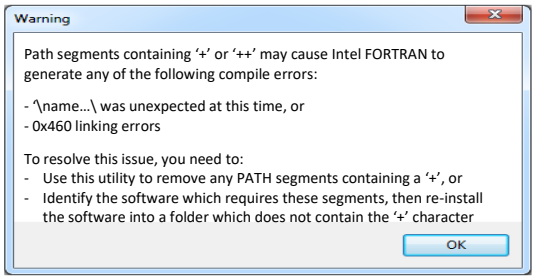
	Error	Solution
11.	**Intel Fortran (*)** Conflicts INTEL_LICENSE_FILE – Duplicate path segments may cause this compiler to fail	Right-clicking on the message displays a dialog with the solution steps:  Select "OK" to proceed with fixing the variable. See Section 7.27 for additional information.
12.	**"PATH (Machine) Environment Variable"** Conflicts The PATH variable is excessively long and you may lose your shortcuts to Notepad and other system programs.	Right-clicking on the error displays the following dialog:  Click "OK" to hide the warning. This conflict results in a problem with trying to launch some software, in this case, MyUpdater. For further details, refer to the manual posted here: https://hvdc.ca/knowledge-base/read,article/217/update-client-myupdater-issues/v:
13.	*Content moved to new manual "Certificate Licensing Issues" (see Section 1.2)*	
14.	**PATH (Machine) Environment Variable** Conflicts Segments containing a '&' character "C:\Program Files (x86)***"	Resolve as per Section 7.24, Problem #2.

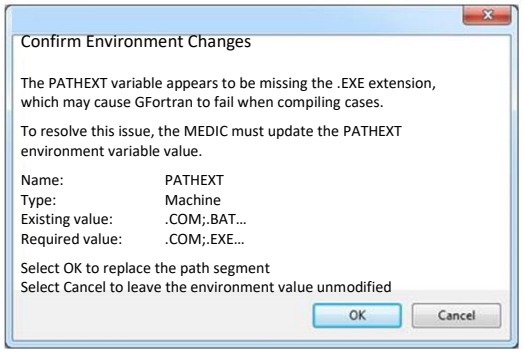
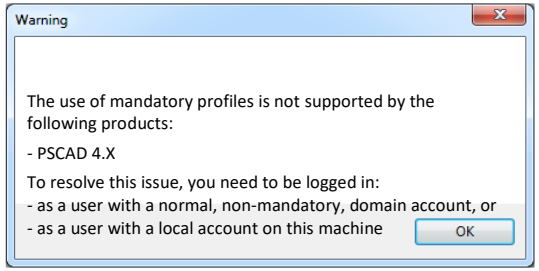
<table>
<tr><th></th><th>Error</th><th>Solution</th></tr>
<tr><td>15.</td><td>Folder Info
…
Conflicts
Windows is not generating short (8.3) pathnames
8.3 filename creation is disabled on all volumes</td><td>Resolve as per Section 7.28.</td></tr>
<tr><td>16.</td><td>Detecting Intel Compilers
Intel Fortran 15.0.0 to 15.0.221 might not work due to Visual Studio 2015 being installed.</td><td>Right-clicking on the error displays a message similar to the following:
 

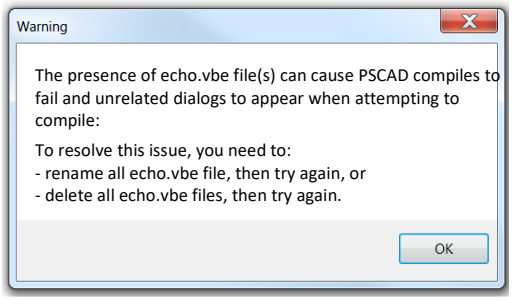
Resolve as per Section 7.30.</td></tr>
<tr><td>17</td><td>Visual Studio [*]
Required Environment and Registry Values
VS90COMNTOOLS (environment)
does not exist</td><td>Compiling might not work. The Medic tool can detect and in some cases repair this missing variable for the following Visual Studio releases:
<table>
<tr><th>Visual Studio edition / version</th><th>Can detect it</th><th>Can repair it (right-click on the error, and select the option to repair it)</th></tr>
<tr><td>2008 (commercial)</td><td>✓</td><td>✓</td></tr>
<tr><td>2008 Premier Partner (Shell Edition)</td><td>✓</td><td>X</td></tr>
<tr><td>2010 (commercial)</td><td>✓</td><td>✓</td></tr>
<tr><td>2010 Premier Partner (Shell Edition)</td><td>✓</td><td>X</td></tr>
<tr><td>2012 (commercial)</td><td>✓</td><td>X</td></tr>
<tr><td>2013 (commercial)</td><td>✓</td><td>X</td></tr>
<tr><td>2013 Premier Partner (Shell Edition)</td><td>✓</td><td>X</td></tr>
<tr><td>2015 (commercial)</td><td>✓</td><td>✓</td></tr>
</table>
See Section 7.56 for further information.</td></tr>
</table>

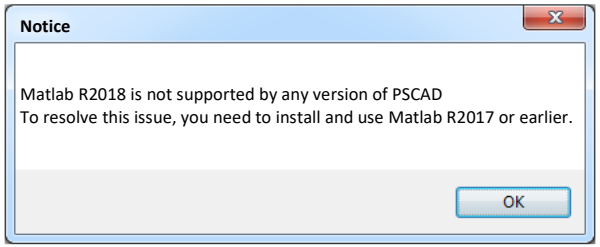
	Error	Solution
18.	**MyUpdater** Products file C:\Users\[Your Windows Username]\AppData\Local\Manitoba HVDC Research Centre\UpdateClient\Products.xml File does not exist Or Products file C:\Users\[Your Windows Username]\AppData\Local\Manitoba HVDC Research Centre\UpdateClient\Products.xml File exists File does not contain a MyCentre identity	These messages indicate that a utility has been installed on this machine (MyUpdater), but has never been logged into. Therefore the corresponding user file has not been created. This message is intended to assist users experiencing an issue with logging in to MyUpdater due to their computer or network restrictions. For further assistance on this a MyUpdater login issue, please contact our Support Desk (support@mhi.ca).
19.	*** Environment Variable** Listing: [length = * characters] … Conflicts Duplicated segments C:***	Multiple unecessary segments should be removed because they might cause performance issues on your computer. Right-click on each duplicated segment, and select “OK” when prompted:  The Fortran Medic utility will retain only the first occurrence of this segment, and delete all duplicates. The following message will display following a successful deletion of duplicates: 

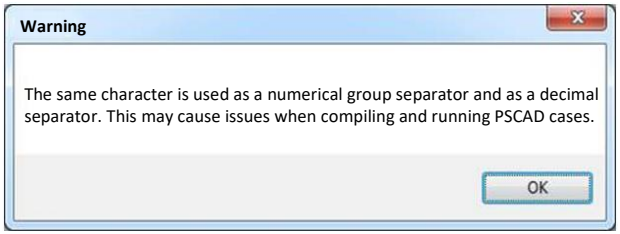
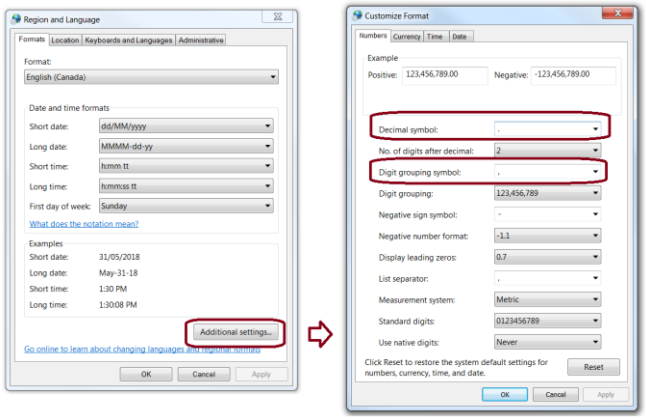
	Error	Solution
20.	Conflicts Identity and access management software can affect operation of PSCAD Double-clicking on this error displays the following dialog box: 	For example, a program called “Single Sign On Engine” is suspected of halting PSCAD simulations. See Section 7.37 for details.
21.	Git/mingw may cause PSCAD to fail when compiling with Intel FORTRAN Double-clicking on this error displays the following dialog box: 	See Section 7.38 for more information.
22.	*Content moved to new manual “Resolving PSCAD Lock-Based Licensing Issues” (See Section 1.2)*	

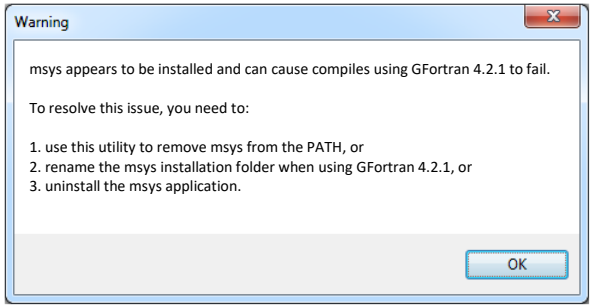
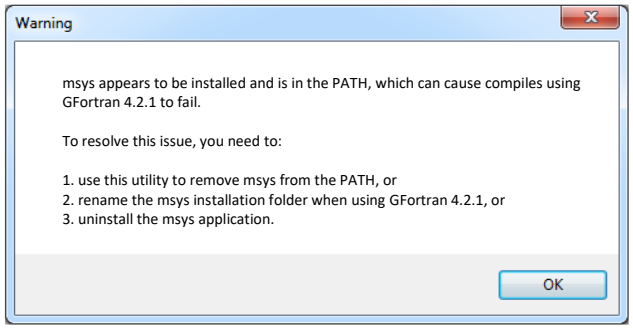
	Error	Solution
23.	**Installed PSCAD versions** Conflicts EMTDC files that are specific to VS 2010 and VS 2015 appear to be missing or corrupted. Right-clicking on this error displays the following dialog box: 	Uninstall then re-install the product, then re-start the Fortran Medic utility to ensure the error is cleared.
24.	Conflicts Segments containing a '+' in path ▸ "c:\program files (x86)\some + folder\" Right-clicking on this error displays the following dialog box: 	Refer to the Fortran Medic warning for options.

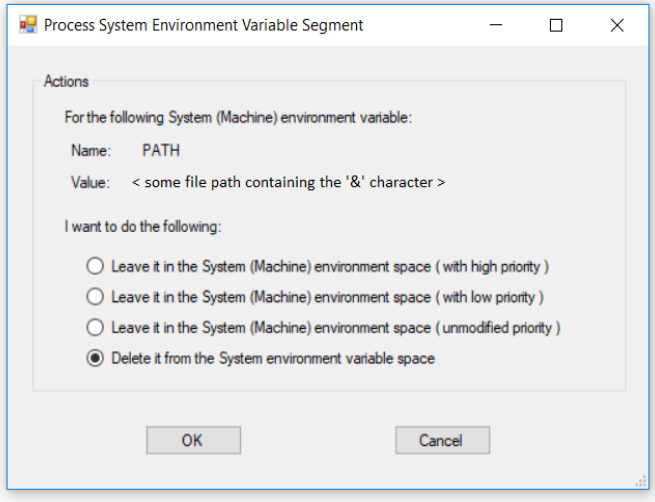
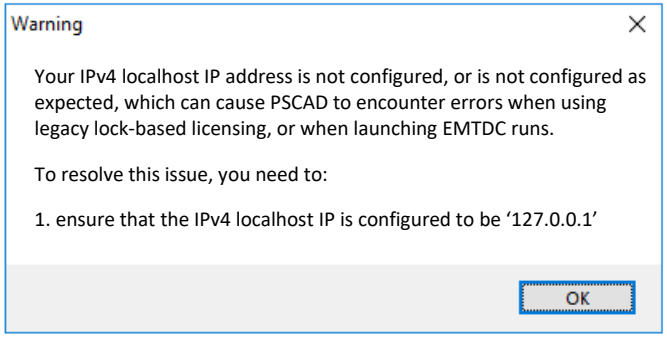
	Error	Solution
25.	*Content moved to manual "Certificate Licensing Issues" (see Section 1.2)*	
26.	*Content moved to manual "Resolving PSCAD Lock-based Licensing Issues" (see Section 1.2)*	
27.	▸ The PATHEXT environment variable is missing the .EXE extension	If you right-click on the message, the following dialog displays:  Select "OK" and the Medic will fix the environment variable. Refer to Section 7.47 for details.
28.	*Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues" (See Section 1.2)*	
29.	**User Info** Account type: Domain account Profile type: mandatory ▸ Mandatory profiles are not supported	Right-clicking on the error displays a warning, similar to the following:  Refer to Section 2.12 for more details.

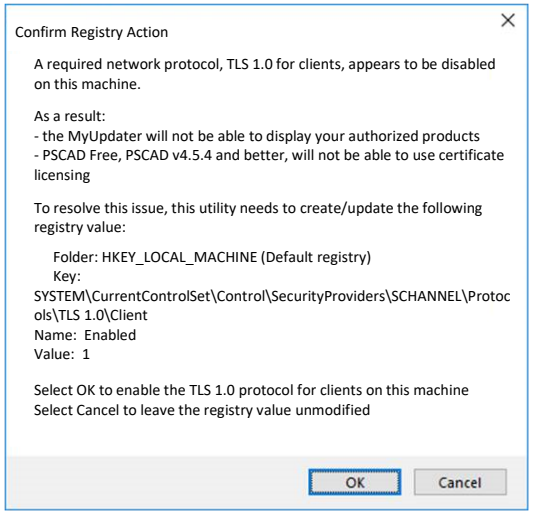
	Error	Solution
30.	**Machine Info** Echo.vbe C:\Windows\echo.vbe	Right-clicking on the error displays the following message:  Refer to Section 7.51 for more details.
31.	**Automation Library** Install folder: C:\Program Files (x86)\PSCAD\Automation Publish date: unknown	The BuildTime.txt file is not detected by the Fortran Medic. (see Appendix A.6 Item 3 to see the text when this file is detected)
32.	**PATH (Machine) Environment Variable** Conflicts The PATH environment variable is missing the \Windows\System32 path segment. The PATH environment variable is missing the \Windows path segment.	Right-click on each message and select the option to create/add that value. See Section 7.57 for more details.
33.	**Visual Studio [Premier Partner Edition – VS 2010 Shell]** Installation Folders: … Premier Partner Edition - Visual C++ folder: c:\Program Files (x86)\Microsoft Visual Studio 10.0\VC\ Visual Studio Premier Partner Edition - 2010 is not fully installed. Folder does not contain any *.exe files. … Required Environment and Registry Values VS100COMNTOOLS (environment) does not exist	These errors may be disregarded. a PSCAD project may be compiled if these errors are present. However, instructions for adding the environment to registry are listed in Section 7.56.

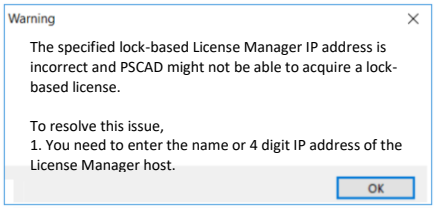
	Error	Solution
34.	**Installed PSCAD versions** … PSCAD * requires Visual C++ 2015 Redistributable (x64/x86) version 14.0.23506 or better	Visual C++ 2015 Redistributable version (x64/x86) 2015 version 14.0.23506 or better must be installed for this version of PSCAD. This software may be installed using the Fortran Medic tool as follows: • Download and launch the Medic (Appendix A.2). • From the "Install" menu, select the installation for both the x86 and x64 editions of Visual C++ 2015 Redistributable
35.	**Matlab (R2018a. 9.4 x64)** … Conflicts Matlab R2018 is not supported by any version of PSCAD	

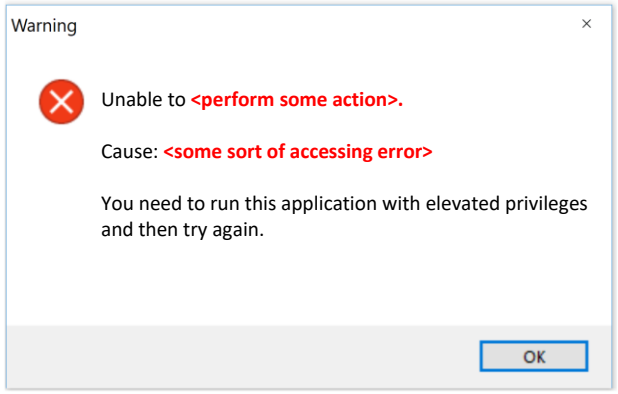
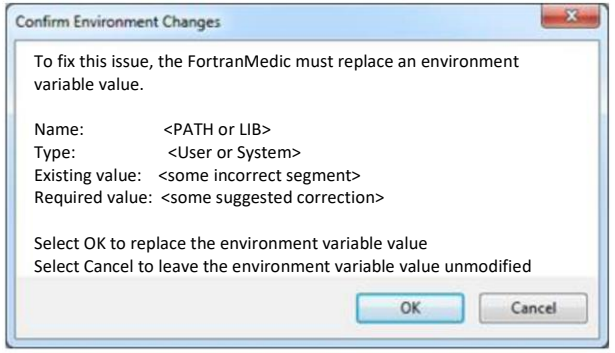
	Error	Solution
36.	Identical numerical group and decimal separators.	 Can occur if the "Decimal symbol" and "Digit grouping symbol" in the Windows Region and Language settings are set to the same character. These should not be set to the same symbol. 
37.	*Content moved to new manual "Certificate Licensing Issues" (see Section 1.2)*	
38.	*Content moved to new manual "Certificate Licensing Issues" (see Section 1.2)*	
39.	**Installed PSCAD versions** [PSCAD version 4.2] Conflicts GNUFortran-compiled cases might not work due to non-English decimal number formats [PSCAD version 4.3 or later] Conflicts GFortran-compiled cases might not work due to non-English decimal number formats	Refer to Section 8.2 for details.

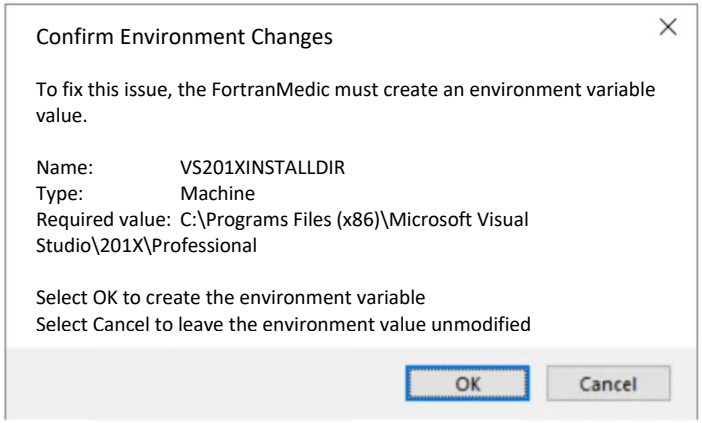
	Error	Solution
40.	**Detected GFortran Versions** GFortran *** ... **Conflicts** **GFortran 4.2.1 compiles might fail - msys appears to be installed** (Or) GFortran *** ... **Conflicts** **GFortran 4.2.1 compiles might fail - msys appears to be installed and is in the PATH** **Note** This issue is applicable also to GFortran 4.6.2	 (or)  Left-click on “OK”, then proceed with one of the suggested solutions. Refer to Section 7.64 for further details.

	Error	Solution
41.	**PATH (Machine) Environment Variable** … **Conflicts** **Segments containing a '&' character** **<some path with '&' character>**	Right-clicking on this conflicts results in a dialog similar to the following being displayed:  Refer to Section 7.67 for details.
42.	**Network Information** … **Your IPv4 localhost IP address is not configured as expected**	Right-clicking on this conflict results in a dialog similar to the following. Configure your machine as directed.  PSCAD expects that a machine's localhost is configured to the standard "127.0.0.1". If a machine's localhost is not configured, or configured to something else, then PSCAD might have issues when using lock-based licensing, and when launching/connecting to EMTDC instances.

	Error	Solution
43.	**Licensing** … **▸This computer is not configured to support a required protocol.**	Existing certificate licensing and the MyUpdater utility require that one of the following protocols be enabled: SSL 3.0, TLS 1.0 If the Medic detects that neither protocols are enable, this will be reported as shown in the left column. Right-clicking on the error displays the screen as shown below. Select OK to enable the TLS 1.0 protocol for clients on this machine, to allow MyUpdater activities and certificate licensing: 
44.	**Folder Info** **…** **Conflicts** **Windows is not generating short (8.3) pathnames** **8.3 filename creating is disabled on all volumes**	Windows short 8.3 pathnames has been disabled, which can affect the compiling of PSCAD projects. Please refer to suggested solutions in Section 7.62.
45.	**Network Information** … Local socket connections: **Error: NotConnected (10057), A request to send or receive data was disallowed because the socket is not connected and (when sending on a datagram socket using a sendto call) no address was supplied**	It has been detected that access to ports 30,000 to 40,000 is not allowed, therefore PSCAD cannot communicate with the EMTDC in order to build and run the simulation. Communication over ports 30,000 to 40,000 must be enabled to run PSCAD simulations. Additional system requirements are as listed in this article. See Section 8.10 for further information on this issue.

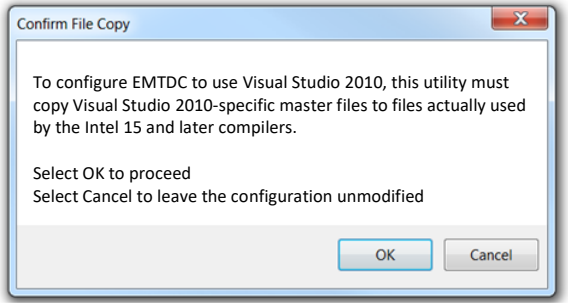
	Error	Solution
46.	**Network Information** … External socket connections **Error: HostNotFound (11001), No such host is known**	It has been detected that PSCAD cannot communicate with external machines. This situation is only an issue if a user is trying to use high performance computing features in PSCAD, wherein multiple simulations are being launched in parallel over multiple machines. This situation is not an issue if a user is simply running PSCAD simulations (singularly or in parallel) on his own machine.
47.	**Visual Studio 2017 Professional** **…** Private registry: privateregistry.bin **Folder does not exist**	This error does not seem to affect compiling PSCAD projects using Intel Fortran and Visual Studio 2017. This error may be disregarded.
48.	**PSCAD settings** … LmgrHost (32-bit): LmgrHost (64-bit): XX.XXX.XX.XX 2053 **An invalid IP address was specified**	If an invalid IP address was entered for PSCAD lock-based licensing, in the Licensing tab of the System Settings dialog, then an error will display in the Medic results as shown in the left column. Right-clicking on this error will display an error dialog similar to the following:  Examples of invalid IP addresses include entering fewer or more than four digits (e.g. x.x.x.x), or when any of the four digits exceed 255.
49.	**Network Information** **…** Conflicts Your default proxy may be hijacked. Your system proxy may be hijacked.	The detected conflict means that all outbound traffic on this machine is actually being routed to port 9000, or some other port, where most likely some service is listening on. That appears, from prior research, to be a favorite port used by malicious software to sniff all traffic for, say, log in credentials, credit card info, etc, before possibly forwarding it on to the intended website or recipient, or maybe not, or maybe even streaming a copy of everything off to some remote hacker server for later analysis. It is recommended that the customer's IT Department look at the issue and resolve it if required.

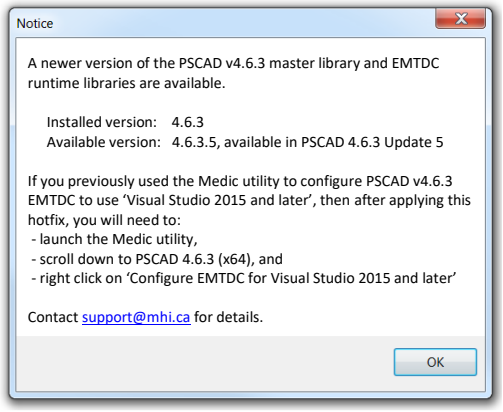
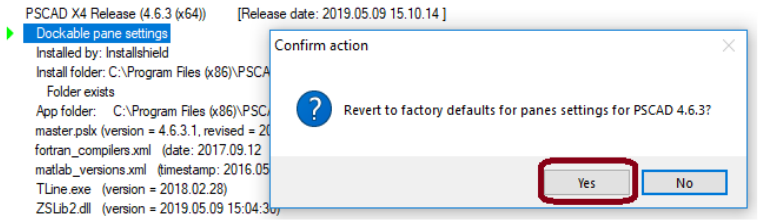
50.	User attempts to perform any changes to their machine using the Fortran Medic utility, and receives an error message instead.	A dialog similar to the following displays:  The Fortran Medic utility was not launched with Windows Administrator privileges, therefore the Medic cannot make any changes to this machine. To allow the Medic to make changes to this machine, close the Medic, and relaunch the Medic with Windows Administrator privileges as specified in Appendix. A.2.
51	Conflicts Segments with unbalanced or mid-string quotes ‣<some segment with unbalanced or mid-string quotes>	For the system or user PATH and LIB environment variables, the Medic can correct path segments containing any of the following: • Contain an odd # of quotes (i.e. have unbalanced quotes), or • Contain any internal quotes (i.e. mid-string quotes), The Medic can now repair them by removing all quotes in/bracketing that segment. Simply right-click on the error as listed in the left column, and select OK in the "Confirm Environment Changes" dialog: 

	Visual Studio 201X Professional (###) … VS201XINSTALLDIR Path not set	Right-clicking on the message in the left column displays the following dialog:  For further infromation, refer to Section 7.81.

A.6 Functions Listed in the Medic

The following table lists the functions within the Medic:

	Function	Action
1.	**PSCAD Configuration…** **Installed PSCAD versions…** PSCAD X4 Release (4.6.X) Configure EMTDC for Visual Studio 2015 and later	Refer to Section 7.36 for instructions on this configuration. **Note** Applicable to PSCAD v4.6.1 and later.
2.	**PSCAD Configuration…** **Installed PSCAD versions…** PSCAD X4 Release (4.6.X) Configure EMTDC for Visual Studio 2010, 2013 Under a PSCAD installation heading (v4.6.1 and newer)	**Note** Applicable to PSCAD v4.6.1 and newer. Right-click on this function, and the following dialog will display:  If you proceed with this action, PSCAD will be configured to use Visual Studio 2010, 2012, or 2013. **Warnings** 1. If you proceed, PSCAD will no longer be configured to use Visual Studio 2015 and newer. 2. It is not recommended to use Visual Studio 2012 as it can interfere with other Visual Studio installations even after the uninstallation of Visual Studio 2012. **Notes** 1. This function does not install Visual Studio 2010, 2012, or 2013, it merely configures PSCAD to be able to use these versions. 2. Configuration with Visual Studio 2015 and newer may be restored as per Section 7.36.
3.	**Automation Library** Install folder: C:\Program Files (x86)\PSCAD\Automation Publish date: YR.MO.DAY…	The BuildTime.txt file has been properly detected by the Fortran Medic. (see Appendix A.5 Item 31 to see the text when this file is not detected)

	Function	Action
4.	*Content moved to new manual "Certificate Licensing Issues" (see Section 1.2)*	
5.	PSCAD X4 Release (4.6.3…) … Conflicts ▸ A newer version of the master library and EMTDC is available	If the Medic detects that PSCAD v4.6.3 is installed, the medic will determine whether the most recent update within v4.6.3 is installed. If not, a conflict will display similar to the text in the left column. Right-clicking on the conflict will display the notice shown below. Information on the updates to v4.6.3 are as listed on this webpage. Updates may be requested from sales@pscad.com. Ensure to provide your PSCAD license number in your request. 
6.	PSCAD X4 Release (4.6.3…) ▸ Dockable pane settings	The dockable pane default settings may be restored using the Medic. Close all instances of PSCAD, then in the Medic, right-click on "Docable pane settings", and select the option to restore the pane settings. 

Appendix B Using the Get Info Utility

Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues"

See Section 1.2

Appendix C Lock-based Legacy Licensing - License Manager Requirements

Content moved to new manual "Resolving PSCAD Lock-Based Licensing Issues"

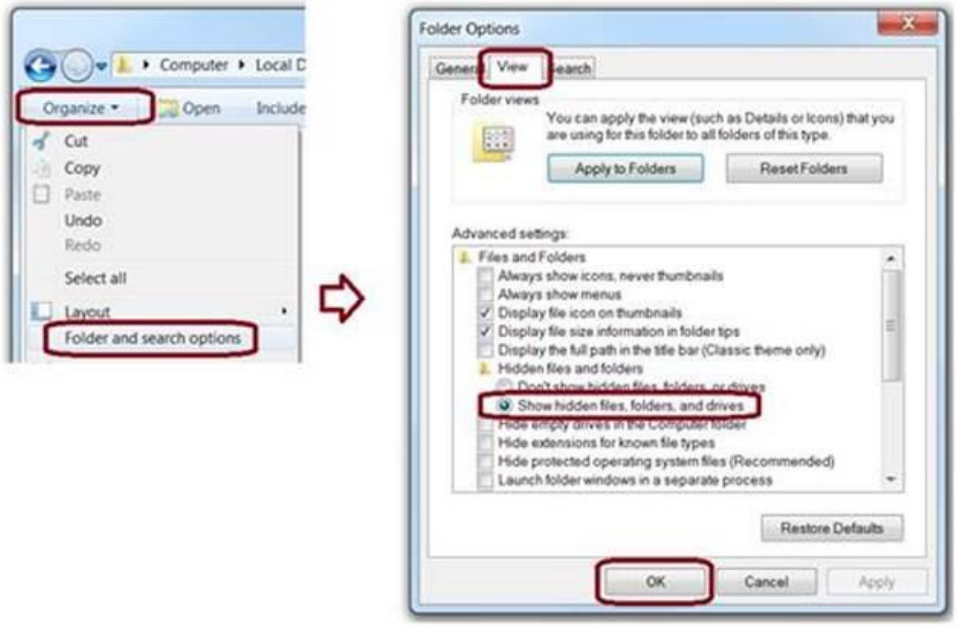
See Section 1.2

Appendix D Certificate Licensing – Requirements

Content moved to new manual "Resolving Certificate Licensing Issues"
See Section 1.2

Appendix E How to display a Hidden Folder

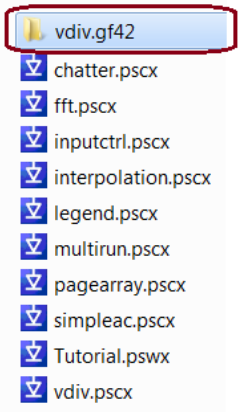
In many issues in this document, there is reference to a folder that may be hidden, called appdata. If this folder is hidden, it may be displayed as shown:



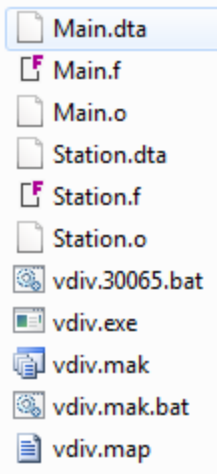
Appendix F File Creation During a Simulation

The following is a description of the files that are created and used when a PSCAD case is compiled:

- When a PSCAD case is compiled, for example C:\Users\Public\Documents\PSCAD\4.6.2\Examples\tutorial\vdiv.pscx
- PSCAD will create a working folder called <case name>.<compilerID>.
- For GFortran 4.2, this would be vdiv.gf42, as shown in the screenshot below. For GFortran 4.6.2, the folder is called .gf46, for Intel, it is called if9, if12, or if15.



- The contents of the PSCAD-created working folder are as follows:



- The vdiv.mak.bat file is created by PSCAD, and has the following format: <*case name*>mak.bat, and is executed by PSCAD, which launches the GFortran 4.2 compiler for the entire case.
- The GFortran make/nmake file creates one batch file in the %USERPROFILE%\AppData\Local\Temp folder for each .f file it needs to compile.

- For GFortran 4.2 and GFortran 4.6.2, two or more batch files are created:
  - One is called BATCH*nnnn*.bat, and
  - One or more other randomly named batch files NOT ending with .bat
- For Intel, we observed two to three randomly named files, NOT ending in .bat
- Once the .f file has been compiled, all of the two to three batch files are deleted, and the next .f file is created.
- The contents of the BATCH*nnnn*.bat file created by GFortran typically looked like the following:

```
@echo off

gfortran.exe -c -ffree-form -fdefault-real-8 -I"C:\PROGRA~2\PS70C7~1\emtdc\gf42\inc" -
I"C:\PROGRA~2\PS70C7~1\emtdc\gf42\windows"  -Wconversion  CtrlSystem.f
```

- We were not able to capture any of the Intel batch files as they existed only very momentarily

As mentioned earlier, these temporary batch files are created by the third party GFortran or Intel Fortran compilers.

Continued…

The files and folders currently used by PSCAD are as follows:

Item	Type	Install time Permissions	Runtime Permissions	Path	State (Released versions)	State (Future)
Licensing log file	File		Write	C:\ProgramData\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt C:\Users\USERID\AppData\Local\Manitoba HVDC Research Centre\LicenseManager\PscadLmgr.txt	Fixed	May possibly be reconfigured
Certificate license (if used)	Folder		Read/Write	For PSCAD v4.5 and v4.6: C:\Users\Public\Documents\Manitoba HVDC Research Centre\Licensing\Licenses For PSCAD v5: C:\Users**<user_id>**\AppData\Local\Manitoba Hydro International\Licensing\Licenses	Fixed	Fixed
Legacy License (if self-licensing or License Manager is used)	File	Write	Read/Write	C:\Users\Public\Documents\Manitoba HVDC Research Centre\LicenseManager\lmgr-hvdc	Fixed	Fixed
User settings	File	Write	Read/Write	C:\Users\USERID\AppData\Local\Manitoba HVDC Research Centre\PSCAD\	Fixed	May possibly be reconfigured
Examples	Folder	Write	Read/Write	C:\Users\Public\Documents\PSCAD\4.6.2	Moveable by user	Moveable by the user

Appendix G Testing Connectivity for Certificate Licensing

Content moved to new manual "Resolving Certificate Licensing Issues"
See Section 1.2

DOCUMENT TRACKING

Rev.	Description	Date
0	Initial	17/Jul/2014
1	Added new sections (7.3, 8.1, 8.2, A.5); Updated sections (6.1, 10.1, A.4); Updated the document title	29/Jul/2014
2	Changed section numbers (Section 6); Added new sections (1.2, 2.1, 2.2, 8.3-8.11, 10.3); Updated sections (1.1, 6.1-6.3, 8.1)	08/Aug/2014
3	Updated sections (3.1, 6.4, 8.1, 8.2, 8.5, 8.8, 8.9, 10.1, 10.2, A.2); Deleted Section 8.4; Renumbered Sections 8.5-8.11 to 8.4-8.10; Added new sections (7.4, 7.5, 8.11-8.17, 10.4-10.6)	17/Oct/2014
4	Modified title; Reorganized content per following changes; Moved all installation instructions to manual “Resolving Installation Issues”; Moved MyCentre launching issue to manual “Update Client – Common Issues”; Section 4.2 moved from later section; New Section 8 (moved issues from Section 7) Added new Sections 4.1, 6.6, 7.15, 7.16, 10.1, 10.2	28/Nov/2014
5	Deleted Section 6.6 (duplicated); Added new Sections 6.6, 6.7, 7.17, 7.18, 7.19	28/Jan/2015
6	Update to Section 3.4, Section 9.2, Appendix A.5; Added new Section 7.20	20/Apr/2015
7	Added new Sections 2.4, 4.3, 9.5	20/May/2015
8	Update to Sections 4.1, 7.8; Added New Sections 2.5, 2.6, 7.21, 7.22	28/May 2015
9	Update to Section 7.8	07/Jul/2015
10	Added new Section 2.7, populated Section 3.3	28/Jul/2015
11	Added new Sections 5.1, 3.5, and 6.8	15/Sep/2015
12	Added new Cause #3 within Section 5.1; Added new Section 3.6; Update to Appendix A.5	22/Sep/2015
13	Added new Cause #3 within Section 7.3, Added new Section 3.7 and new Appendix C	24/Sep/2015
14	Added new Sections 3.8, 7.23, 7.24, and 7.25; Update to Section 3.6	02/Nov/2015
15	Added new Section 2.8; Update to Sections 2.7, 3.4 and Appendix C	13/Nov/2015

Rev.	Description	Date
16	Added new Item 10 to Appendix A.5; Added new Section 7.26 and new Appendix D; Update to Sections 2.8 and 3.8	12/Jan/2016
17	Split contents of Section 3 into existing Section 3, New Section 4 and New Section 5; Renumbered sections (Sections 4 to 8 became Sections 6 to 10); Added new Cause 4 to NEW Section 5.1; Update to Section 3.2 Item 5; Deleted NEW Section 10; Correction to Appendix A.5 Item 7	25/Jan/2016
18	Update to Section 1.1; New Section 7.27; New Item 11 in Appendix A.5	04/Feb/2016
19	Added new Sections 2.9 and 7.28; Update to Section 4.2, Solution 1; Added new Problem 2 to Section 7.24; Added new Items 12 to 15 in Appendix A.5	29/Feb/2016
20	Added new Cause 2 to Section 3.1; Added new Problem 2 to Section 7.15; Added new Section 7.29;	31/Mar/2016
21	Added new Sections 3.9, 3.10, 5.2, 7.30 and 7.31	5/Apr/2016
22	Added new Sections 6.9 and 7.32; Added new Items 16 and 17 in Appendix A.5; Update to Section 3.9, Section 7.24 (Problem 2), Section 9.3, and Appendix A.5 (Item 10)	27/Apr/2016
23	Moved Cause/Solution 2 from Section 3.8 to new Section 4.4; Moved Cause/Solution2 from Section 3.1 to new Section 3.11; Added new Sections 6.10 and 7.33; Update to Section 7.24; Added new Item 18 to Appendix A.5	18/May/2016
24	Added new Sections 2.10, 4.5, 6.11, 7.34, 7.35; Added new Item 19 in Appendix A.5	23/June/2016
25	Added new Section 7.36; Updated Section 3.9	04/Nov/2016
26	Update to Sections 5.1 and 7.36; Added new Sections 4.6, 6.12, 7.37 Added new Item 20 in Appendix A.5	30/Nov/2016
27	Added new Sections 6.13, 6.14, 7.38, 9.6 Added new Problem 3 to Section 7.8 Added new items 21 and 22 to Appendix A.5 Update to Sections 2.5, 3.9, 7.35	14/Dec/2016

Rev.	Description	Date
28	Update to Section 7.8 and Appendix D Added Section 4.7, 4.8, Problem #2 to Section 7.27, Section 8.4, and Appendix E, Added Item 23 to Appendix A.5, Added new Appendix A.6, Items 1 and 2 Updated weblinks	26/Jan/2017
29	Added Sections 6.15, 6.16, 7.39, 8.5, and Appendix A.5 Item 24 Added Solution 3 to Section 7.26, Added Solution 1 to Section 7.36 Updated Appendix A.6 Items 1 and 2	14/Mar/2017
30	Added Sections 3.12, 4.9, 8.6 Update to Sections 6.12, 6.16 and 7.5	03/Apr/2017
31	Update to Sections 6.12 and 7.26 Added Sections 2.11, 3.13, 3.14, 7.40, 10.1, 11.1, and 11.2	31/May/2017
32	Added Sections 4.10 and 4.11 Added Item 25 in Appendix A.5 Added a second solution to Solution 1 in Section 7.10	04/Jul/2017
33	Added Sections 3.15, 6.17, 6.18, 7.41, and 7.42 Added Solution 2 to Problem 1, Section 7.27 Corrections to Solution 2 of Section 7.36	03/Aug/2017
34	Update to Section 7.2 Solution 1 Update to Section 7.30 Update to Section 7.36 “Cause” Added Sections 7.43 and 7.44	31/Aug/2017
35	Added Sections 5.3, 7.45 and 8.7, and Appendix F Added Solution 3 to Section 6.6 Added Solution 26 to Appendix A.5 Update to Sections 7.39, 8.5 and 8.6	07/Sep/2017
36	Added Sections 3.16, 3.17, 7.46, 7.47, 7.48, and 7.49 Update to Section 11.1 and Appendix F Added Items 27 and 28 in Appendix A.5	21/Sep/2017
37	Update to Section 6.9 (Solution 3), Section 9.5, and Appendix A.1 Deleted Section 10 “Issues with Intel Fortran Licensing” (material moved to manual “Setting up the Intel Fortran Compiler”)	08/Nov/2017
38	Added Sections 2.12, 4.12 (Cause 1), and 4.13 Added Item 29, Appendix A.5 Moved Solution 3 (Section 5.1) to Solution 2 (Section 4.12) Renumbered Solution 4 to Solution 3 in Section 5.1	28/Nov/2017
39	Added Sections 3.18, 4.14, 7.50, 7.51 and 7.52 Added Item 30, Appendix A.5 Update to Section 3.17	03/Jan/2018
40	Added Sections 7.53 and 11.1 Update to Section 3.18	04/Jan/2018
41	Added Appendix A.5 Item 31, and Appendix A.6 Item 3 Added Sections 2.13, 6.19, 7.54, and 7.55	07/Feb/2018

Rev.	Description	Date
42	Added Sections 2.14, 5.4 and 7.56 Update to Appendix A.5, Item 17	02/Mar/2018
43	Added Sections 2.15, 7.57, 7.58 and 8.8 Added Items 32, 33 and 34 to Appendix A.5	11/May/2018
44	Update for New Brand Guidelines	16/May/2018
45	Added new Items 35 and 36 to Appendix A.5; Corrections to Section 7.56	28/May/2018
46	Added Section 6.20 Moved copyright	04/Jul/2018
47	Added Section 8.9 Update to Sections 4.13 and 7.54	04/Jul/2018
48	Update to Section 7.59	11/Jul/2018
49	Update to Branding	19/Jul/2018
50	Added Sections 4.15 and 6.21; Added Item 37 to Appendix A.5 and Item 4 to Appendix A.6; Update to Section 8.5	23/Aug/2018
51	Added Sections 2.16, 4.16, and 7.60	02/Oct/2018
52	Added Item 38 to Appendix A.5; Update to Appendix D	25/Nov/2018
53	Added Sections 2.17 and 4.17	27/Nov/2018
54	Combined Sections 2.16 and 2.17; Deleted Section 2.17; Update to titles to Sections 4.16 and 4.17	12/Dec/2018
55	Added Sections 3.19, 7.61, and 7.62	30/Jan/2019
56	Added Section 7.63 and Appendix G; Added Item 5 to Appendix A.6; Added Item 39 to Appendix A.5; Update to Section 8.2	01/Mar/2019
57	Added Section 7.64, and added Item 40 to Appendix A.5	29/Mar/2019
58	Removed materials related to issues with Certificate Licensing and MyCentre, including Section 4, Section 9, Appendix D, Appendix G, Appendix A.5 Items 13, 25, 37 and 38, and Appendix A.6 Item 4. These materials were moved to new documents "*Resolving Certificate Licensing Issues*" and "*Resolving MyCentre Issues*"; Update to Appendix F; Moved content from Section 5.2 to new Section 3.20	01/May/2019
59	Added Sections 7.65, 7.66 and 7.67d; Added Item 41 to Appendix A.5; Deleted Section 5.2; Renumbered other sections accordingly	09/May/2019

Rev.	Description	Date
60	Update to Section 7.31 title; Added Section 7.68; Correction to Section 7.67	22/May/2019
61	Added Section 10.3; Added Solution 2 to Section 7.65; Added Item 42 to Appendix A.6	06/Jun/2019
62	Update to Item 42, Appendix A.5; Added Sections 3.20, 7.69 and 7.70	20/Aug/2019
63	Update to Sections 7.28, 7.31 and 7.70; Added Sections 7.71 and 10.4; Added Item 43 to Appendix A.5; Added Item 6 to Appendix A.6	28/Oct/2019
64	Added Sections 7.72 and 7.73; Update to Sections 3.3, 7.36, and 7.62; Added Cause 3 to Section 3.10; Added Item 44 to Appendix A.5	25/Nov/2019
65	Added Sections 6.22, 6.23, 6.24, 7.74 and 8.10; Added Items 45 and 46 to Appendix A.5; Update to Appendix A.5 Item 33; Added referencing between Sections 7.4 and 8.9	15/Apr/2020
66	Update to Sections 7.57, 7.69, and 7.74; Problem 2 added to Section 7.57; Added Section 7.75	30/Apr/2020
67	Update to Sections 7.36, and 7.73 Solution 1 Added Sections 3.22, 7.73 Solution 2, 7.76, 7.77, and 7.78 Added Item 47 to Appendix A.5	17/Jun/2020
68	Removed materials related to Lock-Based Licensing issues, including Section 3, Section 10, Appendix A.5 (items 9, 22, and 28), Appendix B, and Appendix C. These materials were moved to new document "*Resolving PSCAD Lock-Based Licensing Issues*." Section 1.1 was updated correspondingly	18/Aug/2020
69	Added Problem 4 to Solution 7.9	16/Sep/2020
70	New Section 8.11 New Item 48 to Appendix A.5 Deleted the contents of Section 5 "Issues when Licensing PSCAD – Legacy Lockless", and Appendix A.5 Item 26 were transferred to document, "*Resolving PSCAD Lock-Based Licensing Issues*"	13/Oct/2020
71	Update to title	14/Oct/2020
72	New Sections 7.79, 7.80, 8.12 and 8.13; New Items 49, 50 and 51 added to Appendix A.5; New Issue 2 added to Section 8.10 Update to Section 8.9 and to Appendix A.2	28/Oct/2020

73	Update to Item 5 in Appendix A.6; Update to Sections 7.1, 7.4, 7.5, 7.61, and 8.13; New Sections 2.17 and 7.81; New Item 51 in Appendix A.5;	30/Nov/2020